Elements Reference

ANSYS Release 11.0

January 2007

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

List of Figures

List of Tables

Chapter 1: About This Manual

Welcome to the *Elements Reference*. This manual contains a complete library of detailed ANSYS element descriptions, arranged in order by element number. It is the definitive reference for element documentation.

The *Elements Reference* is intended to give you information on individual ANSYS elements. See *Chapter 2: General Element Features* for detailed information on the features included in element documentation. See *Chapter 3: Element Characteristics* for lists of element characteristics.

This manual is not intended to be your primary source of procedural information - look in the appropriate analysis guides for introductory and procedural guidelines.

The following *Elements Reference* topics are available:

1.1. Conventions Used in this Manual
1.2. ANSYS Product Capabilities

1.1. Conventions Used in this Manual

ANSYS manuals use the following conventions to help you identify various types of information:

Type style or text	Indicates
BOLD	Uppercase, bold text indicates command names (such as **K**, **DDELE**) or elements (LINK1).
Bold>Bold	Bold text in mixed case indicates a GUI menu path, which is a series of menu picks used to access a command from the GUI. One or more angle brackets (>) separate menu items in a menu path. Frequently in text, an ANSYS command is followed by its GUI equivalent in parentheses: the ***GET** command (**Utility Menu> Parameters> Get Scalar Data**)
ITALICS	Uppercase italic letters indicate command arguments for numeric values (such as *VALUE*, *INC*, *TIME*). On some commands, non-numeric convenience labels (for example, ALL and P) can also be entered for these arguments.
Italics	Mixed case italic letters indicate command arguments for alphanumeric values (for example, *Lab* or *Fname*). The manual also uses italic text for emphasis.
TYPEWRITER	Typewriter font indicates command input listings and ANSYS output listings.
Note--	This text introduces note paragraphs. A note contains information that supplements the main topic being discussed.

Any mention of a command or element name in this volume implies a reference to the appropriate command or element description (in the *Commands Reference* or *Elements Reference* manuals, respectively) for more detailed information.

1.1.1. Product Codes

Near the top of the first page of each element description, you will see a list of product codes. These codes represent the products in the ANSYS Family of Products. The element is valid only for those products whose symbols are listed. An element that is valid in the entire set of products would have the following list of products:

MP ME ST PR PRN DS DSS FL EM EH DY PP VT

The codes represent each of the products in the ANSYS suite of products:

Code	Product	Code	Product
MP	ANSYS Multiphysics	EM	ANSYS Emag - Low Frequency
ME	ANSYS Mechanical	EH	ANSYS Emag - High Frequency
ST	ANSYS Structural	FL	ANSYS FLOTRAN
PR	ANSYS Professional - Nonlinear Thermal	DS	ANSYS DesignSpace
PRN	ANSYS Professional - Nonlinear Structural	DSS	ANSYS DesignSpace - Structural
DY	ANSYS LS-DYNA	PP	ANSYS PrepPost
VT	ANSYS DesignXplorer		

For a brief description of each product, see *Section 1.1.2: Applicable ANSYS Products*.

If the symbol for a product does not appear, then that element is either not valid or not applicable in the corresponding product, and should not be used. For example, if the PR and FL symbols are not listed, the pertinent element is not valid in the ANSYS Professional or ANSYS FLOTRAN products, but is valid in each of the remaining ANSYS products.

In most cases, commands and elements that are valid in ANSYS Multiphysics are also valid in the ANSYS ED and ANSYS Multiphysics 1, 2, and 3 products. Exceptions are noted under Product Restrictions for the applicable command or element. ANSYS ED and ANSYS Multiphysics 1, 2, and 3 are not listed as separate product codes.

1.1.2. Applicable ANSYS Products

This manual applies to the following ANSYS products:

ANSYS Multiphysics (includes all structural, thermal, electromagnetics, and computational fluid dynamics (CFD) capabilities, excludes explicit dynamics)
ANSYS Mechanical (includes all structural and thermal capabilities; excludes electromagnetics, CFD, and explicit dynamics capabilities)
ANSYS Structural (includes all structural linear and nonlinear capabilities)
ANSYS Professional (Nonlinear Thermal and Nonlinear Structural)
ANSYS Emag (Low Frequency and High Frequency)
ANSYS FLOTRAN
ANSYS LS-DYNA
ANSYS PrepPost
ANSYS ED

Some command arguments and element KEYOPT settings have defaults in the derived products that are different from those in ANSYS Multiphysics. These cases are clearly documented under the Product Restrictions section of the affected commands and elements. If you plan to use your derived product input file in ANSYS Multiphysics, you should explicitly input these settings in the derived product, rather than letting them default; otherwise, behavior in ANSYS Multiphysics will be different.

Note

While ANSYS connection capabilities, High Performance Computing, and ANSYS LSF/Batch are included as part of the ANSYS release distribution, they are separately-licensed products. Consult your ASD if you want to install and run any of the separately-licensed products at your site.

Even though an element may be available in a particular product, some of its options may not be. Most element descriptions contain a Product Restrictions section which details the specific restrictions the element has in each of the products.

1.2. ANSYS Product Capabilities

A complete list of engineering capabilities available in the various ANSYS products can be found on our website at http://www.ansys.com/assets/brochures/capabilities-brochure.pdf.

Chapter 2: General Element Features

The ANSYS element library consists of more than 100 different element formulations or *types*. (Not all element types or features are available in all ANSYS products. These restrictions are detailed in Section 4.*n*.4, "Product Restrictions," for each element.)

Many features are common to all ANSYS elements in the element library. These features are discussed in this chapter. The individual elements are described in *Chapter 4: Element Library*.

The following element feature topics are available:

2.1. Element Input

Chapter 4: Element Library includes a summary table of element input. See *BEAM3 Input Summary* for a sample input data table. This table usually contains the following items:

Element Name
Nodes
Degrees of Freedom
Real Constants
Material Properties
Surface Loads
Body Loads
Special Features
KEYOPTS

Details on these items follow:

2.1.1. Element Name

An element type is identified by a name (8 characters maximum), such as BEAM3, consisting of a group label (BEAM) and a unique, identifying number (3). The element descriptions in *Chapter 4: Element Library* are arranged in order of these identification numbers. The element is selected from the library for use in the analysis by inputting

its name on the element type command [**ET**]. See *Chapter 3: Element Characteristics* for a list of all available elements.

2.1.2. Nodes

The nodes associated with the element are listed as I, J, K, etc. Elements are connected to the nodes in the sequence and orientation shown on the input figure for each element type. This connectivity can be defined by automatic meshing, or may be input directly by the user with the **E** command. The node numbers must correspond to the order indicated in the "Nodes" list. The I node is the first node of the element. The node order determines the element coordinate system orientation for some element types. See *Section 2.3: Coordinate Systems* for a description of the element coordinate system.

2.1.3. Degrees of Freedom

Each element type has a degree of freedom set, which constitute the primary nodal unknowns to be determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltages, etc. Derived results, such as stresses, heat flows, etc., are computed from these degree of freedom results. Degrees of freedom are not defined on the nodes explicitly by the user, but rather are implied by the element types attached to them. The choice of element types is therefore, an important one in any ANSYS analysis.

2.1.4. Real Constants

Data which are required for the calculation of the element matrix, but which cannot be determined from the node locations or material properties, are input as "real constants." Typical real constants include area, thickness, inner diameter, outer diameter, etc. A basic description of the real constants is given with each element type. The *Theory Reference for ANSYS and ANSYS Workbench* section describing each element type shows how the real constants are used within the element. The real constants are input with the **R** command. The real constant values input on the command must correspond to the order indicated in the "Real Constants" list.

2.1.5. Material Properties

Various material properties are used for each element type. Typical material properties include Young's modulus (of elasticity), density, coefficient of thermal expansion, thermal conductivity, etc. Each property is referenced by an ANSYS label - EX, EY, and EZ for the directional components of Young's modulus, DENS for density, and so on. All material properties can be input as functions of temperature.

Some properties for non-thermal analyses are called *linear* properties because typical solutions with these properties require only a single iteration. Properties such as stress-strain data are called *nonlinear* because an analysis with these properties requires an iterative solution. A basic description of the linear material properties is given in *Section 2.4: Linear Material Properties* and of the nonlinear properties in *Section 2.5: Data Tables - Implicit Analysis*. Linear material properties are input with the **MP** family of commands while nonlinear properties are input with the **TB** family of commands. Some elements require other special data which need to be input in tabular form. These tabular data are also input with the **TB** commands and are described with the element in *Chapter 4: Element Library*, or in *Section 2.5: Data Tables - Implicit Analysis* if they apply to a family of elements. The *Theory Reference for ANSYS and ANSYS Workbench* shows how the properties and special data are actually used within the element.

Material models used in explicit dynamic analyses are discussed in Material Models in the *ANSYS LS-DYNA User's Guide*.

2.1.6. Surface Loads

Various element types allow surface loads. Surface loads are typically pressures for structural element types, convections or heat fluxes for thermal element types, etc. See *Section 2.8: Node and Element Loads* for additional details.

2.1.7. Body Loads

Various element types allow body loads. Body loads are typically temperatures for structural element types, heat generation rates for thermal element types, etc. See *Section 2.8: Node and Element Loads* for details. Body loads are designated in the "Input Summary" table of each element by a label and a list of load values at various locations within the element. For example, for element type PLANE42, the body load list of "Temperatures: T(I), T(J), T(K), T(L)" indicates that temperature body loads are allowed at the I, J, K, and L node locations of the element. Body loads are input with the **BF** or **BFE** commands. The load values input on the **BFE** command must correspond to the order indicated in the "Body Load" list.

2.1.8. Special Features

The keywords in the "Special Features" list indicate that certain additional capabilities are available for the element. Most often these features make the element nonlinear and require that an iterative solution be done. For a description of the special feature "Plasticity," see *Section 2.5.2: Nonlinear Stress-Strain Materials*; for "Creep," see *Section 2.5.13: Creep Equations*; and for "Swelling," see *Section 2.5.15: Swelling Equations*. See Nonlinear Structural Analysis in the *Structural Analysis Guide* and the *Theory Reference for ANSYS and ANSYS Workbench* for information about "Large Deflection," "Large Strain," "Stress Stiffening," "Adaptive Descent," "Error Estimation," "Birth and Death," "Hyperelasticity," and "Viscoelasticity."

2.1.9. KEYOPTS

KEYOPTS (or key options) are switches, used to turn various element options on or off. KEYOPT options include stiffness formulation choices, printout controls, element coordinate system choices, etc. A basic description of the KEYOPTS is given with each element type. The *Theory Reference for ANSYS and ANSYS Workbench* section for the element type shows how some of the KEYOPTS are used within the element. KEYOPTS are identified by number, such as KEYOPT(1), KEYOPT(2), etc., with each numbered KEYOPT able to be set to a specific value. Values for the first six KEYOPTS (KEYOPT(1) through KEYOPT(6)) may be input with the **ET** or **KEYOPT** commands. Values for KEYOPT(7) or greater on any element are input with the **KEYOPT** command.

Note

The defaults for element key options are chosen to be most convenient for the ANSYS product you are using, which means that some of the defaults may be different in some of the ANSYS products. These cases are clearly documented under the "Product Restrictions" section of the affected elements. If you plan to use your input file in more than one ANSYS product, you should explicitly input these settings, rather than letting them default; otherwise, behavior in the other ANSYS product may be different.

2.2. Solution Output

The output from the solution consists of the nodal solution (or the primary degree of freedom solution) and the element solution (or the derived solution). Each of these solutions is described below. Solution output is written to the output file (`Jobname.OUT`, also known as the "printout"), the database, and the results file (`Jobname.RST`, `Jobname.RTH`, `Jobname.RMG`, or `Jobname.RFL`). The output file can be viewed through the GUI, while the database and results file data (sometimes called the "post data") can be postprocessed.

The output file contains the nodal DOF solution, nodal and reaction loads, and the element solutions, depending on the **OUTPR** settings. The element solutions are primarily the centroidal solution values for each element. Most elements have KEYOPTS to output more information (e.g. integration points).

The results file contains data for all requested [**OUTRES**] solutions, or load steps. In POST1, you issue the **SET** command to identify the load step you wish to postprocess. Results items for the area and volume elements are generally retrieved from the database by commands such as **PRNSOL**, **PLNSOL**, **PRESOL**, **PLESOL**, etc. The labels on these commands correspond to the labels shown in the input and output description tables for each element (such as *PLANE42 Input Summary* and *Table 1, "PLANE42 Element Output Definitions"* for PLANE42). For example, postprocessing the X-stress (typically labeled SX) is identified as item S and component X on the postprocessing commands. Coordinate locations XC, YC, ZC are identified as item CENT and component X, Y, or Z. Only items shown both on the individual command and in the element input/output tables are available for use with that command. An exception is EPTO, the total strain, which is available for all structural solid and shell elements even though it is not shown in the output description tables for those elements.

Generic labels do not exist for some results data, such as integration point data, all derived data for structural line elements (such as spars, beams, and pipes) and contact elements, all derived data for thermal line elements, and layer data for layered elements. Instead, a sequence number is used to identify these items (described below).

2.2.1. Nodal Solution

The nodal solution from an analysis consists of:

- the degree of freedom (DOF) solution, such as nodal displacements, temperatures, and pressures
- the reaction solution calculated at constrained nodes - forces at displacement constraints, heat flows at temperature DOF constraints, fluid flows at pressure DOF constraints, and so on.

The DOF solution is calculated for all active degrees of freedom in the model, which are determined by the union of all DOF labels associated with all the active element types. It is output at all degrees of freedom that have a nonzero stiffness or conductivity and can be controlled by **OUTPR**,NSOL (for printed output) and **OUTRES**,NSOL (for results file output).

The reaction solution is calculated at all nodes that are constrained (**D**, **DSYM**, etc.). Its output can be controlled by **OUTPR**,RSOL and **OUTRES**,RSOL.

For vector degrees of freedom and corresponding reactions, the output during solution is in the nodal coordinate system. If a node was input with a rotated nodal coordinate system, the output nodal solution will also be in the rotated coordinate system. For a node with the rotation $\theta_{xy} = 90°$, the printed UX solution will be in the nodal X direction, which in this case corresponds to the global Y direction. Rotational displacements (ROTX, ROTY, ROTZ) are output in radians, and phase angles from a harmonic analysis are output in degrees.

2.2.2. Element Solution

The element output items (and their definitions) are shown along with the element type description. Not all of the items shown in the output table will appear at all times for the element. Normally, items not appearing are either not applicable to the solution or have all zero results and are suppressed to save space. However, except for the coupled-field elements PLANE223, SOLID226, and SOLID227, coupled-field forces appear if they are computed to be zero. The output is, in some cases, dependent on the input. For example, for thermal elements accepting either surface convection (CONV) or nodal heat flux (HFLUX), the output will be either in terms of convection or heat flux. Most of the output items shown appear in the element solution listing. Some items do not appear in the solution listing but are written to the results file.

Most elements have 2 tables which describe the output data and ways to access that data for the element. These tables are the "Element Output Definitions" table and the "Item and Sequence Numbers" tables used for accessing data through the **ETABLE** and **ESOL** commands.

2.2.2.1. The Element Output Definitions Table

The first table, "Element Output Definitions," describes possible output for the element. In addition, this table outlines which data are available for solution printout (`Jobname.OUT` and/or display to the terminal), and which data are available on the results file (`Jobname.RST`, `Jobname.RTH`, `Jobname.RMG`, etc.). It's important to remember that only the data which you request with the solution commands **OUTPR** and **OUTRES** are included in printout and on the results file, respectively. See *Table 1, "BEAM3 Element Output Definitions"* for a sample element output definitions table. As an added convenience, items in this table which are available through the Component Name method of the **ETABLE** command are identified by special notation (:) included in the output label. See The General Postprocessor (POST1) in the *Basic Analysis Guide* for more information. The label portion before the colon corresponds to the Item field on the **ETABLE** command, and the portion after the colon corresponds to the *Comp* field. For example, S:EQV is defined as equivalent stress, and the **ETABLE** command for accessing this data would be:

ETABLE,ABC,S,EQV

where ABC is a user-defined label for future identification on listings and displays. Other data having labels without colons can be accessed through the Sequence Number method, discussed with the "Item and Sequence Number" tables below.

In some cases there is more than one label which can be used after the colon, in which case they are listed and separated by commas. The **Definition** column defines each label and, in some instances, also lists the label used on the printout, if different. The **O** column indicates those items which are written to the output window and/or the output file. The **R** column indicates items which are written to the results file and which can be obtained in postprocessing.

Note

If an item is not marked in the **R** column, it cannot be stored in the "element table."

2.2.2.2. The Item and Sequence Number Table

Many elements also have a table, or set of tables, that list the *Item* and sequence number required for data access using the Sequence Number method of the **ETABLE** command. See The General Postprocessor (POST1) in the *Basic Analysis Guide* for an example. The number of columns in each table and the number of tables per element vary depending on the type of data available and the number of locations on the element where data was calculated. For structural line elements, for example, the KEYOPT(9) setting will determine the number of locations (intermediate points) along the element where data is to be calculated.

For example, assume we want to determine the sequence number required to access the member moment in the Z direction (MMOMZ) for a BEAM3 element. Assume also that the data we want to obtain is at end J, and that KEYOPT(9) = 1, that is, data has also been calculated at one intermediate location. See *Table 4, "BEAM3 Item and Sequence Numbers (KEYOPT(9) = 3)"* for a sample item and sequence numbers table. Locate MMOMZ under the "Name" column. Notice that the Item is listed as SMISC. SMISC refers to summable miscellaneous items, while NMISC refers to nonsummable miscellaneous items (see the *Basic Analysis Guide* for more details). Follow across the row until you find the sequence number, 18, in the J column. The correct command to move MMOMZ at end J for BEAM3 (KEYOPT(9) = 1) to the element table is:

ETABLE,ABC,SMISC,18

ABC is a user-defined label for later identification on listings and displays.

2.2.2.3. Surface Loads

Pressure output for structural elements shows the input pressures expanded to the element's full tapered-load capability. See the **SF**, **SFE**, and **SFBEAM** commands for pressure input. For example, for element type PLANE42, which has an input load list of "Pressures: Face 1 (J-I), Face 2 (K-J), Face 3 (L-K), Face 4 (I-L)," the output PRESSURE line expands the pressures to P1(J), P1(I); P2(K), P2(J); P3(L), P3(K); and P4(I), P4(L). P1(J) should be interpreted as the pressure for load key 1 (the pressure normal to face 1) at node J; P1(I) is load key 1 at node I; etc. If the pressure is input as a constant instead of tapered, both nodal values of the pressure will be the same. Beam elements which allow an offset from the node have addition output labeled OFFST. To save space, pressure output is often omitted when values are zero. Similarly, other surface load items (such as convection (CONV) and heat flux (HFLUX)), and body load input items (such as temperature (TEMP), fluence (FLUE), and heat generation (HGEN)), are often omitted when the values are zero (or, for temperatures, when the T-TREF values are zero).

2.2.2.4. Centroidal Solution [output listing only]

Output such as stress, strain, temperature, etc. in the output listing is given at the centroid (or near center) of the element. The location of the centroid is updated if large deflections are used. The output quantities are calculated as the average of the integration point values (see the *Theory Reference for ANSYS and ANSYS Workbench*). The component output directions for vector quantities correspond to the input material directions which, in turn, are a function of the element coordinate system. For example, the SX stress is in the same direction as EX. In postprocessing, **ETABLE** may be used to compute the centroidal solution of each element from its nodal values.

2.2.2.5. Surface Solution

Surface output is available in the output listing on certain *free* surfaces of solid elements. A free surface is a surface not connected to any other element and not having any DOF constraint or nodal force load on the surface. Surface output is not valid on surfaces which are not free or for elements having nonlinear material properties. Surface output is also not valid for elements deactivated [**EKILL**] and then reactivated [**EALIVE**]. Surface output does not include large strain effects.

The surface output is automatically suppressed if the element has nonlinear material properties. Surface calculations are of the same accuracy as the displacement calculations. Values are not extrapolated to the surface from the integration points but are calculated from the nodal displacements, face load, and the material property relationships. Transverse surface shear stresses are assumed to be zero. The surface normal stress is set equal to the surface pressure. Surface output should not be requested on condensed faces or on the zero-radius face (center line) of an axisymmetric model.

For 3-D solid elements, the face coordinate system has the x-axis in the same general direction as the first two nodes of the face, as defined with pressure loading. The exact direction of the x-axis is on the line connecting the midside nodes or midpoints of the two opposite edges. The y-axis is normal to the x-axis, in the plane of the face.

Table 2.1, "Output Available through ETABLE" lists output available through the **ETABLE** command using the Sequence Number method (Item = SURF). See the appropriate table (4.xx.2) in the individual element descriptions for definitions of the output quantities.

Table 2.1 Output Available through ETABLE

	Element Dimensionality		
snum	**3-D**	**2-D**	**Axisymm**
1	FACE	FACE	FACE

	Element Dimensionality		
snum	**3-D**	**2-D**	**Axisymm**
2	AREA	AREA	AREA
3	TEMP	TEMP	TEMP
4	PRES	PRES	PRES
5	EPX	EPPAR	EPPAR
6	EPY	EPPER	EPPER
7	EPZ	EPZ	EPZ
8	EPXY	0	EPSH [*1*]
9	SX	SPAR	SPAR
10	SY	SPER	SPER
11	SZ	SZ	SZ
12	SXY	0	0
13	0	0	0
14	0	0	SSH [*1*]
15	S1	S1	S1
16	S2	S2	S2
17	S3	S3	S3
18	SINT	SINT	SINT
19	SEQV	SEQV	SEQV

1. Axiharmonic only

If an additional face has surface output requested, then snum 1-19 are repeated as snum 20-38.

Convection heat flow output may be given on convection surfaces of solid thermal elements. Output is valid on interior as well as exterior surfaces. Convection conditions should not be defined on condensed faces or on the zero-radius face (center line) of an axisymmetric model.

2.2.2.6. Integration Point Solution [output listing only]

Integration point output is available in the output listing with certain elements. The location of the integration point is updated if large deflections are used. See the element descriptions in the *Theory Reference for ANSYS and ANSYS Workbench* for details about integration point locations and output. Also the **ERESX** command may be used to request integration point data to be written as nodal data on the results file.

2.2.2.7. Element Nodal Solution

The term *element nodal* means element data reported for each element at its nodes. This type of output is available for 2-D and 3-D solid elements, shell elements, and various other elements. Element nodal data consist of the element derived data (e.g. strains, stresses, fluxes, gradients, etc.) evaluated at each of the element's nodes. These data are usually calculated at the interior integration points and then extrapolated to the nodes. Exceptions occur if an element has active (nonzero) plasticity, creep, or swelling at an integration point or if **ERESX**,NO is input. In such cases the nodal solution is the value at the integration point nearest the node. See the *Theory Reference for ANSYS and ANSYS Workbench* for details. Output is usually in the element coordinate system. Averaging of the nodal data from adjacent elements is done within POST1.

2.2.2.8. Element Nodal Loads

These are an element's loads (forces) acting on each of its nodes. They are printed out at the end of each element output in the nodal coordinate system and are labeled as static loads. If the problem is dynamic, the damping loads and inertia loads are also printed. The output of element nodal loads can be controlled by **OUTPR**,NLOAD (for printed output) and **OUTRES**,NLOAD (for results file output).

Element nodal loads relate to the reaction solution in the following way: the sum of the static, damping, and inertia loads at a particular degree of freedom, summed over all elements connected to that degree of freedom, plus the applied nodal load (**F** or **FK** command), is equal to the negative of the reaction solution at that same degree of freedom.

2.2.2.9. Nonlinear Solution

For information about nonlinear solution due to material nonlinearities, see the *Theory Reference for ANSYS and ANSYS Workbench*. Nonlinear strain data (EPPL, EPCR, EPSW, etc.) is always the value from the nearest integration point. If creep is present, stresses are computed after the plasticity correction but before the creep correction. The elastic strains are printed after the creep corrections.

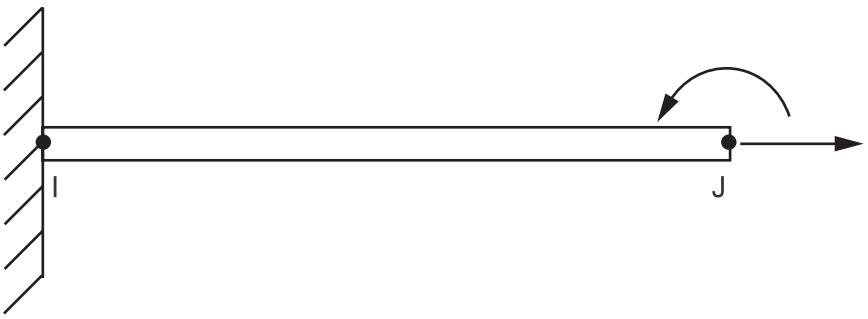
2.2.2.10. Plane and Axisymmetric Solutions

A 2-D solid analysis is based upon a "per unit of depth" calculation and all appropriate output data are on a "per unit of depth" basis. Many 2-D solids, however, allow an option to specify the depth (thickness). A 2-D axisymmetric analysis is based on a full 360°. Calculation and all appropriate output data are on a full 360° basis. In particular, the total forces for the 360° model are output for an axisymmetric structural analysis and the total convection heat flow for the 360° model is output for an axisymmetric thermal analysis. For axisymmetric analyses, the X, Y, Z, and XY stresses and strains correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively. The global Y axis must be the axis of symmetry, and the structure should be modeled in the +X quadrants.

2.2.2.11. Member Force Solution

Member force output is available with most structural line elements. The listing of this output is activated with a KEYOPT described with the element and is in addition to the nodal load output. Member forces are in the element coordinate system and the components correspond to the degrees of freedom available with the element. For example, member forces printed for BEAM3 would be MFORX, MFORY, MMOMZ.

For BEAM3, BEAM4, BEAM44, BEAM54, SHELL61, PIPE16, PIPE17, PIPE18, PIPE20, PIPE59, and PIPE60, the signs of their member forces at all locations along the length of the elements are based on force equilibrium of the member segment from end I to that location. For example, for the simple one-element cantilever beam loaded as shown, the tensile force and the bending moments are positive at all points along the element, including both ends.



2.2.2.12. Failure Criteria

Failure criteria are commonly used for orthotropic materials. They can be input using either the **FC** commands or the **TB** commands. The **FC** command input is used in POST1. The **TB** command input is used directly in the composite elements and is described below.

The failure criteria table is started by using the **TB** command (with *Lab* = FAIL). The data table is input in two parts:

- the failure criterion keys
- the failure stress/strain data.

Data not input are assumed to be zero. See the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the predefined failure criteria. The six failure criterion keys are defined with the **TBDATA** command following a special form of the **TBTEMP** command [**TBTEMP**,,CRIT] to indicate that the failure criterion keys are defined next. The constants (C1-C6) entered on the **TBDATA** command are:

Table 2.2 Orthotropic Material Failure Criteria Data

Constant	Meaning
1	Maximum Strain Failure Criterion - Output as FC1 (uses strain constants 1-9) 0 - Do not include this predefined criterion. 1 - Include this predefined criterion. -1 - Include user-defined criterion with subroutine USRFC1.
2	Maximum Stress Failure Criterion - Output as FC2 (uses stress constants 10-18) Options are the same as for constant 1, except subroutine is USRFC2.
3	Tsai-Wu Failure Criterion - Output as FC3 (uses constants 10-21) 0 - Do not include this predefined criterion 1 - Include the Tsai-Wu strength index 2 - Include the inverse of the Tsai-Wu strength ratio -1 - Include user-defined criterion with subroutine USRFC3
4-6	User-defined Failure Criteria - Output as FC4 TO FC6 0 - Do not include this criterion. -1 - Include user-defined criteria with subroutines USRFC4, USRFC5, USRFC6, respectively.

The failure data, which may be temperature-dependent, must be defined with the **TBDATA** command following a temperature definition on the **TBTEMP** command. Strains must have absolute values less than 1.0. Up to six temperatures (*NTEMP* = 6 maximum on the **TB** command) may be defined with the **TBTEMP** commands. The constants (C1-C21) entered on the **TBDATA** command (6 per command), after each **TBTEMP** command, are:

TBDATA Constants for the TBTEMP Command

Constant - **(Symbol)** - **Meaning**

1 - (ε_{xt}^{f}) - Failure strain in material x-direction in tension (must be positive).

2 - (ε_{xc}^{f}) - Failure strain in material x-direction in compression (default = -ε_{xt}^{f}) (may not be positive).

3 - (ε^f_{yt}) - Failure strain in material y-direction in tension (must be positive).

4 - (ε^f_{yc}) - Failure strain in material y-direction in compression (default = - ε^f_{yt}) (may not be positive).

5 - (ε^f_{zt}) - Failure strain in material z-direction in tension (must be positive).

6 - (ε^f_{zc}) - Failure strain in material z-direction in compression (default = - ε^f_{zt}) (may not be positive).

7 - (ε^f_{xy}) - Failure strain in material x-y plane (shear) (must be positive).

8 - (ε^f_{yz}) - Failure strain in material y-z plane (shear) (must be positive).

9 - (ε^f_{xz}) - Failure strain in material x-z plane (shear) (must be positive).

10 - (σ^f_{xt}) - Failure stress in material x-direction in tension (must be positive).

11 - (σ^f_{xc}) - Failure stress in material x-direction in compression (default = - σ^f_{xt}) (may not be positive).

12 - (σ^f_{yt}) - Failure stress in material y-direction in tension (must be positive).

13 - (σ^f_{yc}) - Failure stress in material y-direction in compression (default = - σ^f_{yt}) (may not be positive).

14 - (σ^f_{zt}) - Failure stress in material z-direction in tension (must be positive).

15 - (σ^f_{zc}) - Failure stress in material z-direction in compression (default = - σ^f_{zt}) (may not be positive).

16 - (σ^f_{xy}) - Failure stress in material x-y plane (shear) (must be positive).

17 - (σ^f_{yz}) - Failure stress in material y-z plane (shear) (must be positive).

18 - (σ^f_{xz}) - Failure stress in material x-z plane (shear) (must be positive).

19 - (C^*_{xy}) - x-y coupling coefficient for Tsai-Wu Theory (default = -1.0).

20 - (C^*_{yz}) - y-z coupling coefficient for Tsai-Wu Theory (default = -1.0).

21 - (C^*_{xz}) - x-z coupling coefficient for Tsai-Wu Theory (default = -1.0).

Note

Tsai-Wu coupling coefficients must be between -2.0 and 2.0. Values between -1.0 and 0.0 are recommended. For 2-D analysis, set σ_{zt}^{f}, σ_{zc}^{f}, σ_{yz}^{f}, and σ_{xz}^{f} to a value several orders of magnitude larger than σ_{xt}^{f}, σ_{xc}^{f}, or σ_{xy}^{f}; and set C_{xz} and C_{yz} to zero.

See the **TB** command for a listing of the elements that can be used with the FAIL material option.

See Specifying Failure Criteria in the *Structural Analysis Guide* for more information on this material option.

2.3. Coordinate Systems

2.3.1. Element Coordinate Systems

The element coordinate system is used for orthotropic material input directions, applied pressure directions, and, under some circumstances, stress output directions. (See Rotating Results to a Different Coordinate System in the *Basic Analysis Guide* for a discussion of the circumstances in which the program uses the element coordinate system for stress output directions.) A default element coordinate system orientation is associated with each element type. In general, these systems are described below. Elements departing from this description have their default element coordinate system orientation described in *Chapter 4: Element Library*.

Element coordinate systems are right-handed, orthogonal systems. For line elements (such as LINK1), the default orientation is generally with the x-axis along the element I-J line. For solid elements (such as PLANE42 or SOLID45), the default orientation is generally parallel to the global Cartesian coordinate system. For area shell elements (such as SHELL63), the default orientation generally has the x-axis aligned with element I-J side, the z-axis normal to the shell surface (with the outward direction determined by the right-hand rule around the element from node I to J to K), and the y-axis perpendicular to the x and z-axes.

Unless otherwise changed, the element coordinate system orientation is the default orientation for that element type as described above. The orientation may be changed for area and volume elements by making it parallel to a previously defined local system (see the **ESYS** command) or, for some elements, by a KEYOPT selection (see KEYOPT(1) for PLANE42). If both are specified, the **ESYS** definition overrides. A further rotation, relative to the previous orientation, is allowed for some elements by a real constant angle specification (see, for example, the real constant THETA for SHELL63). Note that if no **ESYS** or KEYOPT orientation is specified, the real constant angle rotation (if any) is relative to the default orientation. The coordinate systems of axisymmetric elements may only be rotated about the global Z-axis.

For shell elements, the **ESYS** orientation uses the projection of the local system on the shell surface. The element x-axis is determined from the projection of the local x-axis on the shell surface. If the projection is a point (or the angle between the local x-axis and the normal to the shell is 0° (plus a tolerance of 45°)), the local y-axis projection is used for the element x-axis direction. The z and y-axes are determined as described for the default orientation. For non-midside-node elements, the projection is evaluated at the element centroid and is assumed constant in direction throughout the element. For midside-node elements, the projection is evaluated at each integration point and may vary in direction throughout the element. For axisymmetric elements, only rotations in the X-Y plane are valid. Some elements also allow element coordinate system orientations to be defined by user written subroutines (see the *Guide to ANSYS User Programmable Features*).

Layered elements use the x-axis of the element coordinate system as a base from which to rotate each layer to the layer coordinate system. The layers are rotated by the angles input on the **SECDATA** or **RMORE** commands. Material properties, stresses, and strains for layered elements are based on the layer coordinate system, not the element coordinate system.

All element coordinate systems shown in the element figures assume that no ESYS orientation is specified. Element coordinate systems may be displayed as a triad with the **/PSYMB** command or as an ESYS number (if specified) with the **/PNUM** command. Triad displays do not include the effects of any real constant angle, except for BEAM4 elements. For large deflection analyses, the element coordinate system rotates from the initial orientation described above by the amount of rigid body rotation of the element.

2.3.2. Elements that Operate in the Nodal Coordinate System

A few special elements operate totally in the nodal coordinate system:

COMBIN14 Spring-Damper with KEYOPT(2) = 1, 2, 3, 4, 5, or 6
MATRIX27 Stiffness, Damping, or Mass Matrix
COMBIN37 Control Element
FLUID38 Dynamic Fluid Coupling
COMBIN39 Nonlinear Spring with KEYOPT(4) = 0
COMBIN40 Combination Element
TRANS126 Electromechanical Transducer

These elements are defined in the nodal coordinate systems. This allows for easy directional control, especially for the case of 2-node elements with coincident nodes. If UX, UY, or UZ degrees of freedom are being used, the nodes are not coincident, and the load is not acting parallel to the line connecting the 2 nodes, there is no mechanism for the element to transfer the resulting moment load, resulting in loss of moment equilibrium. The one exception is MATRIX27, which can include moment coupling when appropriate additional terms are added to the matrix.

There are some things to consider if any of the nodes have been rotated, for example with the **NROTAT** command:

- If the nodes of elements containing more than one node are not rotated in the exact same way, force equilibrium may not be maintained.
- Accelerations operate normally in the global Cartesian system. But since there is no transformation done between the nodal and global systems, the accelerations will effectively act on any element mass in the nodal system, giving unexpected results. Therefore, it is recommended not to apply accelerations when these elements use rotated nodes.
- Mass and inertia relief calculations will not be correct.

2.4. Linear Material Properties

The linear material properties used by the element type are listed under "Material Properties" in the input table for each element type and are input using the **MP** command. A brief description (including the label used in the MP command) of all linear material properties is given in *Table 2.3, "Linear Material Properties "* at the end of this section. These properties (which may be functions of temperature) are called *linear* properties because typical non-thermal analyses with these properties require only a single iteration. Conversely, if properties needed for a thermal analysis (e.g., KXX) are temperature-dependent, the problem is nonlinear. Properties such as stress-strain data (described in *Section 2.5.2: Nonlinear Stress-Strain Materials*) are called *nonlinear* properties because an analysis with these properties requires an iterative solution.

Linear material properties that are required for an element, but which are not defined, use the default values as described below (except that EX and KXX must be input with a nonzero value where applicable). Any additional material properties are ignored.

The X, Y, and Z refer to the element coordinate system. In general, if a material is isotropic, only the "X" and possibly the "XY" term is input. See the *Theory Reference for ANSYS and ANSYS Workbench* for material property details.

Structural material properties must be input as an isotropic, orthotropic, or anisotropic material.

If the material is isotropic: Young's modulus (EX) must be input. Poisson's ratio (PRXY or NUXY) defaults to 0.3. If a zero value is desired, input PRXY or NUXY with a zero or blank value. The shear modulus (GXY) defaults to EX/(2(1+NUXY)). If GXY is input, it must match EX/(2 (1+NUXY)). Hence, the only reason for inputting GXY is to ensure consistency with the other two properties. Also, Poisson's ratio should not be equal to or greater than 0.5.

If the material is orthotropic: EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ), GXY, GYZ, and GXZ must all be input if the element type uses the material property. There are no defaults. Note that, for example, if only EX and EY are input (with different values) to a plane stress element, an error will result indicating that the material is orthotropic and that GXY and NUXY are also needed.

Poisson's ratio may be input in either major (PRXY, PRYZ, PRXZ) or minor (NUXY, NUYZ, NUXZ) form, but not both for a particular material. The major form is converted to the minor form during the solve operation [**SOLVE**]. Solution output is in terms of the minor form, regardless of how the data was input. If zero values are desired, input the labels with a zero (or blank) value.

For axisymmetric analyses, the X, Y, and Z labels refer to the radial (R), axial (Z), and hoop (θ) directions, respectively. Orthotropic properties given in the R, Z, θ system should be input as follows: EX = ER, EY = EZ, and EZ = E θ. An additional transformation is required for Poisson's ratios. If the given R, Z, θ properties are column-normalized (see the *Theory Reference for ANSYS and ANSYS Workbench*), NUXY = NURZ, NUYZ = NUZ θ = (ET/EZ) *NU θZ, and NUXZ = NUR θ. If the given R, Z, θ properties are row-normalized, NUXY = (EZ/ER)*NURZ, NUYZ = (E θ/EZ)*NUZ θ = NU θZ, and NUXZ = (E θ/ER)*NUR θ.

If the material is anisotropic: The input for this is described in *Section 2.5.7: Anisotropic Elastic Materials*.

For all other orthotropic materials (including ALPX, ALPY, and ALPZ), the X, Y, and Z part of the label (e.g. KXX, KYY, and KZZ) refers to the direction (in the element coordinate system) in which that particular property acts. The Y and Z directions of the properties default to the X direction (e.g., KYY and KZZ default to KXX) to reduce the amount of input required.

Material dependent damping (DAMP) is an additional method of including structural damping for dynamic analyses and is useful when different parts of the model have different damping values. If DAMP is included, the DAMP value is added to the value defined with **BETAD** as appropriate (see the *Theory Reference for ANSYS and ANSYS Workbench*). DAMP is not assumed to be temperature dependent and is always evaluated at T = 0.0. Special purpose elements, such as COMBIN7, LINK11, CONTAC12, MATRIX27, FLUID29, and VISCO88, generally do not require damping. However, if material property DAMP is specified for these elements, the value will be used to create the damping matrix at solution time.

Constant material damping coefficient (DMPR) is a material-dependent damping coefficient that is constant with respect to the excitation frequency in harmonic analysis and is useful when different parts of the model have different damping values (see *Section 15.3: Damping Matrices* in the *Theory Reference for ANSYS and ANSYS Workbench*). DMPR is not temperature dependent and is always evaluated at T = 0.0. See *Section 5.9.3: Damping* in the *Structural Analysis Guide* for more information about DMPR.

EMIS defaults to 1.0 if not defined; however, if defined with a 0.0 (or blank) value, EMIS is taken to be 0.0.

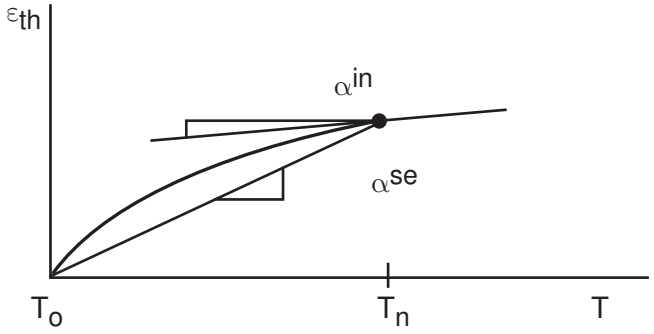
The uniform temperature does not default to REFT (but does default to TREF on the **TREF** command).

The effects of thermal expansion can be accounted for in three different (and mutually exclusive) ways:

- secant coefficient of thermal expansion (ALPX, ALPY, ALPZ)
- instantaneous coefficient of thermal expansion (CTEX, CTEY, CTEZ)

- thermal strain (THSX, THSY, THSZ)

When you use ALPX to enter values for the secant coefficient of thermal expansion (α^{se}), the program interprets those values as secant or mean values, taken with respect to some common datum or *definition* temperature. For instance, suppose you measured thermal strains in a test laboratory, starting at 23°C, and took readings at 200°, 400°, 600°, 800°, and 1000°. When you plot this strain-temperature data, you could input this directly using THSX. The slopes of the secants to the strain-temperature curve would be the mean (or secant) values of the coefficient of thermal expansion, defined with respect to the common temperature of 23° (T_o). You can also input the *instantaneous* coefficient of thermal expansion (α^{in}, using CTEX). The slopes of the tangents to this curve represent the instantaneous values. Hence, the figure below shows how the alternate ways of inputting coefficients of thermal expansion relate to each other.



The program calculates structural thermal strain as follows:

$$\varepsilon^{th} = \alpha^{se}(T) * (T - TREF)$$

where:

T = element evaluation temperature
TREF = temperature at which zero thermal strains exist (**TREF** command or REFT)
$\alpha^{se}(T)$ = secant coefficient of thermal expansion, with respect to a definition temperature (in this case, same as TREF) (ALPX)

If the material property data is in terms of instantaneous values of α, then the program will convert those instantaneous values into secant values as follows:

$$\alpha^{se}(T_n) = \frac{\int_{T_o}^{T_n} \alpha^{in}(T) * dT}{(T_n - TREF)}$$

where:

T_n = temperature at which an α^{se} value is being evaluated
T_o = definition temperature at which the α^{se} values are defined (in this case, same as TREF)
$\alpha^{in}(T)$ = instantaneous coefficient of thermal expansion at temperature T (CTEX)

If the material property data is in terms of thermal strain, the program will convert those strains into secant values of coefficients of thermal expansion as follows:

$$\alpha^{se}(T_n) = \frac{\varepsilon^{ith}(T)}{(T_n - T_{ref})}$$

where:

$\varepsilon^{ith}(T)$ = thermal strain at temperature T (THSX)

If necessary, the data is shifted so that the thermal strain is zero when $T_n = T_{ref}$. If a data point at T_{ref} exists, a discontinuity in α^{se} may be generated at $T_n = T_{ref}$. This can be avoided by ensuring that the slopes of ε^{ith} on both sides of T_{ref} match.

If the α^{se} values are based upon a definition temperature other than TREF, then you need to convert those values to TREF. This can be done using the **MPAMOD** command. Also see the *Theory Reference for ANSYS and ANSYS Workbench*.

Specific heat effects may be input with either the C (specific heat) property or the ENTH (enthalpy) property. Enthalpy has units of heat/volume and is the integral of C x DENS over temperature. If both C and ENTH are specified, ENTH will be used. ENTH should be used only in a transient thermal analysis. For phase change problems, the user must input ENTH as a function of temperature using the **MP** family of commands [**MP**, **MPTEMP**, **MPTGEN**, and **MPDATA**].

Temperature-dependent properties may be input in tabular form (value vs. temperature) or as a fourth order polynomial (value = f(temperature)). If input as a polynomial, however, evaluation is done by PREP7 at discrete temperature points and converted to tabular form. The tabular properties are then available to the elements.

Material properties are evaluated at or near the centroid of the element *or* at each of the integration points, as follows:

- For heat transfer elements, all properties are evaluated at the centroid (except for the specific heat or enthalpy, which is evaluated at the integration points).
- For structural elements PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, VISCO106, VISCO107, VISCO108, BEAM161, PLANE162, SHELL163, SOLID164, SOLID168, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, SHELL209, and SHELL281, all properties are evaluated at the integration points.
- For layered elements SOLID46, SHELL91, SHELL99, and SOLID191, all properties are evaluated at the centroid of each element.
- For all other elements, all properties are evaluated at the centroid.

If the temperature of the centroid or integration point falls below or rises above the defined temperature range of tabular data, ANSYS assumes the defined extreme minimum or maximum value, respectively, for the material property outside the defined range.

Film coefficients are evaluated as described with the **SF** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for additional details. Property evaluation at element temperatures beyond the supplied tabular range assumes a constant property at the extreme range value. An exception occurs for the ENTH property, which continues along the last supplied slope.

Table 2.3 Linear Material Properties

Lab on MP Command	Units	Description
EX	Force/Area	Elastic modulus, element x direction
EY		Elastic modulus, element y direction
EZ		Elastic modulus, element z direction
PRXY	None	Major Poisson's ratio, x-y plane

Lab on MP Command	Units	Description
PRYZ		Major Poisson's ratio, y-z plane
PRXZ		Major Poisson's ratio, x-z plane
NUXY		Minor Poisson's ratio, x-y plane
NUYZ		Minor Poisson's ratio, y-z plane
NUXZ		Minor Poisson's ratio, x-z plane
GXY	Force/Area	Shear modulus, x-y plane
GYZ		Shear modulus, y-z plane
GXZ		Shear modulus, x-z plane
ALPX	Strain/Temp	Secant coefficient of thermal expansion, element x direction
ALPY		Secant coefficient of thermal expansion, element y direction
ALPZ		Secant coefficient of thermal expansion, element z direction
CTEX	Strain/Temp	Instantaneous coefficient of thermal expansion, element x direction
CTEY		Instantaneous coefficient of thermal expansion, element y direction
CTEZ		Instantaneous coefficient of thermal expansion, element z direction
THSX	Strain	Thermal strain, element x direction
THSY		Thermal strain, element y direction
THSZ		Thermal strain, element z direction
REFT	Temp	Reference temperature (as a property) [see also **TREF**]
MU	None	Coefficient of friction (or, for FLUID29 and FLUID30 elements, boundary admittance)
DAMP	None	K matrix multiplier for damping [see also **BETAD**]
DMPR	None	Constant material damping coefficient
DENS	Mass/Vol	Mass density
KXX	Heat*Length/ (Time*Area*Temp)	Thermal conductivity, element x direction
KYY		Thermal conductivity, element y direction
KZZ		Thermal conductivity, element z direction
C	Heat/Mass*Temp	Specific heat
ENTH	Heat/Vol	Enthalpy ($\int$ DENS*C d(Temp))
HF	Heat / (Time*Area*Temp)	Convection (or film) coefficient
EMIS	None	Emissivity
QRATE	Heat/Time	Heat generation rate (MASS71 element only)
VISC	Force*Time/ Length^2	Viscosity
SONC	Length/Time	Sonic velocity (FLUID29, FLUID30, FLUID129, and FLUID130 elements only)
MURX	None	Magnetic relative permeability, element x direction
MURY		Magnetic relative permeability, element y direction
MURZ		Magnetic relative permeability, element z direction
MGXX	Current/Length	Magnetic coercive force, element x direction
MGYY		Magnetic coercive force, element y direction
MGZZ		Magnetic coercive force, element z direction

Lab on MP Command	Units	Description
RSVX	Resistance*Area/Length	Electrical resistivity, element x direction
RSVY		Electrical resistivity, element y direction
RSVZ		Electrical resistivity, element z direction
PERX	None	Electric relative permittivity, element x direction
PERY		Electric relative permittivity, element y direction
PERZ		Electric relative permittivity, element z direction
LSST	None	Dielectric loss tangent
SBKX	Voltage/Temp	Seebeck coefficient, element x direction
SBKY		Seebeck coefficient, element y direction
SBKZ		Seebeck coefficient, element z direction

2.5. Data Tables - Implicit Analysis

A data table is a series of constants that are interpreted when they are used. Data tables are always associated with a material number and are most often used to define nonlinear material data (stress-strain curves, creep constants, swelling constants, and magnetization curves). Other material properties are described in *Section 2.4: Linear Material Properties*. For some element types, the data table is used for special element input data other than material properties. The form of the data table (referred to as the **TB** table) depends upon the data being defined. Where the form is peculiar to only one element type, the table is described with the element in *Chapter 4: Element Library*. If the form applies to more than one element, it is described below and referenced in the element description. The following topics are described in this section:

- *Section 2.5.1: GUI-Inaccessible Material Properties*
- *Section 2.5.2: Nonlinear Stress-Strain Materials*
- *Section 2.5.3: Hyperelastic Material Constants*
- *Section 2.5.4: Viscoelastic Material Constants*
- *Section 2.5.5: Magnetic Materials*
- *Section 2.5.6: High-Frequency Electromagnetic Materials*
- *Section 2.5.7: Anisotropic Elastic Materials*
- *Section 2.5.8: Piezoelectric Materials*
- *Section 2.5.9: Piezoresistive Materials*
- *Section 2.5.10: Anisotropic Electric Permittivity Materials*
- *Section 2.5.11: Rate-Dependent Plastic (Viscoplastic) Materials*
- *Section 2.5.12: Gasket Materials*
- *Section 2.5.13: Creep Equations*
- *Section 2.5.14: Shape Memory Alloys*
- *Section 2.5.15: Swelling Equations*
- *Section 2.5.16: MPC184 Joint Materials*
- *Section 2.5.17: Contact Friction*
- *Section 2.5.18: Cohesive Zone Materials*

Explicit dynamics materials are discussed in Material Models in the *ANSYS LS-DYNA User's Guide*.

See Nonlinear Structural Analysis in the *Structural Analysis Guide* for additional details.

2.5.1. GUI-Inaccessible Material Properties

The following material properties are not available via the material property menus of the interactive GUI. You may specify them from the command line, and subsequent graphic display and postprocessing will still be displayed.

Material Property	TB Command Item
Anisotropic Hyperelasticity	AHYPER
Cohesive Zone Separation	CZM
Extended Drucker Prager	EDP
Contact Friction	FRIC

2.5.2. Nonlinear Stress-Strain Materials

If Table 4.*n*-1 lists "plasticity" as a "Special Feature," then several options are available to describe the material behavior of that element. Ten rate-independent plasticity options, two rate-dependent plasticity options, an elasticity option, and a user option are shown below. Select the material behavior option via menu path **Main Menu> Preprocessor> Material Props> Material Models** [**TB**,Lab].

Lab	Material Behavior Option
BKIN	Bilinear Kinematic Hardening (Rate-independent plasticity)
MKIN	Multilinear Kinematic Hardening (Rate-independent plasticity)
KINH	Multilinear Kinematic Hardening (Rate-independent plasticity)
CHABOCHE	Chaboche Nonlinear Kinematic Hardening (Rate-independent plasticity)
MISO	Multilinear Isotropic Hardening (Rate-independent plasticity)
BISO	Bilinear Isotropic Hardening (Rate-independent plasticity)
NLISO	Nonlinear Isotropic Hardening (Rate-independent plasticity)
ANISO	Anisotropic (Rate-independent plasticity)
HILL	Hill Anisotropic Potential
DP	Drucker-Prager (Rate-independent plasticity)
EDP	Extended Drucker-Prager
GURSON	Gurson Plasticicity and Damage
ANAND	Anand's Model (Rate-dependent plasticity)
MELAS	Multilinear Elastic
USER	User-defined Nonlinear Stress-Strain Material Option

All options except CHABOCHE, NLISO, HILL, DP, ANAND, and USER require a uniaxial stress-strain curve to be input. All options except HILL, ANISO, and USER must have elastically isotropic (EX = EY = EZ) materials. Required values that aren't included in the data table are assumed to be zero. If the data table is not defined (or contains all zero values), the material is assumed to be linear. The material behavior options are briefly described below. See the *Theory Reference for ANSYS and ANSYS Workbench* for more detail.

2.5.2.1. Bilinear Kinematic Hardening

This option (BKIN) assumes the total stress range is equal to twice the yield stress, so that the Bauschinger effect is included. BKIN may be used for materials that obey von Mises yield criteria (which includes most metals). The material behavior is described by a bilinear total stress-total strain curve starting at the origin and with positive stress and strain values. The initial slope of the curve is taken as the elastic modulus of the material. At the specified yield stress (C1), the curve continues along the second slope defined by the tangent modulus, C2 (having the

same units as the elastic modulus). The tangent modulus cannot be less than zero nor greater than the elastic modulus.

Initialize the stress-strain table with **TB**,BKIN. For each stress-strain curve, define the temperature [**TBTEMP**], then define C1 and C2 [**TBDATA**]. You can define up to six temperature-dependent stress-strain curves (*NTEMP* = 6 maximum on the **TB** command) in this manner. The constants C1 and C2 are:

Constant	Meaning
C1	Yield stress (Force/Area)
C2	Tangent modulus (Force/Area)

BKIN can be used with the *TB0PT* option. In this case, *TB0PT* takes two arguments. For **TB**,BKIN,,,,0, there is no stress relaxation with an increase in temperature. This option is *not* recommended for nonisothermal problems. For **TB**,BKIN,,,,1, Rice's hardening rule is applied (which does take stress relaxation with temperature increase into account). This is the default.

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

You can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

2.5.2.2. Multilinear Kinematic Hardening

You can use either **TB**,KINH (alternately **TB**,PLASTIC,,,,KINH), or **TB**,MKIN to model metal plasticity behavior under cyclic loading. These two options use the Besseling model (see the *Theory Reference for ANSYS and ANSYS Workbench*), also called the sublayer or overlay model. The material response is represented by multiple layers of perfectly plastic material; the total response is obtained by weighted average behavior of all the layers. Individual weights are derived from the uniaxial stress-strain curve. The uniaxial behavior is described by a piece-wise linear "total stress-total strain curve", starting at the origin, with positive stress and strain values. The slope of the first segment of the curve must correspond to the elastic modulus of the material and no segment slope should be larger. The slope of the stress-strain curve is assumed to be zero beyond the last user-defined stress-strain data point. In the following, the option **TB**,KINH is described first, followed by that of **TB**,MKIN.

The KINH option is recommended because layers are scaled (Rice's model), providing better representations. The KINH option allows you to define up to 40 temperature-dependent stress-strain curves. If you define more than one stress-strain curve for temperature-dependent properties, then each curve should contain the same number of points (up to a maximum of 20 points in each curve). The assumption is that the corresponding points on the different stress-strain curves represent the temperature dependent yield behavior of a particular sublayer.

For stress vs. total strain input, initialize the stress-strain table with **TB**,KINH. For stress vs. plastic strain input, initialize the stress-strain table with either **TB**,KINH,,,,PLASTIC or **TB**,PLASTIC,,,,KINH. Input the temperature of the first curve with the **TBTEMP**, then input stress and strain values using the **TBPT**. Input the remaining temperatures and stress-strain values using the same sequence (**TBTEMP** followed by **TBPT**).

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

You can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

The curve defined with the MKIN option is continuous from the origin with a maximum of five total stress-total strain points. The slope of the first segment of the curve must correspond to the elastic modulus of the material and no segment slope should be larger.

The MKIN option has the following restrictions:

- You may define up to five temperature dependent stress-strain curves.
- You may use only five points for each stress-strain curve.
- Each stress-strain curve must have the same set of strain values.

This option is used as follows:

Initialize the stress-strain table with **TB**,MKIN, followed by a special form of the **TBTEMP** command (**TB-TEMP**,,STRAIN) to indicate that strains are defined next. The constants (C1-C5), entered on the next **TBDATA** command, are the five corresponding strain values (the origin strain is not input). The temperature of the first curve is then input with **TBTEMP**, followed by the **TBDATA** command with the constants C1-C5 representing the five stresses corresponding to the strains at that temperature. You can define up to five temperature-dependent stress-strain curves (*NTEMP* = 5 max on the **TB** command) with the **TBTEMP** command.

MKIN can also be used in conjunction with the *TB0PT* option (**TB**,MKIN,,,,*TB0PT*). *TB0PT* has the following three valid arguments:

0 - No stress relaxation with temperature increase (this is not recommended for nonisothermal problems); also produces thermal ratcheting.
1 - Recalculate total plastic strain using new weight factors of the subvolume.
2 - Scale layer plastic strains to keep total plastic strain constant; agrees with Rice's model (**TB**, BKIN with *TB0PT* = 1). Produces stable stress-strain cycles.

See the **TB** command for a listing of the elements that can be used with this material option.

Note

The mechanical behavior of the **TB**,KINH option is the same as **TB**,MKIN with *TB0PT* = 2.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

You can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

2.5.2.3. Nonlinear Kinematic Hardening

This option (CHABOCHE) uses the Chaboche model (see the *Theory Reference for ANSYS and ANSYS Workbench*) for simulating the cyclic behavior of materials. Like the BKIN and MKIN options, you can use this model to simulate monotonic hardening and the Bauschinger effect. In addition, you can superpose up to five kinematic hardening models and an isotropic hardening model to simulate the complicated cyclic plastic behavior of materials, such as cyclic hardening or softening, and ratcheting or shakedown.

The Chaboche model implemented in ANSYS is:

$$\dot{X} = \sum_{i}^{n} \dot{X}_i = \frac{2}{3} \sum_{i}^{n} C_i \dot{\varepsilon}^{pl} - \gamma_i X_i \dot{p} + \frac{1}{C_i} \frac{dC_i}{d\theta} \dot{\theta} X_i$$

where:

X = back stress tensor
ε^{pl} = plastic strain tensor
p = accumulated equivalent plastic strain
θ = temperature
[A dot located above any of these quantities indicates the first derivative of the quantity with respect to time.]
C_i and γ_i = material constants that you enter as inputs
n = number of nonlinear kinematic models that you specify as *NPTS* in the **TB** command

The yield function is:

$$f(\sigma,\varepsilon^{pl}) = \bar{\sigma} - k = 0$$

where:

$\bar{\sigma}$ = effective equivalent stress

k = yield stress of materials that you enter as an input. You can also define k using BISO, MISO, or NLISO, through the **TB** command.

Initialize the data table with **TB**,CHABOCHE. For each set of data, define the temperature [**TBTEMP**], then define C1 through C*m* [**TBDATA**], where *m* = 1 + 2*NPTS*. The maximum number of constants, *m* is 11, which corresponds to 5 kinematic models [*NPTS* = 5 on the **TB** command]. The default value for *m* is 3, which corresponds to one kinematic model [*NPTS* = 1]. You can define up to 1000 temperature-dependent constants ([*NTEMP* x *m* $\leq$ 1000] maximum on the **TB** command) in this manner. The constants C1 through C(1 + 2*NPTS*) are:

Constant	Meaning
C1	k = Yield stress
C2	C_1 = Material constant for first kinematic model
C3	γ_1 = Material constant for first kinematic model
C4	C_2 = Material constant for second kinematic model
C5	γ_2 = Material constant for second kinematic model
...	...
C(2*NPTS*)	C_{NPTS} = Material constant for last kinematic model
C(1 + 2*NPTS*)	γ_{NPTS} = Material constant for last kinematic model

k, and all C and γ values in the right column are material constants in the Chaboche model (see the *Theory Reference for ANSYS and ANSYS Workbench* for details).

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

As mentioned above, you can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

2.5.2.4. Bilinear Isotropic Hardening

This option (BISO) uses the von Mises yield criteria coupled with an isotropic work hardening assumption. The material behavior is described by a bilinear stress-strain curve starting at the origin with positive stress and strain values. The initial slope of the curve is taken as the elastic modulus of the material. At the specified yield stress (C1), the curve continues along the second slope defined by the tangent modulus C2 (having the same units as the elastic modulus). The tangent modulus cannot be less than zero nor greater than the elastic modulus.

Initialize the stress-strain table with **TB**,BISO. For each stress-strain curve, define the temperature [**TBTEMP**], then define C1 and C2 [**TBDATA**]. Define up to six temperature-dependent stress-strain curves (*NTEMP* = 6 max on the **TB** command) in this manner. The constants C1 and C2 are:

Constant	Meaning
C1	Yield stress (Force/Area)
C2	Tangent modulus (Force/Area)

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

You can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

2.5.2.5. Multilinear Isotropic Hardening

This option (MISO) is similar to BISO except that a multilinear curve is used instead of a bilinear curve. It can be used for non-cyclic load histories or for those elements that do not support the multilinear kinematic hardening option (MKIN). This option may be preferred for large strain cycling where kinematic hardening could exaggerate the Bauchinger effect. The uniaxial behavior is described by a piece-wise linear total stress-total strain curve, starting at the origin, with positive stress and strain values. The curve is continuous from the origin through 100 (max) stress-strain points. The slope of the first segment of the curve must correspond to the elastic modulus of the material and no segment slope should be larger. No segment can have a slope less than zero. The slope of the stress-strain curve is assumed to be zero beyond the last user-defined stress-strain data point.

You can specify up to 20 temperature-dependent stress-strain curves. For stress vs. total strain input, initialize the curves with **TB**,MISO. For stress vs. plastic strain input, initialize the curves with TB,PLASTIC,,,,MISO. Input the temperature for the first curve [**TBTEMP**], followed by up to 100 stress-strain points (the origin stress-strain point is not input) [**TBPT**]. Define up to 20 temperature-dependent stress-strain curves (*NTEMP* = 20, maximum on the **TB** command) in this manner. The constants (X, Y) entered on the **TBPT** command (two per command) are:

Constant	Meaning
X	Strain value (Dimensionless)
Y	Corresponding stress value (Force/Area)

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

You can combine this option with other material options to simulate more complex material behaviors. See *Section 2.6: Material Model Combinations* for further information.

2.5.2.6. Nonlinear Isotropic Hardening

This option (NLISO) uses either the Voce hardening law or the power law to describe the isotropic hardening behavior of materials. You select either method by specifying the appropriate TBOPT value in **TB**,NLISO. It is especially suitable for large deformation analyses. When the power law option is combined with GURSON plasticity, ductile plasticity and damage can be modeled. See *Section 2.6: Material Model Combinations* for further information. Also, combining the NLISO Voce hardening option with the CHABOCHE nonlinear kinematic hardening option simulates cyclic hardening or softening behavior of materials (see the *Theory Reference for ANSYS and ANSYS Workbench* for details).

Once you initialize the data table with **TB**,NLISO, you define the temperature via the **TBTEMP** command, and then define either C1 through C4 (TBOPT = Voce) or C1 and C2 (TBOPT = Power) via the **TBDATA** command. You can define up to twenty temperature-dependent stress-strain curves (*NTEMP* = 20 maximum on the **TB** command) in this manner.

Voce Constants	**Meaning**
C1	k = Yield stress
C2	R_o = Material constant in Voce hardening law
C3	R_∞ = Material constant in Voce hardening law
C4	b = Material constant in Voce hardening law

Power Constants	**Meaning**
C1	k = Yield stress
C2	N = Power value in the power hardening law

See the **TB** command for a listing of the applicable elements for this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

2.5.2.7. Anisotropic

This option (ANISO) allows for different stress-strain behavior in the material x, y, and z directions as well as different behavior in tension and compression (see *Section 2.5.7: Anisotropic Elastic Materials*). A modified von Mises yield criterion is used to determine yielding. The theory is an extension of Hill's formulation as noted in the *Theory Reference for ANSYS and ANSYS Workbench*. This option is not recommended for cyclic or highly nonproportional load histories since work hardening is assumed. The principal axes of anisotropy coincide with the material (or element) coordinate system and are assumed not to change over the load history.

The material behavior is described by the uniaxial tensile and compressive stress-strain curves in three orthogonal directions and the shear stress-engineering shear strain curves in the corresponding directions. A bilinear response in each direction is assumed. The initial slope of the curve is taken as the elastic moduli of the material. At the specified yield stress, the curve continues along the second slope defined by the tangent modulus (having the same units as the elastic modulus). The tangent modulus cannot be less than zero or greater than the elastic modulus. Temperature dependent curves cannot be input. All values must be input as no defaults are defined. Input the magnitude of the yield stresses (without signs). No yield stress can have a zero value. The tensile x-direction is used as the reference curve for output quantities SEPL and EPEQ.

Initialize the stress-strain table with **TB**,ANISO. You can define up to 18 constants with **TBDATA** commands. The constants (C1-C18) entered on **TBDATA** commands (6 per command) are:

Constant	**Meaning (all units are Force/Area)**
C1-C3	Tensile yield stresses in the material x, y, and z directions
C4-C6	Corresponding tangent moduli
C7-C9	Compressive yield stresses in the material x, y, and z directions
C10-C12	Corresponding tangent moduli
C13-C15	Shear yield stresses in the material xy, yz, and xz directions
C16-C18	Corresponding tangent moduli

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

2.5.2.8. Hill's Anisotropy

This option (HILL), is used to define stress ratios for anisotropic yield and creep. Specifically, the following simulations are available by combining the HILL option with other material options, as noted:

- *Rate-independent anisotropic plasticity with isotropic hardening* - **TB**,HILL combined with **TB**,BISO or **TB**,MISO or **TB**,NLISO.
- *Rate-independent anisotropic plasticity with kinematic hardening* - **TB**,HILL combined with **TB**,BKIN or **TB**,MKIN or **TB**,KINH or **TB**,CHAB.
- *Rate-independent anisotropic plasticity with combined hardening* - **TB**,HILL combined with **TB**,CHAB and **TB**,BISO or **TB**,MISO or **TB**,NLISO.
- *Rate-dependent anisotropic plasticity (anisotropic viscoplasticity) with isotropic hardening* - **TB**,HILL combined with **TB**,BISO or **TB**,MISO or **TB**,NLISO and **TB**,RATE.
- *Anisotropic creep* - **TB**,HILL combined with **TB**,CREEP (implicit).
- *Anisotropic creep and anisotropic plasticity with isotropic hardening* - **TB**,HILL combined with **TB**,CREEP and **TB**,BISO or **TB**,MISO or **TB**,NLISO (implicit).
- *Anisotropic creep and anisotropic plasticity with kinematic hardening* - **TB**,HILL combined with **TB**,CREEP and **TB**,BKIN (implicit)

See *Section 2.6: Material Model Combinations* for more information on combining the HILL option with the plasticity and creep options.

The HILL option's material behavior is described by six constants that define the stress ratios in different directions (see the *Theory Reference for ANSYS and ANSYS Workbench* for details). All cases can be used with the following elements: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, and SHELL209.

Initialize the data table with **TB**,HILL. For each set of data, you then define the temperatures using the **TBTEMP** command, then define C1 through C6 using the **TBDATA** command. The input must then be followed by the **TB** command again, but with one of the plasticity and / or creep options.

For each set of data, you then define the temperature using the **TBTEMP** command, and then define the constants using the **TBDATA** command.

The constants C1 through C6 for the HILL option are:

Constant	Meaning	
C1	r_{xx}	Tension / Compression ii
C2	r_{yy}	
C3	r_{zz}	
C4	r_{xy}	Shear ij
C5	r_{yz}	
C6	r_{xz}	

For plasticity, r_{ij} is the ratio of the yield stress in the ij direction, to the yield stress specified for the plasticity input as part of the **TB** command.

For creep, r_{ij} is the ratio of the creep strain in the ij direction to the reference value calculated by the implicit creep equation.

2.5.2.9. Drucker-Prager

This option (DP) is applicable to granular (frictional) material such as soils, rock, and concrete and uses the outer cone approximation to the Mohr-Coulomb law (see the *Theory Reference for ANSYS and ANSYS Workbench*). The input consists of only three constants:

- the cohesion value (must be > 0)
- the angle of internal friction
- the dilatancy angle.

The amount of dilatancy (the increase in material volume due to yielding) can be controlled with the dilatancy angle. If the dilatancy angle is equal to the friction angle, the flow rule is associative. If the dilatancy angle is zero (or less than the friction angle), there is no (or less of an) increase in material volume when yielding and the flow rule is nonassociated. Temperature-dependent curves are not allowed.

Initialize the constant table with **TB**,DP. You can define up to three constants with **TBDATA** commands. The constants (C1-C3) entered on **TBDATA** are:

Constant	Meaning
C1	Cohesion value (Force/Area)
C2	Angle (in degrees) of internal friction
C3	dilatancy angle (in degrees)

See the **TB** command for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

2.5.2.10. Extended Drucker-Prager

The Extended Drucker Prager (EDP) model is also used for granular material. This model supports various combinations of yield and potential functions as noted below:

Linear Yield Function	$F = q + \alpha\sigma_m - \sigma_Y(\hat{\varepsilon}_{pl}) = 0$ where: α = material parameter referred to pressure sensitive parameter (input as C1 on **TBDATA** command using **TB**,EDP) $q = \left[\frac{3}{2}\{s\}^T[M]\{s\}\right]^{\frac{1}{2}}$ $\sigma_Y(\hat{\varepsilon}_{pl})$ = yield stress of material (input as C2 on **TBDATA** command or input using **TB**,MISO; **TB**,BISO; **TB**,NLISO; or **TB**,PLAST)

Power Law Yield Function	$q^b + \alpha\sigma_m - \sigma_Y{}^b(\hat{\varepsilon}_{pl}) = 0$ where: α = material parameter referred to pressure sensitive parameter (input as C1 on **TBDATA** command using **TB**,EDP) b = material parameter characterizing the shape of yield surface (input as C2 on **TBDATA** command using **TB**,EDP): $\sigma_Y(\hat{\varepsilon}_{pl})$ = yield stress of material (input as C3 on **TBDATA** command or input using **TB**,MISO; **TB**,BISO; **TB**,NLISO; or **TB**,PLAST)
Hyperbolic Yield Function	$\sqrt{a^2 + q^2} + \alpha\sigma_m - \sigma_Y(\hat{\varepsilon}_{pl}) = 0$: where: α = material parameter referred to pressure sensitive parameter (input as C1 on **TBDATA** command using **TB**,EDP) a = the parameter input as C2 on the **TBDATA** command. $\sigma_Y(\hat{\varepsilon}_{pl})$ = yield stress of material (input as C3 on **TBDATA** command or input using **TB**,MISO; **TB**,BISO; **TB**,NLISO; or **TB**,PLAST)
Linear Plastic Flow Potential Function	$Q = q + \alpha\sigma_m - \sigma_Y(\hat{\varepsilon}_{pl})$ Refer to the Linear Yield Function for parameter definitions.
Power Law Plastic Flow Potential Function	$Q = q^b + \alpha\sigma_m - \sigma_Y{}^b(\hat{\varepsilon}_{pl})$ Refer to the Power Law Yield Function for parameter definitions.
Hyperbolic Plastic Flow Potential Function	$Q = \sqrt{a^2 + q^2} + \alpha\sigma_m - \sigma_Y(\hat{\varepsilon}_{pl})$ Refer to the Hyperbolic Yield Function for parameter definitions.

You can use any combination of the yield and potential functions listed (above), but one of each must be specified. The CAP model, however, is a special EDP case that requires exclusive use of combined CAP potential and yield functions. More information on the EDP CAP model is provided in the following section.

When plasticity is defined by **TB**,MISO, **TB**,BISO, **TB**,NLISO, or **TB**,PLAS, that definition overrides the yield stress definition you define using **TB**,EDP and **TBDATA**.

See the EDP argument and associated specifications in the **TB** command, and also The Extended Drucker-Prager Model in the *Theory Reference for ANSYS and ANSYS Workbench* for more information.

2.5.2.11. EDP CAP Model

The cap model is a subset of the **TB**,EDP data table operations. It is used for modeling the geomaterial plasticity that results from compaction at low mean stresses, followed by significant pre-failure dilation, but before shear failure occurs. The following table lists the coefficients that are addressed for the CAP model. If CAP flow potential is not defined, ANSYS assumes an associative CAP model.

CAP Model Constants*	TBOPT = CYFU Yielding Function (NPTS = 11)	TBOPT = CFPOT Flow Potential (NPTS = 4)
C1	R_c^Y	R_c^F
C2	R_t^Y	R_t^F
C3	X_i	β^F
C4	σ_i	α^F
C5	β^Y	
C6	A	
C7	α^Y	
C8	ψ	
C9	W_1^c	
C10	D_1^c	
C11	D_2^c	

See CAP Model in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on the constant conventions.

You can input the cohesion-related shear hardening constant σ_i by using **TB**,MISO, **TB**,BISO, **TB**,NLISO, or **TB**,PLAS. This input regulates the relationship between the cohesion and the effective deviatoric plastic strain.

Note

Calibrating the CAP constants σ_i, β^Y, A, α^Y, β^F, α^F and the hardening input for σ_i differs significantly from the other EDP options. The CAP parameters are all defined in relation to I_1 and J_2, while the other EDP coefficients are defined according to p and q.

2.5.2.12. Gurson's Model

Gurson's Model represents plasticity and damage in ductile and porous metals. A TB data table is used to handle the data for both stress and strain controlled nucleation, both with and without coalescence. Specify a *TB0PT* value in the **TB**, GURSON command to correspond to the Gurson basic model (*TB0PT* = BASE), strain controlled nucleation (*TB0PT* = SNNU), stress controlled nucleation (*TB0PT* = SSNU), or coalescence (*TB0PT* = COAL). The following table lists the coefficient values that are addressed for the available *TB0PT* values.

Gurson Model Constants	*TB0PT* = BASE	*TB0PT* = SNNU	*TB0PT* = SSNU	*TB0PT* = COAL
	Basic Model (*NPTS* = 5)	**Strain-Controlled Nucleation (*NPTS* = 3)**	**Stress-Controlled Nucleation (*NPTS* = 3)**	**Coalescence (*NPTS* = 2)**
C1	σ_Y - initial yielding strength	f_N - nucleation porosity	f_N - nucleation porosity	f_c - critical porosity
C2	f_o - initial porosity	ε_N - mean strain	σ_N - mean stress	f_F - failure porosity
C3	q_1 - first Tvergaard-Needleman	S_N - standard strain deviation	S_N^{σ} - mean stress deviation	
C4	q_2 - second Tvergaard-Needleman constant			
C5	q_3 - third Tvergaard-Needleman constant			

You can combine certain *TB0PT* values to address specific model behaviors. The following combinations are valid:

BASE and SNNU
BASE and SSNU
BASE and SNNU and COAL
BASE and SSNU and COAL
BASE and COAL

Although all three constants (q_1, q_2, q_3) are independent input parameters, setting $q_3 = q_1^2$ is recommended, unless specific data is available.

Isotropic hardening can be modeled using **TB**, BISO, **TB**, MISO, **TB**, NLISO or **TB**, PLASTIC.

2.5.2.13. Anand's Model

This option (ANAND) has input consisting of 9 constants. The Anand model is applicable to viscoplastic elements VISCO106, VISCO107, and VISCO108. See the *Theory Reference for ANSYS and ANSYS Workbench* for details. Initialize the constant table with **TB**,ANAND. You can define up to nine constants (C1-C9) with **TBDATA** commands (6 per command):

Constant	Meaning	Material Property	Units
C1	s_o	initial value of deformation resistance	stress
C2	Q/R	Q = activation energy R = universal gas constant	energy /volume energy /(volume temp)
C3	A	pre-exponential factor	1 / time
C4	xi	multiplier of stress	dimensionless
C5	m	strain rate sensitivity of stress	dimensionless
C6	h_o	hardening / softening constant	stress
C7	$\hat{S}$	coefficient for deformation resistance saturation value	stress
C8	n	strain rate sensitivity of saturation (deformation resistance) value	dimensionless
C9	a	strain rate sensitivity of hardening or softening	dimensionless

See Viscoplasticity in the *Structural Analysis Guide* for more information on this material option.

2.5.2.14. Multilinear Elastic

This option (MELAS) is such that unloading occurs along the same path as loading. This behavior, unlike the other options, is conservative (path-independent). The plastic strain (ε_{pl}) for this option should be interpreted as a "pseudo plastic strain" since it returns to zero when the material is unloaded (no hysteresis). See the *Theory Reference for ANSYS and ANSYS Workbench* for details. The material behavior is described by a piece-wise linear stress-strain curve, starting at the origin, with positive stress and strain values. The curve is continuous from the origin through 100 (max) stress-strain points. Successive slopes can be greater than the preceding slope; however, no slope can be greater than the elastic modulus of the material. The slope of the first curve segment usually corresponds to the elastic modulus of the material, although the elastic modulus can be input as greater than the first slope to ensure that all slopes are less than or equal to the elastic modulus.

Specify up to 20 temperature-dependent stress-strain curves. Initialize the curves with **TB**,MELAS. The temperature for the first curve is input with **TBTEMP**, followed by **TBPT** commands for up to 100 stress-strain points (the origin stress-strain point is not input). You can define up to 20 temperature- dependent stress-strain curves (*NTEMP* = 20 max on the **TB** command) in this manner. The constants (X, Y) entered on **TBPT** (two per command) are:

Constant	Meaning
X	Strain value (Dimensionless)
Y	Corresponding stress value (Force/Area)

See the **TB** command for a listing of the elements that can be used with this material option.

See Multilinear Elasticity in the *Structural Analysis Guide* for more information on this material option.

2.5.2.15. Cast Iron Plasticity

The cast iron plasticity option uses a composite yield surface to describe the different behavior in tension and compression. In tension the yielding is pressure-dependent and the Rankine maximum stress criterion is used. In compression, the behavior is pressure independent and the Mises yield criterion is used. A modified Mises potential is used as the flow potential. The elastic behavior is isotropic and is the same in tension and compression. Cast Iron Plasticity with isotropic hardening is intended for monotonic loading only and cannot be combined with any other material model.

Initiate the cast iron material model with **TB**,CAST. Activate the stress-strain table in tension using **TB**, UNIAXIAL with the *TENSION* option, then enter the stress-strain relation using the **TBPT** command. Activate the stress-strain table in compression using the**TB**, UNIAXIAL with the *COMPRESSION* option, then enter the stress-strain relation using the **TBPT** command. The slope of the stress-strain curve is assumed to be zero beyond the last user-defined stress-strain data point.

The **NROPT**,UNSYM command should be used at the solution level as the flow rule is not associated and the material Jacobian matrix is unsymmetric.

Initialize the database with **TB**,CAST. For each set of data, define the temperature using **TBTEMP**, then define the constant C1.

Constant	Meaning
C1	Plastic Poisson's ratio in tension

See the **TB** command description for a listing of the elements that can be used with this material option.

See Plastic Material Options in the *Structural Analysis Guide* for more information on this material option.

2.5.2.16. User

The **User Defined** (USER) material option describes input parameters for defining a material model based on either of two subroutines, which are ANSYS user-programmable features (see the *Guide to ANSYS User Programmable Features*). The choice of which subroutine to use is based on which element you are using.

The USER option works with the USERMAT subroutine in defining any material model (except incompressible materials), when you use any of the following elements: LINK180, SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, and SHELL209.

The USER option works with the USERPL subroutine in defining plasticity or viscoplasticity material models, when you use any of the following elements: LINK1, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95.

The USER option's input is determined by user-defined constants. The number of constants can be any combination of the number of temperatures (*NTEMP*) and the number of data points per temperature (*NPTS*), to a maximum limit of *NTEMP* x *NPTS* = 1000. Initialize the constant table with **TB**,USER. The constants are defined with **TBDATA** commands (6 per command).

State variables can also be used in the USERMAT subroutine (*not* in USERPL). To use state variables, initialize the constant table with **TB**,STATE then define the constants with the **TBDATA** command. You can define a maximum of 1000 state variables (*NPTS* = 1000). See User Defined Material in the *Structural Analysis Guide* for more information on this material option.

2.5.3. Hyperelastic Material Constants

Hyperelasticity is listed in the Special Features section of the Input Summary for elements SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, and SHELL209. The options described in the following sections are available to describe the material behavior for these elements. As described in these sections, you choose the option using *TBOPT* with **TB**,HYPER. Several forms of strain energy potentials are used to describe the hyperelasticity of materials. These are based on either strain invariants or principal stretches. The behavior of materials is assumed to be incompressible or nearly incompressible.

2.5.3.1. Neo-Hookean Hyperelastic Material Constants

The option, **TB**,HYPER,,,,NEO uses the Neo-Hookean form of strain energy potential, which is given by:

$$W = \frac{\mu}{2}(\bar{I}_1 - 3) + \frac{1}{d}(J-1)^2$$

where:

W = strain energy per unit reference volume

$\bar{I}_1$ = first deviatoric strain invariant

μ = initial shear modulus of the material

d = material incompressibility parameter.

J = determinant of the elastic deformation gradient **F**

The initial bulk modulus is defined by:

$$K = \frac{2}{d}$$

The constants μ and d are defined using the **TBDATA** command.

See the **TB** command for a listing of the elements that can be used with this material option.

See Neo-Hookean Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.2. Anisotropic Hyperelastic Material Constants

Anisotropic hyperelasticity is available with SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, and SHELL209. Anisotropic hyperelasticity is a potential-based-function with parameters to define the volumetric part, the isochoric part and the material directions.

You can use anisotropic hyperelasticity to model elastomers with reinforcements, and for biomedical materials such as muscles or arteries.

The strain energy potential for anisotropic hyperelasticity is given by

$$W = W_v(J) + W_d(\bar{\mathbf{C}}, \mathbf{A} \otimes \mathbf{A}, \mathbf{B} \otimes \mathbf{B})$$

Where:

$$W_v(J) = \frac{1}{d} \cdot (J-1)^2$$

and

$$W_d(\mathbf{C}, \mathbf{A} \otimes \mathbf{A}, \mathbf{B} \otimes \mathbf{B}) = \sum_{i=1}^{3} a_i(\bar{I}_1 - 3)^i + \sum_{j=1}^{3} b_j(\bar{I}_2 - 3)^j + \sum_{k=2}^{6} c_k(\bar{I}_4 - 1)^k$$
$$+ \sum_{l=2}^{6} d_l(\bar{I}_5 - 1)^l + \sum_{m=2}^{6} e_m(\bar{I}_6 - 1)^m + \sum_{n-2}^{6} f_n(\bar{I}_7 - 1)^n + \sum_{o=2}^{6} g_o(\bar{I}_8 - \varsigma)^o$$

Use **TB**,AHYPER,, *TB0PT* to define the isochoric part, material directions and the volumetric part. Only one TB table can be defined for each option.

TB0PT	**Constants**	**Purpose**	**Input Format**
POLY	C1 to C31	Anisotropic strain energy potential	`TB,AHYPER,,,POLY` `TBDATA,,A1,A2,A3,B1....`
AVEC	C1 to C3	Material direction constants	`TB,AHYPER,,,AVEC` `TBDATA,,A1,A2,A3`
BVEC	C1 to C3	Material direction constants	`TB,AHYPER,,,BVEC` `TBDATA,,B1,B2, B3`
PVOL	C1	Volumetric potential	`TB,AHYPER,,,POLY` `TBDATA,,D`

You can enter temperature dependent data for anisotropic hyperelastic material with the **TBTEMP** command. For the first temperature curve, you issue **TB**, AHYPER,,, *TB0PT*, then input the first temperature using the **TBTEMP** command. The subsequent **TBDATA** command inputs the data.

ANSYS interpolates the temperature data to the material points automatically using linear interpolation. When the temperature is out of the specified range, the closest temperature point is used.

See the **TB** command, and Anisotropic Hyperelasticity in the *Theory Reference for ANSYS and ANSYS Workbench* for more information.

2.5.3.3. Mooney-Rivlin Hyperelastic Material Constants (TB,HYPER)

Note that this section applies to the Mooney-Rivlin model with elements SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, and SHELL209.

This option, **TB**,HYPER,,,,MOONEY allows you to define 2, 3, 5, or 9 parameter Mooney-Rivlin models using *NPTS* = 2, 3, 5, or 9, respectively.

For *NPTS* = 2 (2 parameter Mooney-Rivlin option, which is also the default), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + \frac{1}{d}(J-1)^2$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

$\bar{I}_2$ = second deviatoric strain invariant

c_{10}, c_{01} = material constants characterizing the deviatoric deformation of the material
d = material incompressibility parameter

The initial shear modulus is defined as:

$$\mu = 2(c_{10} + c_{01})$$

and the initial bulk modulus is defined as:

$$K = \frac{2}{d}$$

where:

$d = (1 - 2*\nu) / (C_{10} + C_{01})$

The constants c_{10}, c_{01}, and d are defined by C1, C2, and C3 using the **TBDATA** command.

For *NPTS* = 3 (3 parameter Mooney-Rivlin option, which is also the default), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + \frac{1}{d}(J-1)^2$$

The constants c_{10}, c_{01}, c_{11}; and d are defined by C1, C2, C3, and C4 using the **TBDATA** command.

For *NPTS* = 5 (5 parameter Mooney-Rivlin option), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{20}(\bar{I}_1 - 3)^2 + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + c_{02}(\bar{I}_1 - 3)^2 + \frac{1}{d}(J-1)^2$$

The constants c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, and d are material constants defined by C1, C2, C3, C4, C5, and C6 using the **TBDATA** command.

For *NPTS* = 9 (9 parameter Mooney-Rivlin option), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{20}(\bar{I}_1 - 3)^2 + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + c_{02}(\bar{I}_2 - 3)^2 + c_{30}(\bar{I}_1 - 3)^3 + c_{21}(\bar{I}_1 - 3)^2(\bar{I}_2 - 3) + c_{12}(\bar{I}_1 - 3)(\bar{I}_2 - 3)^2 + c_{03}(\bar{I}_2 - 3)^3 + \frac{1}{d}(J-1)^2$$

The constants c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, c_{30}, c_{21}, c_{12}, c_{03}, and d are material constants defined by C1, C2, C3, C4, C5, C6, C7, C8, C9, and C10 using the **TBDATA** command.

See Mooney-Rivlin Hyperelastic Option (TB,HYPER) in the *Structural Analysis Guide* for more information on this material option.

2.5.3.4. Polynomial Form Hyperelastic Material Constants

The option, **TB**,HYPER,,,,POLY allows you to define a polynomial form of strain energy potential. The form of the strain energy potential for the Polynomial option is given by:

$$W = \sum_{i+j=1}^{N} c_{ij}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + \sum_{k=1}^{N} \frac{1}{d_k}(J-1)^{2k}$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

$\bar{I}_2$ = second deviatoric strain invariant

J = determinant of the elastic deformation gradient **F**

N, c_{ij}, and d = material constants

In general there is no limitation on the value of N in ANSYS (see the **TB** command). A higher value of N can provide a better fit to the exact solution. It may however cause a numerical difficulty in fitting the material constants, and it also requests enough data to cover the whole range of deformation for which you may be interested. For these reasons, a very high value of N is not recommended.

The initial shear modulus μ is defined by:

$$\mu = 2(c_{10} + c_{01})$$

and the initial bulk modulus is defined as:

$$K = \frac{2}{d_1}$$

For N = 1 and $c_{01} = 0$, the polynomial form option is equivalent to the Neo-Hookean option. For N = 1, it is equivalent to the 2 parameter Mooney-Rivlin option. For N = 2, it is equivalent to the 5 parameter Mooney-Rivlin option, and for N = 3, it is equivalent to the 9 parameter Mooney-Rivlin option.

The constants c_{ij} and d are defined using the **TBDATA** command in the following order:

For N (*NPTS*) = 1:

c_{10}, c_{01}, d_1

For N (*NPTS*) = 2:

$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, d_1, d_2$

For N (*NPTS*) = 3:

$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, c_{30}, c_{21}, c_{12}, c_{03}, d_1, d_2, d_3$

For N (*NPTS*) = k:

$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, c_{30}, c_{21}, c_{12}, c_{03}, ..., c_{k0}, c_{(k-1)1}, ..., c_{0k}, d_1, d_2, ..., d_k$

See the **TB** command for a listing of the elements that can be used with this material option.

See Polynomial Form Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.5. Ogden Hyperelastic Material Constants

This option, **TB**,HYPER,,,,,OGDEN uses the Ogden form of strain energy potential. The Ogden form is based on the principal stretches of the left Cauchy-Green tensor. The strain energy potential is:

$$W = \sum_{i=1}^{N} \frac{\mu_i}{\alpha_i}(\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i} - 3) + \sum_{k=1}^{N} \frac{1}{d_k}(J-1)^{2k}$$

where:

W = strain energy potential

$\bar{\lambda}_p$ (p = 1,2,3) = deviatoric principal stretches, defined as $\bar{\lambda}_p = J^{-\frac{1}{3}}\lambda_p$

λ_p = principal stretches of the left Cauchy-Green tensor
J = determinant of the elastic deformation gradient
N, μ_p, α_p and d_p = material constants

In general there is no limitation on the value of N in ANSYS (see the **TB** command). A higher value of N can provide a better fit to the exact solution. It may however cause numerical difficulties in fitting the material constants. For this reason, very high values of N are not recommended.

The initial shear modulus μ is defined by:

$$\mu = \frac{1}{2}\sum_{i=1}^{N} \alpha_i \mu_i$$

The initial bulk modulus K is defined by:

$$K = \frac{2}{d_1}$$

For N = 1 and $\alpha_1 = 2$, the Ogden option is equivalent to the Neo-Hookean option. For N = 2, $\alpha_1 = 2$, and $\alpha_2 = -2$, the Ogden option is equivalent to the 2 parameter Mooney-Rivlin option.

The constants μ_p, α_p and d_p are defined using the **TBDATA** command in the following order:

For N (*NPTS*) = 1:

μ_1, α_1, d_1

For N (*NPTS*) = 2:

$\mu_1, \alpha_1, \mu_2, \alpha_2, d_1, d_2$

For N (*NPTS*) = 3:

$\mu_1, \alpha_1, \mu_2, \alpha_2, \mu_3, \alpha_3, d_1, d_2, d_3$

For N (*NPTS*) = k:

$\mu_1, \alpha_1, \mu_2, \alpha_2, ..., \mu_k, \alpha_k, d_1, d_2, ..., d_k$

See the **TB** command for a listing of the elements that can be used with this material option.

See Ogden Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.6. Arruda-Boyce Hyperelastic Material Constants

This option, **TB**,HYPER,,,,BOYCE uses the Arruda-Boyce form of strain energy potential, given by:

$$W = \mu\left[\frac{1}{2}(\bar{I}_1 - 3) + \frac{1}{20\lambda_L^2}(\bar{I}_1^2 - 9) + \frac{11}{1050\lambda_L^4}(\bar{I}_1^3 - 27) + \frac{19}{7000\lambda_L^6}(\bar{I}_1^4 - 81) + \frac{519}{673750\lambda_L^8}(\bar{I}_1^5 - 243)\right] + \frac{1}{d}\left(\frac{J^2 - 1}{2} - \ln J\right)$$

where:

W = strain energy per unit reference volume

$\bar{I}_1$ = first deviatoric strain invariant

J = determinant of the elastic deformation gradient ***F***

μ = initial shear modulus of materials

λ_L = limiting network stretch

d = material incompressibility parameter

The initial bulk modulus is defined as:

$$K = \frac{2}{d}$$

As λ_L approaches infinity, the option becomes equivalent to the Neo-Hookean option.

The constants μ, λ_L and d are defined by C1, C2, and C3 using the **TBDATA** command.

See the **TB** command for a listing of the elements that can be used with this material option.

See Arruda-Boyce Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.7. Gent Hyperelastic Material Constants

This option, **TB**,HYPER,,,,GENT uses the Gent form of strain energy potential, given by:

$$W = \frac{\mu J_m}{2} \ln\left(1 - \frac{\bar{I}_1 - 3}{J_m}\right)^{-1} + \frac{1}{d}\left(\frac{J^2 - 1}{2} - \ln J\right)$$

where:

W = strain energy per unit reference volume
μ = initial shear modulus of material
J_m = limiting value of $\bar{I}_1 - 3$, $\bar{I}_1$
$\bar{I}_1$ = first deviatoric strain invariant
J = determinant of the elastic deformation gradient ***F***
d = material incompressibility parameter

The initial bulk modulus K is defined as:

$$K = \frac{2}{d}$$

As J_m approaches infinity, the option becomes equivalent to the Neo-Hookean option.

The constants μ, J_m, and d are defined by C1, C2, and C3 using the **TBDATA** command.

See the **TB** command for a listing of the elements that can be used with this material option.

See Gent Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.8. Yeoh Hyperelastic Material Constants

The option, **TB**,HYPER,,,,YEOH follows a reduced polynomial form of strain energy potential by Yeoh. The form of the strain energy potential for the Yeoh option is given by:

$$W = \sum_{i=1}^{N} c_{i0} \, (\bar{I}_1 - 3)^i + \sum_{k=1}^{N} \frac{1}{d_k}(J-1)^{2k}$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

J = determinant of the elastic deformation gradient **F**

N, c_{i0}, and d_k = material constants

In general there is no limitation on the value of N in ANSYS (see the **TB** command). A higher value of N can provide a better fit to the exact solution. It may however cause a numerical difficulty in fitting the material constants, and it also requests enough data to cover the whole range of deformation for which you may be interested. For these reasons, a very high value of N is not recommended.

The initial shear modulus μ is defined by:

$$\mu = 2c_{10}$$

and the initial bulk modulus K is defined as:

$$K = \frac{2}{d_1}$$

For N = 1 the Yeoh form option is equivalent to the Neo-Hookean option.

The constants c_{i0} and d_k are defined using the **TBDATA** command in the following order:

For N (*NPTS*) = 1:

c_{10}, d_1

For N (*NPTS*) = 2:

c_{10}, c_{20}, d_1, d_2

For N (*NPTS*) = 3:

$c_{10}, c_{20}, c_{30}, d_1, d_2, d_3$

For N (*NPTS*) = k:

$c_{10}, c_{20}, c_{30}, ..., c_{k0}, d_1, d_2, ..., d_k$

See the **TB** command for a listing of the elements that can be used with this material option.

See Yeoh Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.9. Blatz-Ko Foam Hyperelastic Material Constants

This option, **TB**,HYPER,,,,BLATZ uses the Blatz-Ko form of strain energy potential, given by:

$$W = \frac{\mu}{2}\left(\frac{I_2}{I_3} + 2\sqrt{I_3} - 5\right)$$

where:

W = strain energy per unit reference volume
μ = initial strain shear modulus
I_2 and I_3= second and third strain invariants

The initial bulk modulus k is defined as:

$$k = \frac{5}{3}\mu$$

The model has only one constant μ and is defined by C1 using the **TBDATA** command.

See the **TB** command for a listing of the elements that can be used with this material option.

See Blatz-Ko Foam Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.10. Ogden Compressible Foam Hyperelastic Material Constants

This option, **TB**,HYPER,,,,FOAM uses the Ogden form of strain energy potential for highly compressible elastomeric foam material. The strain energy potential is based on the principal stretches of the left Cauchy-Green tensor and is given by:

$$W = \sum_{i=1}^{N} \frac{\mu_i}{\alpha_i}(J^{\alpha_i/3}(\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i}) - 3) + \sum_{i=1}^{N} \frac{\mu_i}{\alpha_i\beta_i}(J^{-\alpha_i\beta_i} - 1)$$

where:

W = strain energy potential

$\bar{\lambda}_p^{\alpha_i}$ (p=1,2,3) = deviatoric principal stretch

J = determinant of the elastic deformation gradient
N, μ_i, α_i and β_k = material constants

For this material option, the volumetric and deviatoric terms are tightly coupled. Hence, this model is meant to simulate highly compressible elastomers.

In general there is no limitation on the value of N in ANSYS (see the **TB** command). A higher value of N can provide a better fit to the exact solution. It may however cause numerical difficulties in fitting the material constants. For this reason, very high values of N are not recommended.

The initial shear modulus μ is defined by:

$$\mu = \frac{\sum_{i=1}^{N} \mu_i\alpha_i}{2}$$

and the initial bulk modulus K is defined by:

$$K = \sum_{i=1}^{N} \mu_i\alpha_i \left(\frac{1}{3} + \beta_i\right)$$

For N = 1, $\alpha_1 = -2$, $\mu_1 = -\mu$, and $\beta_1 = 0.5$, the Ogden foam option is equivalent to the Blatz-Ko option.

The constants μ_i, α_i and β_i are defined using the **TBDATA** command in the following order:

For N (*NPTS*) = 1:

μ_1, α_1, β_1

For N (*NPTS*) = 2:

$\mu_1, \alpha_1, \mu_2, \alpha_2, \beta_1, \beta_2$

For N (*NPTS*) = 3:

$\mu_1, \alpha_1, \mu_2, \alpha_2, \mu_3, \alpha_3, \beta_1, \beta_2, \beta_3$

For N (*NPTS*) = k:

$\mu_1, \alpha_1, \mu_2, \alpha_2, ..., \mu_k, \alpha_k, \beta_1, \beta_2, ..., \beta_k$

See the **TB** command for a listing of the elements that can be used with this material option.

See Ogden Compressible Foam Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.3.11. User-Defined Hyperelastic Material

You can define a strain energy potential by using the option **TB**,HYPER,,,,USER. This allows you to provide a subroutine USERHYPER to define the derivatives of the strain energy potential with respect to the strain invariants. Refer to the *Guide to ANSYS User Programmable Features* for a detailed description on writing a user hyperelasticity subroutine.

See the **TB** command for a listing of the elements that can be used with this material option.

See User-Defined Hyperelastic Option in the *Structural Analysis Guide* for more information on this material option.

2.5.4. Viscoelastic Material Constants

The viscoelastic material model is available with the viscoelastic elements VISCO88 and VISCO89 for small deformation viscoelasticity and LINK180, SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, and SHELL209 for small and large deformation viscoelasticity.

Elements VISCO88 and VISCO89 use a viscoelastic material model that is defined by entering the following data in the data table with **TB** commands. Data not input are assumed to be zero. You must enter the data table to perform the viscoelastic computation. A generalized Maxwell model is used to represent the material characteristics. See the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of terms. Initialize the constant table with **TB**,EVISC. You can define up to 95 constants (C1-C95) with **TBDATA** commands (6 per command):

Constant	Meaning
1	Shift Function Constant 1 Value of C5. C5 = 0 H/R (activation energy divided by ideal constant R). C5 = 1 WLF constant C1.
2	Shift Function Constant 2 depending on C5. C5 = 0 Value of Constant x ($0 \leq x \leq 1$). C5 = 1 WLF constant C2.

Constant	Meaning
3	No. of Maxwell elements (10 max) in volume decay function M_V.
4	Shift Function Constant 3 depending on C5. C5 = 1 WLF reference temperature.
5	Shift Function Key. C5 = 0, Tool-Narayanaswamy Shift Function (applicable to glass). C5 = 1, Williams-Landau-Ferry Shift Function (applicable to polymers). C5 = 11, User Subroutine for Fictive Temperature/Shift Function (UsrFictive.F). C5 = 20, User Subroutine for Viscoelasticity (UsrViscEl.F).
6-15	Up to ten values of C_{fi} (coefficients of the Maxwell element representing the volume decay function M_V). Used to define the fictive temperature. $\Sigma C_{fi} = 1.0$)
16-25	Up to ten values of τ_{fi} (constants associated with a discrete relaxation spectrum). Used to define the fictive temperature. Each τ_{fi} is also known as a relaxation time.
26-30	Up to five values of C_{li} (coefficients of thermal expansion for the liquid state). ($\alpha_l = C_{l1} + C_{l2} T_f + C_{l3} T_f^2 + C_{l4} T_f^3 + C_{l5} T_f^4$, where T_f = fictive temperature)
31-35	Up to five values of C_{gi} (coefficients of thermal expansion for the glass state). ($\alpha_g = C_{g1} + C_{g2}T + C_{g3}T^2 + C_{g4}T^3 + C_{g5}T^4$, where T = actual temperature)
36-45	Up to ten values of T_{fi} (fictive temperature). $T_f = \Sigma C_{fi} T_{fi}$
46	GXY(0) (shear modulus at time = zero (the full shear modulus)).
47	GXY(∞) (shear modulus at time = infinity (the residual shear modulus after the full decay)). If no relaxation of the shear modulus, use GXY(∞) = GXY(0).
48	K(0) (bulk modulus at time = zero).
49	K(∞) (bulk modulus at time = infinity). If no relaxation of the bulk modulus, use K(∞) = K(0).
50	No. of Maxwell elements (10 max) used to approximate the shear modulus (GXY(0) - GXY(∞)) relaxation.
51-60	Up to ten values of C_{smi} (coefficients for shear modulus relaxation using Maxwell elements, $\Sigma C_{smi} = 1.0$ if shear modulus relaxes).
61-70	Up to ten values of λ_{smi} (relaxation times for shear modulus relaxation using Maxwell elements).
71	No. of Maxwell elements (10 max) used to approximate the bulk modulus (K(0) - K(∞)) relaxation.
76-85	Up to ten values of C_{bmi} (coefficients for bulk modulus relaxation using Maxwell elements, $\Sigma C_{bmi} = 1.0$ if bulk modulus relaxes).
86-95	Up to ten values of λ_{bmi} (relaxation times for bulk modulus relaxation using Maxwell elements).

The viscoelasticity input for SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, and SHELL209 consists of elasticity properties and relaxation properties. The underlying elasticity is specified by either the **MP** command (for hypoelasticity) or by the **TB**,HYPER command (for hypere-

lasticity). For LINK180, BEAM188, and BEAM189, the underlying elasticity is specified by the **MP** command (hypo-elasticity) only. Use the **TB**,PRONY or **TB**,SHIFT commands to input the relaxation properties. Enter the required data using the **TBDATA** command using the following constants.

For **TB**,PRONY:

Constant	Meaning
C1	Relative modulus
C2	Relative time

For **TB**,SHIFT (*Tbopt* = WLF):

The William-Landel-Ferry shift function, A, takes the form:

$$\log_{10}(A) = \frac{C_2(T - C_1)}{C_3 + T - C_1}$$

Constant	Meaning
C1	Relative temperature (T_r)
C2-C3	WLF constants

For **TB**,SHIFT (*Tbopt* = TN):

The Tool-Narayanaswamy shift function, A, takes the form:

$$A = \exp\left(C_2\left(\frac{1}{C_1} - \frac{1}{T}\right)\right)$$

Constant	Meaning
C1	Relative temperature (T_r)
C2	TN constant

See Viscoelasticity in the *Structural Analysis Guide* for more information.

2.5.5. Magnetic Materials

Elements with magnetic capability use the **TB** table to input points characterizing B-H curves. See the *Theory Reference for ANSYS and ANSYS Workbench* for details. These curves are available in elements SOLID5, PLANE13, PLANE53, SOLID62, SOLID96, and SOLID98. Temperature-dependent curves cannot be input. Initialize the curves with **TB**,BH. Use **TBPT** commands to define up to 500 points (H, B). The constants (X, Y) entered on **TBPT** (two per command) are:

Constant	Meaning
X	Magnetic field intensity (H) (Magnetomotive force/length)
Y	Corresponding magnetic flux density (B) (Flux/Area)

Specify the system of units (MKS or user defined) with **EMUNIT**, which also determines the value of the permeability of free space. Free-space permeability is available in elements SOLID5, INFIN9, PLANE13, INFIN47, PLANE53, SOLID62, SOLID96, SOLID97, SOLID98, INFIN110, and INFIN111. This value is used with the relative permeability property values [**MP**] to establish absolute permeability values. The defaults (also obtained for Lab = MKS) are

MKS units and free-space permeability of 4 πE-7 Henries/meter. You can specify Lab = MUZRO to define any system of units, then input free-space permeability.

See Additional Guidelines for Defining Regional Material Properties and Real Constants in the *Low-Frequency Electromagnetic Analysis Guide* for more information on this material option.

2.5.6. High-Frequency Electromagnetic Materials

High-frequency elements HF119 and HF120 use the **TB** table to specify the following material properties:

- **TB**, CNDE - anisotropic electric current conductivity (mhos/m)(1/Ωm)
- **TB**, CNDM - anisotropic magnetic current conductivity (ohms/m)
- **TB**, DPER - anisotropic relative permittivity
- **TB**, MUR - anisotropic relative permeability

For electric current conductivity (CNDE), magnetic current conductivity (CNDM), relative permittivity (DPER), and relative permeability (**TB**,MUR,*MAT*,,,*TB0PT* with *TB0PT* = 0 – input a permeability matrix), a 3 x 3 matrix relates terms ordered by x, y, and z.

$$[X] = \begin{bmatrix} X_{11} & X_{12} & X_{13} \\ X_{21} & X_{22} & X_{23} \\ X_{31} & X_{32} & X_{33} \end{bmatrix}$$

The constants (C1-C9) entered on the **TBDATA** command are:

Constant	Meaning
C1-C9	$X_{11}, X_{22}, X_{33}, X_{12}, X_{23}, X_{13}, X_{21}, X_{32}, X_{31}$

If X_{ij} is 0 where i and j are indexes, then X_{ji} must also be zero.

For **TB**, DPER and **TB**,MUR the diagonal elements cannot be zero.

TB,MUR,MAT,,,TBOPT with TBOPT = 1 or 2 generates a B-H nonlinear material permeability matrix with a uniform or non-uniform dc magnetic field, respectively. A non-uniform dc magnetic field can be the solution of an ANSYS magneto static analysis.

The matrix is give by the following equations for a static magnetic field (H_o) in the x, y, and z directions, respectively.

$$[\dot{\mu}_r] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \mu_r & -j\kappa \\ 0 & j\kappa & \mu_r \end{bmatrix} \text{ (For } H_o \text{ in x-direction)}$$

$$[\dot{\mu}_r] = \begin{bmatrix} \mu_r & 0 & j\kappa \\ 0 & 1 & 0 \\ -j\kappa & 0 & \mu_r \end{bmatrix} \text{ (For } H_o \text{ in y-direction)}$$

$$[\dot{\mu}_r] = \begin{bmatrix} \mu_r & -j\kappa & 0 \\ j\kappa & \mu_r & 0 \\ 0 & 0 & 1 \end{bmatrix} \text{ (For } H_o \text{ in z-direction)}$$

where:

$$\mu_r = 1 + \frac{\omega_m(\omega_0 + j\omega\alpha)}{(\omega_0 + j\omega\alpha)^2 - \omega^2}$$

$$\kappa = \frac{\omega_m\omega}{(\omega_0 + j\omega\alpha)^2 - \omega^2}$$

$$\omega_0 = \gamma\mu_0 H_o$$

$$\omega_m = \gamma\mu_0 M_s$$

$$\alpha = \gamma\mu_0 \Delta H / 2\omega$$

$$\gamma = \gamma_0 g / 2$$

and

- γ is the gyromagnetic ratio
- γ_0 is the electron gyromagnetic ratio
- g is the Lande factor
- H_o is the static magnetic field in the z, y, or x direction
- M_s is the saturation magnetization introduced by H_o
- ΔH is the resonance line width
- ω is the working angular frequency

The constants (C1-C7) entered on **TBDATA** are:

Constant	**Meaning**
C1	Saturation magnetization $4\pi M_s$ (Gauss) (no default).
C2	Lande g-factor (1.8 to 2.5, defaults to 2.0).
C3	Resonance line width ΔH (Oe) (defaults to 0).
C4	Internal dc magnetic field H_o (Oe) (no default).
C5	Direction of H_o. 0 - z direction (default) 1 - y direction (default) 2 - x direction (default)
C6	Sign of off-diagonal element of permeability matrix. $C6 \geq 0$, $[\dot{\mu}_r] = \begin{bmatrix} \mu_r & -j\kappa & 0 \\ j\kappa & \mu_r & 0 \\ 0 & 0 & 1 \end{bmatrix}$ (default)

Constant	Meaning
	C6 < 0, $[\dot{\mu}_r] = \begin{bmatrix} \mu_r & j\kappa & 0 \\ -j\kappa & \mu_r & 0 \\ 0 & 0 & 1 \end{bmatrix}$ (default)
C7	Symmetry key for permeability matrix. 0 – antisymmetric matrix (default) 1 – symmetric matrix

The high-frequency 2-D modal analysis element HF118 uses the **TB** table to specify the following material properties:

- **TB**, DPER – anisotropic relative permittivity
- **TB**,MUR,*MAT*,,,*TB0PT* with *TB0PT* = 0 – anisotropic relative permeability

The element must lie in an X-Y plane and a 3 x 3 matrix relates terms ordered by x, y, and z.

$$[X] = \begin{bmatrix} X_{xx} & Xxy & 0 \\ Xyx & X_{yy} & 0 \\ 0 & 0 & X_{zz} \end{bmatrix}$$

The constants (C1-C9) entered on the **TBDATA** command are:

Constant	Meaning
C1-C9	$X_{xx}, X_{yy}, X_{zz}, X_{xy}, 0, 0, X_{yx}, 0, 0$

Note

HF118 can not use the **TB** table to specify the following:

- Anisotropic electric current conductivity (**TB**,CNDE)
- Anisotropic magnetic current conductivity (**TB**,CNDM)
- B-H nonlinear material permeability matrix with a uniform or non-uniform dc magnetic field (**TB**,MUR,MAT,,,TBOPT with TBOPT = 1 or 2).

2.5.7. Anisotropic Elastic Materials

Anisotropic elastic capability is available with the PLANE182, SOLID185, PLANE183, SOLID186, SOLID187, and SOLSH190 structural elements (see *Section 2.5.2.7: Anisotropic*) and the SOLID5, PLANE13, SOLID98, PLANE223, SOLID226, and SOLID227 coupled-field elements. Input the elastic coefficient matrix [D] either by specifying the stiffness constants (EX, EY, etc.) with **MP** commands, or by specifying the terms of the matrix with data table commands as described below. The matrix should be symmetric and positive definite (requiring all determinants to be positive).

The full 6 x 6 elastic coefficient matrix [D] relates terms ordered x, y, z, xy, yz, xz via 21 constants as shown below.

$$\begin{bmatrix} D_{11} & & & & & \\ D_{21} & D_{22} & & & & \\ D_{31} & D_{32} & D_{33} & \text{Symmetric} & & \\ D_{41} & D_{42} & D_{43} & D_{44} & & \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

For 2-D problems, a 4 x 4 matrix relates terms ordered x, y, z, xy via 10 constants (D_{11}, D_{21}, D_{22}, D_{31}, D_{32}, D_{33}, D_{41}, D_{42}, D_{43}, D_{44}). Note, the order of the vector is expected as {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the "D" matrix terms to be converted to the expected format. The "D" matrix can be defined in either "stiffness" form (with units of Force/Area operating on the strain vector) or in "compliance" form (with units of the inverse of Force/Area operating on the stress vector), whichever is more convenient. Select a form using *TB0PT* on the **TB** command. Both forms use the same data table input as described below.

Enter the constants of the elastic coefficient matrix in the data table with the **TB** commands. Initialize the constant table with **TB**,ANEL. Define the temperature with **TBTEMP**, followed by up to 21 constants input with **TBDATA** commands. The matrix may be input in either stiffness or flexibility form, based on the *TB0PT* value. For the coupled-field elements, temperature- dependent matrix terms are not allowed. You can define up to six temperature-dependent sets of constants (*NTEMP* = 6 max on the **TB** command) in this manner. Matrix terms are linearly interpolated between temperature points. The constants (C1-C21) entered on **TBDATA** (6 per command) are:

Constant	**Meaning**
C1-C6	Terms D_{11}, D_{21}, D_{31}, D_{41}, D_{51}, D_{61}
C7-C12	Terms D_{22}, D_{32}, D_{42}, D_{52}, D_{62}, D_{33}
C13-C18	Terms D_{43}, D_{53}, D_{63}, D_{44}, D_{54}, D_{64}
C19-C21	Terms D_{55}, D_{65}, D_{66}

See the **TB** command for a listing of the elements that can be used with this material option.

2.5.8. Piezoelectric Materials

Piezoelectric capability is available with the SOLID5, PLANE13, SOLID98, PLANE223, SOLID226, and SOLID227 coupled-field elements. SOLID5, PLANE13, and SOLID98 have this capability in the ANSYS Multiphysics and ANSYS Mechanical products; PLANE223, SOLID226, and SOLID227 have this capability in the ANSYS Multiphysics product. Material properties required for the piezoelectric effects include the dielectric (relative permittivity) constants, the elastic coefficient matrix, and the piezoelectric matrix.

Input the dielectric constants either by specifying orthotropic dielectric permittivity (PERX, PERY, PERZ) on the **MP** command or by specifying the terms of the anisotropic permittivity matrix [ε] on the **TB**,DPER command. The values input on the **MP** command will be interpreted as permittivity at constant strain [ε^S]. Using **TB**,DPER, you can specify either permittivity at constant strain [ε^S] (*TB0PT* = 0), or permittivity at constant stress [ε^T] (*TB0PT* = 1).

Input the elastic coefficient matrix [c] either by specifying the stiffness constants (EX, EY, etc.) with **MP** commands, or by specifying the terms of the anisotropic elasticity matrix with **TB** commands as described in *Section 2.5.2.7: Anisotropic*.

You can define the piezoelectric matrix in [e] form (piezoelectric stress matrix) or in [d] form (piezoelectric strain matrix). The [e] matrix is typically associated with the input of the anisotropic elasticity in the form of the stiffness matrix [c], and the permittivity at constant strain [ε^S]. The [d] matrix is associated with the input of compliance

matrix [s] and permittivity at constant stress [ε^T]. Select the appropriate matrix form for your analysis using the **TB**,PIEZ command.

The full 6 x 3 piezoelectric matrix relates terms x, y, z, xy, yz, xz to x, y, z via 18 constants as shown:

$$\begin{bmatrix} e_{11} & e_{12} & e_{13} \\ e_{21} & e_{22} & e_{23} \\ e_{31} & e_{32} & e_{33} \\ e_{41} & e_{42} & e_{43} \\ e_{51} & e_{52} & e_{53} \\ e_{61} & e_{62} & e_{63} \end{bmatrix}$$

For 2-D problems, a 4 x 2 matrix relates terms ordered x, y, z, xy via 8 constants ($e_{11}, e_{12}, e_{21}, e_{22}, e_{31}, e_{32}, e_{41}, e_{42}$). The order of the vector is expected as {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the piezoelectric matrix terms to be converted to the expected format.

Use the **TB** commands to enter the constants of the piezoelectric matrix in the data table. Initialize the constant table with **TB**,PIEZ. You can define up to 18 constants (C1-C18) with **TBDATA** commands (6 per command):

Constant	Meaning
C1-C6	Terms $e_{11}, e_{12}, e_{13}, e_{21}, e_{22}, e_{23}$
C7-C12	Terms $e_{31}, e_{32}, e_{33}, e_{41}, e_{42}, e_{43}$
C13-C18	Terms $e_{51}, e_{52}, e_{53}, e_{61}, e_{62}, e_{63}$

See Piezoelectric Analysis in the *Coupled-Field Analysis Guide* for more information on this material model.

2.5.9. Piezoresistive Materials

Elements with piezoresistive capabilities (PLANE223, SOLID226, SOLID227) use **TB**,PZRS to calculate the change in electric resistivity produced by elastic stress or strain. Material properties required to model piezoresistive materials are electrical resistivity, the elastic coefficient matrix, and the piezoresistive matrix.

You can define the piezoresistive matrix either in the form of piezoresistive stress matrix [π] (*TBOPT* = 0) or piezoresistive strain matrix [m] (*TBOPT* = 1).

The piezoresistive stress matrix [π] uses stress to calculate the change in electric resistivity due to piezoresistive effect, while the piezoresistive strain matrix [m] (*TBOPT* = 1) uses strain to calculate the change in electric resistivity. See *Section 11.5: Piezoresistivity* in the *Theory Reference for ANSYS and ANSYS Workbench* for more information.

The full 6x6 piezoresistive matrix relates the x, y, z, xy, yz, xz terms of stress to the x, y, z, xy, yz, xz terms of electric resistivity via 36 constants:

$$\begin{bmatrix} \pi_{11} & \pi_{12} & \pi_{13} & \pi_{14} & \pi_{15} & \pi_{16} \\ \pi_{21} & \pi_{22} & \pi_{23} & \pi_{24} & \pi_{25} & \pi_{26} \\ \pi_{31} & \pi_{32} & \pi_{33} & \pi_{34} & \pi_{35} & \pi_{36} \\ \pi_{41} & \pi_{42} & \pi_{43} & \pi_{44} & \pi_{45} & \pi_{46} \\ \pi_{51} & \pi_{52} & \pi_{53} & \pi_{54} & \pi_{55} & \pi_{56} \\ \pi_{61} & \pi_{62} & \pi_{63} & \pi_{64} & \pi_{65} & \pi_{66} \end{bmatrix}$$

Constant	Meaning
C1-C6	Terms $\pi_{11}, \pi_{12}, \pi_{13}, \pi_{14}, \pi_{15}, \pi_{16}$
C7-C12	Terms $\pi_{21}, \pi_{22}, \pi_{23}, \pi_{24}, \pi_{25}, \pi_{26}$
C13-C18	Terms $\pi_{31}, \pi_{32}, \pi_{33}, \pi_{34}, \pi_{35}, \pi_{36}$
C19-C24	Terms $\pi_{41}, \pi_{42}, \pi_{43}, \pi_{44}, \pi_{45}, \pi_{46}$
C25-C30	Terms $\pi_{51}, \pi_{52}, \pi_{53}, \pi_{54}, \pi_{55}, \pi_{56}$
C31-C36	Terms $\pi_{61}, \pi_{62}, \pi_{63}, \pi_{64}, \pi_{65}, \pi_{66}$

For 2-D problems, a 4x4 matrix relates terms ordered x, y, z, xy via 16 constants.

Constant	Meaning
C1-C4	Terms $\pi_{11}, \pi_{12}, \pi_{13}, \pi_{14}$
C7-C10	Terms $\pi_{21}, \pi_{22}, \pi_{23}, \pi_{24}$
C13-C16	Terms $\pi_{31}, \pi_{32}, \pi_{33}, \pi_{34}$
C19-C22	Terms $\pi_{41}, \pi_{42}, \pi_{43}, \pi_{44}$

The order of the vector is expected as {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the piezoresistive matrix terms to be converted to the expected format.

See Piezoresistive Analysis in the *Coupled-Field Analysis Guide* for more information on this material model.

2.5.10. Anisotropic Electric Permittivity Materials

Elements with piezoelectric capabilities (PLANE223, SOLID226, SOLID227) use **TB**,DPER to specify anisotropic relative electric permittivity. You can define electric permittivity at constant strain [ε^S] (*TB0PT* = 0) or constant stress [ε^T] (*TB0PT* = 1)

Note

ANSYS will convert matrix [ε^T] to [ε^S] using piezoelectric strain and stress matrices.

The full 3x3 electric permittivity matrix relates x, y, z components of electric field to the x, y, z components of electric flux density via 6 constants:

$$\begin{bmatrix} \varepsilon_{11} & \varepsilon_{12} & \varepsilon_{13} \\ & \varepsilon_{22} & \varepsilon_{23} \\ \text{sym} & & \varepsilon_{33} \end{bmatrix}$$

Constant	Meaning
C1-C6	$\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}, \varepsilon_{12}, \varepsilon_{23}, \varepsilon_{13}$

For 2-D problems, a 2x2 matrix relates terms ordered x, y via 3 constants (ε_{11} ε_{22} ε_{12}):

Constant	Meaning
C1, C2, C4	$\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{12}$

2.5.11. Rate-Dependent Plastic (Viscoplastic) Materials

The RATE option, when combined with other material options, defines the strain rate dependency of isotropic plasticity. To simulate viscoplasticity, you combine the RATE option with the BISO, MISO, or NLISO options. To

simulate anisotropic viscoplasticity, you combine the RATE option and the HILL option with the BISO, MISO, or NLISO options. See *Section 2.6: Material Model Combinations* for further information. The RATE option is applicable to elements: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, SHELL281, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, and SHELL209.

There are two models available for use with the RATE option, the Perzyna model or the Peirce model. You specify the model using *TB0PT* in the form **TB**,RATE,,,,PERZYNA or **TB**,RATE,,,,PEIRCE. Each of these models is described below.

The Perzyna model has the following form:

$$\sigma = \left[1 + \left(\frac{\dot{\varepsilon}^{pl}}{\gamma}\right)^{m}\right]\sigma_0$$

and the Peirce model has the following form:

$$\sigma = \left[1 + \frac{\dot{\varepsilon}^{pl}}{\gamma}\right]^{m}\sigma_0$$

In both cases σ is the material yield stress, $\dot{\varepsilon}^{pl}$ is the equivalent plastic strain rate, m is the strain rate hardening parameter, γ is the material viscosity parameter, and σ_0 is the static yield stress of material. σ_0 is a function of some hardening parameter and can be defined by isotropic plasticity (for example, **TB**,BISO). As γ approaches ∞, or m approaches zero, or $\dot{\varepsilon}^{pl}$ approaches zero, the solution approaches the static (rate-independent) solution.

When m is very small, the Peirce model has less difficulty converging, compared to the Perzyna model. See the *Theory Reference for ANSYS and ANSYS Workbench* for details.

The two constants for either model that are defined by **TBDATA** are:

Constant	Meaning
C1	m - material strain rate hardening parameter
C2	γ - material viscosity parameter

Initialize the data table with **TB**,RATE, and specify the model option using *TB0PT*. For each set of data, define the temperature [**TBTEMP**], then define material constants C1 and C2 [**TBDATA**]. The data table command for the combination option must also be defined for the same material number to specify the static hardening behavior of the materials (rate-independent and isotropic).

See Viscoplasticity in the *Structural Analysis Guide* for more information on this material option.

2.5.12. Gasket Materials

The GASKET option allows you to simulate gasket joints with the ANSYS interface elements INTER192, INTER193, INTER194, and INTER195. The gasket material is usually under compression and is highly nonlinear. The material also exhibits quite complicated unloading behavior when compression is released. The GASKET option allows you to define some general parameters including the initial gap, stable stiffness for numerical stabilization, and stress cap for a gasket in tension. The GASKET option also allows you to directly input data for the experimentally measured complex pressure closure curves for the gaskets. The GASKET option also offers two sub-options to define gasket unloading behavior including linear and nonlinear unloading. The linear unloading option simplifies

the input by defining the starting closure at the compression curves and the slope. The nonlinear unloading option allows you to directly input unloading curves to more accurately model the gasket unloading behavior. When no unloading curves are defined, the material behavior follows the compression curve while it is unloaded.

You enter the general parameters and the pressure closure behavior data using the *TB0PT* field when issuing **TB**,GASKET. You then input the material data using either the **TBDATA** command or the **TBPT** command as shown in the table below that describes the various gasket data types and presents the command input format.

You can enter temperature dependent data using the **TBTEMP** command for any of the gasket data types. For the first temperature curve, you issue **TB**,GASKET,,,,*TB0PT*, then input the first temperature using **TBTEMP**, followed by the data using either **TBDATA** or **TBPT** depending on the value of *TB0PT* as shown in the table. ANSYS automatically interpolates the temperature data to the material points using linear interpolation. When the temperature is out of the specified range, the closest temperature point is used.

Gasket Data Type	*TB0PT*	Constants	Meaning	Input Format
General parameters	PARA	C1	Initial gap (default = 0, meaning there is no initial gap).	`TB,GASKET,,,,PARA` `TBDATA,1,C1,C2,C3`
		C2	Stable stiffness (default = 0, meaning there is no stable stiffness. [*1*]	
		C3	Maximum tension stress allowed when the gasket material is in tension (default = 0, meaning there is no tension stress in the gasket material).	

Compression load closure curve	COMP	Xi	Closure value.	`TB,GASKET,,,2,COMP`
				`TBPT,,X1,Y1`
				`TBPT,,X2,Y2`

		Yi	Pressure value.	

Linear unloading data	LUNL	Xi	Closure value on compression curve where unloading started.	`TB,GASKET,,,2,LUNL`
				`TBPT,,X1,Y1`
				`TBPT,,X2,Y2`

		Yi	Unloading slope value.	

Nonlinear unloading data [*2*]	NUNL	Xi	Closure value.	`TB,GASKET,,,2,NUNL`
				`TBPT,,X1,Y1`
				`TBPT,,X2,Y2`

		Yi	Pressure value.	
Transverse shear	TSS	XY, XZ	Transverse shear values	`TB,GASKET,,,2,TSS` `TBDATA,1,TSSXY,TSSXZ`

1. Stable stiffness is used for numerical stabilization such as the case when the gasket is opened up and thus no stiffness is contributed to the element nodes, which in turn may cause numerical difficulty.
2. Multiple curves may be required to define the complex nonlinear unloading behavior of a gasket material.

   When there are several nonlinear unloading curves defined, ANSYS requires that the starting point of each unloading curve be on the compression curve to ensure the gasket unloading behavior is correctly simulated. Though it is not a requirement that the temperature dependency of unloading data be the same as the compression data, when there is a missing temperature, ANSYS uses linear interpolation to obtain the material data of the missing temperature. This may result in a mismatch between the compression data and the unloading data. Therefore, it is generally recommended that the number of temperatures and temperature points be the same for each unloading curve and compression curve.

When using the material GUI to enter data for the nonlinear unloading curves, an indicator at the top of the dialog box states the number of the unloading curve whose data is currently displayed along with the total number of unloading curves defined for the particular material (example: **Curve number 2/5**). To enter data for the multiple unloading curves, type the data for the first unloading curve, then click on the **Add Curve** button and type the data for the second curve. Repeat this procedure for entering data for the remaining curves. Click the **Del Curve** button if you want to remove the curve whose data is currently displayed. Click the **>** button to view the data for the next curve in the sequence, or click the **<** button to view the data for the previous curve in the sequence. To insert a curve at a particular location in the sequence, click on the **>** or **<** buttons to move to the curve before the insertion location point and click on the **Add Curve** button. For example, if the data for **Curve number 2/5** is currently displayed and you click on the **Add Curve** button, the dialog box changes to allow you to enter data for **Curve number 3/6**. You can define a total of 100 nonlinear unloading curves per material.

For a more detailed description of the gasket joint simulation capability in ANSYS, see the Gasket Joints Simulation chapter in the *Structural Analysis Guide*.

2.5.13. Creep Equations

If Table 4.*n*-1 lists "creep" as a "Special Feature," then the element can model creep behavior.

The creep strain rate, $\dot{\varepsilon}_{cr}$, can be a function of stress, strain, temperature, and neutron flux level. Libraries of creep strain rate equations are included under the *Section 2.5.13.1: Implicit Creep Equations* and *Section 2.5.13.2: Explicit Creep Equations* sections. Enter the constants shown in these equations using **TB**,CREEP and **TBDATA** as described below. These equations (expressed in incremental form) are characteristic of materials being used in creep design applications (see the *Theory Reference for ANSYS and ANSYS Workbench* for details).

Three different types of creep equations are available:

- Primary creep
- Secondary creep
- Irradiation induced creep

You can define the combined effects of more than one type of creep using the implicit equations specified by *TB0PT* = 11 or 12, the explicit equations, or a user-defined creep equation.

ANSYS analyzes creep using the implicit and the explicit time integration method. The implicit method is robust, fast, accurate, and recommended for general use, especially with problems involving large creep strain and large deformation. It has provisions for including temperature-dependent constants. ANSYS can model pure creep, creep with isotropic hardening plasticity, and creep with kinematic hardening plasticity, using both von Mises and Hill potentials. See *Section 2.6: Material Model Combinations* for further information. Since the creep and plasticity are modeled simultaneously (no superposition), the implicit method is more accurate and efficient than the explicit method. Temperature dependency can also be incorporated by the Arrhenius function (see the *Theory Reference for ANSYS and ANSYS Workbench* for details).

The explicit method is useful for cases involving very small time steps, such as in transient analyses. There are no provisions for temperature-dependent constants, nor simultaneous modeling of creep with any other material models such as plasticity. However, there is temperature dependency using the Arrhenius function, and you can combine explicit creep with other plasticity options using non-simultaneous modeling (superposition). In these cases, ANSYS first performs the plastic analysis, then the creep calculation.

Note

The terms "implicit" and "explicit" as applied to creep, have no relationship to "explicit dynamics," or any elements referred to as "explicit elements."

2.5.13.1. Implicit Creep Equations

Enter an implicit creep equation using *TB0PT* within the **TB** command. Enter the value of *TB0PT* corresponding to the equation, as shown in *Table 2.4, "Implicit Creep Equations"*.

Table 2.4 Implicit Creep Equations

Creep Model (*TB0PT*)	Name	Equation		Type
1	Strain Hardening	$\dot{\varepsilon}_{cr} = C_1\sigma^{C2}\varepsilon_{cr}^{C3}e^{-C4/T}$	$C_1>0$	Primary
2	Time Hardening	$\dot{\varepsilon}_{cr} = C_1\sigma^{C2}t^{C3}e^{-C4/T}$	$C_1>0$	Primary
3	Generalized Exponential	$\dot{\varepsilon}_{cr} = C_1\sigma^{C2}re^{-rt}$, $r = C_5\sigma^{C3}e^{-C4/T}$	$C_1>0$, $C_5>0$	Primary
4	Generalized Graham	$\dot{\varepsilon}_{cr} = C_1\sigma^{C_2}(t^{C_3} + C_4t^{C_5} + C_6t^{C_7})e^{-C_8/T}$	$C_1>0$	Primary
5	Generalized Blackburn	$\dot{\varepsilon}_{cr} = f(1-e^{-rt}) + gt$ $f = C_1e^{C_2\sigma}$, $r = C_3(\sigma/C_4)^{C_5}$, $g = C_6e^{C_7\sigma}$	$C_1>0$, $C_3>0$, $C_6>0$	Primary
6	Modified Time Hardening	$\varepsilon_{cr} = C_1\sigma^{C_2}t^{C_3+1}e^{-C_4/T}/(C_3+1)$	$C_1>0$	Primary
7	Modified Strain Hardening	$\dot{\varepsilon}_{cr} = \{C_1\sigma^{C_2}[(C_3+1)\varepsilon_{cr}]^{C_3}\}^{1/(C_3+1)}e^{-C4/T}$	$C_1>0$	Primary
8	Generalized Garofalo	$\dot{\varepsilon}_{cr} = C_1[\sinh(C_2\sigma)]^{C_3}e^{-C_4/T}$	$C_1>0$	Secondary
9	Exponential form	$\dot{\varepsilon}_{cr} = C_1e^{\sigma/C_2}e^{-C_3/T}$	$C_1>0$	Secondary
10	Norton	$\dot{\varepsilon}_{cr} = C_1\sigma^{C_2}e^{-C_3/T}$	$C_1>0$	Secondary
11	Time Hardening	$\varepsilon_{cr} = C_1\sigma^{C_2}t^{C_3+1}e^{-C_4/T}/(C_3+1) + C_5\sigma^{C_6}te^{-C_7/T}$	$C_1>0$, $C_5>0$	Primary + Secondary
12	Rational polynomial	$\dot{\varepsilon}_{cr} = C_1\frac{\partial\varepsilon_c}{\partial t}$, $\varepsilon_c = \frac{cpt}{1+pt} + \dot{\varepsilon}_m t$ $\dot{\varepsilon}_m = C_2 10^{C_3\sigma}\sigma^{C_4}$ $c = C_7\dot{\varepsilon}_m^{C_8}\sigma^{C_9}$, $p = C_{10}\dot{\varepsilon}_m^{C_{11}}\sigma^{C_{12}}$	$C_2>0$	Primary + Secondary
13	Generalized Time Hardening	$\varepsilon_{cr} = f\,t^r\,e^{-C_6/T}$ $f = C_1\sigma + C_2\sigma^2 + C_3\sigma^3$ $r = C_4 + C_5\sigma$		Primary
100		User Creep		

where:

ε_{cr} = equivalent creep strain

$\dot{\varepsilon}_{cr}$ = change in equivalent creep strain with respect to time

σ = equivalent stress

T = temperature (absolute). The offset temperature (from **TOFFST**), is internally added to all temperatures for convenience.

C_1 through C_{12} = constants defined by the **TBDATA** command

t = time at end of substep

e = natural logarithm base

You can define the user creep option by setting *TB0PT* = 100, and using **TB**,STATE to specify the number of state variables for the user creep routine. See the *Guide to ANSYS User Programmable Features* for more information. The **RATE** command is necessary to activate implicit creep for specific elements (see the **RATE** command description for details). The **RATE** command has no effect for explicit creep.

For temperature dependent constants, define the temperature using **TBTEMP** for each set of data. Then, define constants C1 through C*m* using **TBDATA** (where *m* is the number of constants, and depends on the creep model you choose).

The following example shows how you would define the implicit creep model represented by *TB0PT* = 1 at two temperature points.

```
TB,CREEP,1,,,1           !Activate creep data table, specify creep model 1
TBTEMP,100               !Define first temperature
TBDATA,1,c11,c12,c13,c14 !Creep constants c11, c12, c13, c14 at first temp.
TBTEMP,200               !Define second temperature
TBDATA,1,c21,c22,c23,c24 !Creep constants c21, c22, c23, c24 at second temp.
```

Coefficients are linearly interpolated for temperatures that fall between user defined **TBTEMP** values. For some creep models, where the change in coefficients spans several orders of magnitude, this linear interpolation might introduce inaccuracies in solution results. Be sure to use enough curves to accurately capture the temperature dependency. Also, consider using the curve fitting routine to calculate a temperature dependent coefficient that includes the Arrhenius term.

See the **TB** command for a listing of the elements that can be used with this material option.

See Creep in the *Structural Analysis Guide* for more information on this material option.

2.5.13.2. Explicit Creep Equations

Enter an explicit creep equation by setting *TB0PT* = 0 (or leaving it blank) within the **TB** command, then specifying the constants associated with the creep equations using the **TBDATA** command.

Specify primary creep with constant C_6. *Section 2.5.13.2.1: Primary Explicit Creep Equation for C6 = 0*, through *Section 2.5.13.2.11: Primary Explicit Creep Equation for C6 = 100*, show the available equations. You select an equation with the appropriate value of C_6 (0 to 15). If $C_1 \leq 0$, or if $T + T_{offset} \leq 0$, no primary creep is computed.

Specify secondary creep with constant C_{12}. *Section 2.5.13.2.12: Secondary Explicit Creep Equation for C12 = 0* and *Section 2.5.13.2.13: Secondary Explicit Creep Equation for C12 = 1* show the available equations. You select an equation with the appropriate value of C_{12} (0 or 1). If $C_7 \leq 0$, or if $T + T_{offset} \leq 0$, no secondary creep is computed. Also, primary creep equations C_6 = 9, 10, 11, 13, 14, and 15 bypass any secondary creep equations since secondary effects are included in the primary part.

Specify irradiation induced creep with constant C_{66}. *Section 2.5.13.2.14: Irradiation Induced Explicit Creep Equation for C66 = 5* shows the single equation currently available; select it with C_{66} = 5. This equation can be used in conjunction with equations C_6 = 0 to 11. The constants should be entered into the data table as indicated by their subscripts. If $C_{55} \leq 0$ and $C_{61} \leq 0$, or if $T + T_{offset} \leq 0$, no irradiation induced creep is computed.

A linear stepping function is used to calculate the change in the creep strain within a time step ($\Delta \varepsilon_{cr} = (\dot{\varepsilon}_{cr})(\Delta t)$). The creep strain rate is evaluated at the condition corresponding to the beginning of the time interval and is assumed to remain constant over the time interval. If the time step is less than 1.0e-6, then no creep strain increment is computed. Primary equivalent stresses and strains are used to evaluate the creep strain rate. For highly nonlinear creep strain vs. time curves, use a small time step if you are using the explicit creep algorithm. A creep time step optimization procedure is available for automatically increasing the time step whenever possible. A nonlinear stepping function (based on an exponential decay) is also available (C_{11} = 1) but should be used with caution since it can underestimate the total creep strain where primary stresses dominate. This function is available only for creep equations C_6 = 0, 1 and 2. Temperatures used in the creep equations should be based on an absolute scale [**TOFFST**].

Use the **BF** or **BFE** commands to enter temperature and fluence values. The input fluence (Φ_t) includes the integrated effect of time and time explicitly input is not used in the fluence calculation. Also, for the usual case of a constant flux (Φ), the fluence should be linearly ramp changed.

Temperature dependent creep constants are not permitted for explicit creep. You can incorporate other creep options by setting *C6* = 100. See the *Guide to ANSYS User Programmable Features* for more information.

The following example shows how you would use the explicit creep equation defined by *C6* = 1.

```
TB,CREEP,1              !Activate creep data table
TBDATA,1,c1,c2,c3,c4,,1  !Creep constants c1, c2, c3, c4 for equation C6=1
```

The explicit creep constants that you enter with the **TBDATA** are:

Constant	Meaning
C1-CN	Constants C_1, C_2, C_3, etc. (as defined in *Section 2.5.13.2.1: Primary Explicit Creep Equation for C6 = 0* to *Section 2.5.13.2.14: Irradiation Induced Explicit Creep Equation for C66 = 5*) These are obtained by curve fitting test results for your material to the equation you choose. Exceptions are defined below.

2.5.13.2.1. Primary Explicit Creep Equation for C6 = 0

$$\dot{\varepsilon}_{cr} = C_1 \sigma^{C2} \varepsilon_{cr}^{C3} e^{-C4/T}$$

where:

$\dot{\varepsilon}$ = change in equivalent strain with respect to time
σ = equivalent stress
T = temperature (absolute). The offset temperature (from **TOFFST**) is internally added to all temperatures for convenience.
t = time at end of substep
e = natural logarithm base

2.5.13.2.2. Primary Explicit Creep Equation for C6 = 1

$$\dot{\varepsilon}_{cr} = C_1 \sigma^{C2} t^{C3} e^{-C4/T}$$

2.5.13.2.3. Primary Explicit Creep Equation for C6 = 2

$$\dot{\varepsilon}_{cr} = C_1 \sigma^{C2} r e^{-rt}$$

where:

$$r = C_5 \sigma^{C3} e^{-C4/T}$$

2.5.13.2.4. Primary Explicit Creep Equation for C6 = 9

Annealed 304 Stainless Steel:

$$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$$

2.5.13.2.4.1. Double Exponential Creep Equation (C4 = 0)

To use the following Double Exponential creep equation to calculate

$$\varepsilon_c = \varepsilon_x (1 - e^{-st}) + \varepsilon_t (1 - e^{-rt}) + \dot{\varepsilon}_m t$$

where:

$\varepsilon_x = 0$ for $\sigma \leq C_2$
$\varepsilon_x = G + H\sigma$ for $C_2 < \sigma \leq C_3$
$C_2 = 6000$ psi (default), $C_3 = 25000$ psi (default)

s, r, $\dot{\varepsilon}_m$, G, and H = functions of temperature and stress as described in the reference.

This double exponential equation is valid for Annealed 304 Stainless Steel over a temperature range from 800 to 1100°F. The equation, known as the Blackburn creep equation when $C_1 = 1$, is described completely in the *1.*. The first two terms describe the primary creep strain and the last term describes the secondary creep strain.

To use this equation, input a nonzero value for C_1, $C_6 = 9.0$, and $C_7 = 0.0$. Temperatures should be in °R (or °F with $T_{offset} = 460.0$). Conversion to °K for the built-in property tables is done internally. If the temperature is below the valid range, no creep is computed. Time should be in hours and stress in psi. The valid stress range is 6,000 - 25,000 psi.

2.5.13.2.4.2. Rational Polynomial Creep Equation with Metric Units (C4 = 1)

To use the following standard Rational Polynomial creep equation (with metric units) to calculate ε_c, enter $C_4 = 1.0$:

$$\varepsilon_c + \frac{cpt}{1 + pt} + \dot{\varepsilon}_m t$$

where:

c = limiting value of primary creep strain
p = primary creep time factor

$\dot{\varepsilon}_m$ = secondary (minimum) creep strain rate

This standard rational polynomial creep equation is valid for Annealed 304 SS over a temperature range from 427°C to 704°C. The equation is described completely in the *1.*. The first term describes the primary creep strain.

The last term describes the secondary creep strain. The average "lot constant" is used to calculate $\dot{\varepsilon}_m$.

To use this equation, input $C_1 = 1.0$, $C_4 = 1.0$, $C_6 = 9.0$, and $C_7 = 0.0$. Temperature must be in °C and T_{offset} must be 273 (because of the built-in property tables). If the temperature is below the valid range, no creep is computed. Also, time must be in hours and stress in Megapascals (MPa).

Various hardening rules governing the rate of change of creep strain during load reversal may be selected with the C_5 value: 0.0 - time hardening, 1.0 - total creep strain hardening, 2.0 - primary creep strain hardening. These options are available only with the standard rational polynomial creep equation.

2.5.13.2.4.3. Rational Polynomial Creep Equation with English Units (C4 = 2)

To use the above standard Rational Polynomial creep equation (with English units), enter $C_4 = 2.0$.

This standard rational polynomial equation is the same as described above except that temperature must be in °F, T_{offset} must be 460, and stress must be in psi. The equivalent valid temperature range is 800 - 1300°F.

2.5.13.2.5. Primary Explicit Creep Equation for C6 = 10

Annealed 316 Stainless Steel:

$$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$$

2.5.13.2.5.1. Double Exponential Creep Equation (C4 = 0)

To use the same form of the Double Exponential creep equation as described for Annealed 304 SS ($C_6 = 9.0$, $C_4 = 0.0$) in *Section 2.5.13.2.4: Primary Explicit Creep Equation for C6 = 9* to calculate ε_c, enter $C_4 = 0.0$.

This equation, also described in *1.*, differs from the Annealed 304 SS equation in that the built-in property tables are for Annealed 316 SS, the valid stress range is 4000 - 30,000 psi, C_2 defaults to 4000 psi, C_3 defaults to 30,000 psi, and the equation is called with $C_6 = 10.0$ instead of $C_6 = 9.0$.

2.5.13.2.5.2. Rational Polynomial Creep Equation with Metric Units (C4 = 1)

To use the same form of the standard Rational Polynomial creep equation with *metric* units as described for Annealed 304 SS ($C_6 = 9.0$, $C_4 = 1.0$) in *Section 2.5.13.2.4: Primary Explicit Creep Equation for C6 = 9*, enter $C_4 = 1.0$.

This standard rational polynomial equation, also described in *1.*, differs from the Annealed 304 SS equation in that the built-in property tables are for Annealed 316 SS, the valid temperature range is 482 - 704°C, and the equation is called with $C_6 = 10.0$ instead of $C_6 = 9.0$. The hardening rules for load reversal described for the $C_6 = 9.0$ standard Rational Polynomial creep equation are also available. The average "lot constant" from *1.* is used in the calculation of $\dot{\varepsilon}_m$.

2.5.13.2.5.3. Rational Polynomial Creep Equation with English Units (C4 = 2)

To use the previous standard Rational Polynomial creep equation with *English* units, enter $C_4 = 2.0$.

This standard rational polynomial equation is the same as described above except that the temperatures must be in °F, T_{offset} must be 460, and the stress must be in psi (with a valid range from 0.0 to 24220 psi). The equivalent valid temperature range is 900 - 1300°F.

2.5.13.2.6. Primary Explicit Creep Equation for C6 = 11

Annealed 2 1/4 Cr - 1 Mo Low Alloy Steel:

$$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$$

2.5.13.2.6.1. Modified Rational Polynomial Creep Equation (C4 = 0)

To use the following Modified Rational Polynomial creep equation to calculate ε_c, enter $C_4 = 0.0$:

$$\varepsilon_c = \frac{t}{A + Bt} + \dot{\varepsilon}_m t$$

A, B, and $\dot{\varepsilon}_m$ are functions of temperature and stress as described in the reference.

This modified rational polynomial equation is valid for Annealed 2 1/4 Cr -1 Mo Low Alloy steel over a temperature range of 700 - 1100°F. The equation is described completely in the *2.*. The first term describes the primary creep strain and the last term describes the secondary creep strain. No modification is made for plastic strains.

To use this equation, input $C_1 = 1.0$, $C_6 = 11.0$, and $C_7 = 0.0$. Temperatures must be in °R (or °F with $T_{offset} = 460.0$). Conversion to °K for the built-in property tables is done internally. If the temperature is below the valid range, no creep is computed. Time should be in hours and stress in psi. Valid stress range is 1000 - 65,000 psi.

2.5.13.2.6.2. Rational Polynomial Creep Equation with Metric Units (C4 = 1)

To use the following standard Rational Polynomial creep equation (with *metric* units) to calculate ε_c, enter $C_4 = 1.0$:

$$\varepsilon_c + \frac{cpt}{1 + pt} + \dot{\varepsilon}_m t$$

where:

c = limiting value of primary creep strain
p = primary creep time factor
$\dot{\varepsilon}_m$ = secondary (minimum) creep strain rate

This standard rational polynomial creep equation is valid for Annealed 2 1/4 Cr - 1 Mo Low Alloy Steel over a temperature range from 371°C to 593°C. The equation is described completely in the *2.*. The first term describes the primary creep strain and the last term describes the secondary creep strain. No tertiary creep strain is calculated. Only Type I (and not Type II) creep is supported. No modification is made for plastic strains.

To use this equation, input $C_1 = 1.0$, $C_4 = 1.0$, $C_6 = 11.0$, and $C_7 = 0.0$. Temperatures must be in °C and T_{offset} must be 273 (because of the built-in property tables). If the temperature is below the valid range, no creep is computed. Also, time must be in hours and stress in Megapascals (MPa). The hardening rules for load reversal described for the $C_6 = 9.0$ standard Rational Polynomial creep equation are also available.

2.5.13.2.6.3. Rational Polynomial Creep Equation with English Units (C4 = 2)

To use the above standard Rational Polynomial creep equation with *English* units, enter $C_4 = 2.0$.

This standard rational polynomial equation is the same as described above except that temperatures must be in °F, T_{offset} must be 460, and stress must be in psi. The equivalent valid temperature range is 700 - 1100°F.

2.5.13.2.7. Primary Explicit Creep Equation for C6 = 12

$$\dot{\varepsilon}_{cr} = MK(C_1\sigma)^N t^{(M-1)}$$

where:

C_1 = Scaling constant
M, N, K = Function of temperature (determined by linear interpolation within table) as follows:

C_5	Number of temperature values to describe M, N, or K function (2 minimum, 6 maximum)
C_{49}	First absolute temperature value
C_{50}	Second absolute temperature value
...	
$C_{48} + C_5$	C_5th absolute temperature value
$C_{48} + C_5 + 1$	First M value
...	
$C_{48} + 2C_5$	C_5th M value
$C_{48} + 2C_5$	C_5th M value
...	
$C_{48} + 2C_5$	C_5th M value
$C_{48} + 2C_5 + 1$	First N value
...	
$C_{48} + 3C_5$	C_5th N value
$C_{48} + 3C_5 + 1$	First K value
...	

This power function creep law having temperature dependent coefficients is similar to Equation $C_6 = 1.0$ except with $C_1 = f_1(T)$, $C_2 = f_2(T)$, $C_3 = f_3(T)$, and $C_4 = 0$. Temperatures must not be input in decreasing order.

2.5.13.2.8. Primary Explicit Creep Equation for C6 Equals 13

Sterling Power Function:

$$\dot{\varepsilon}_{cr} = \frac{\varepsilon_{acc}}{B\varepsilon_{acc}^B \sigma^A 10^{(3A+2B+C)}}$$

where:

ε_{acc} = creep strain accumulated to this time (calculated by the program). Internally set to 1×10^{-5} at the first substep with nonzero time to prevent division by zero.
$A = C_1/T$
$B = C_2/T + C_3$

$C = C_4/T + C_5$

This equation is often referred to as the Sterling Power Function creep equation. Constant C_7 should be 0.0. Constant C_1 should not be 0.0, unless no creep is to be calculated.

2.5.13.2.9. Primary Explicit Creep Equation for C6 = 14

$$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$$

where:

$\varepsilon_c = cpt/(1+pt) + \dot{\varepsilon}_m$
$\ln c = -1.350 - 5620/T - 50.6 \times 10^{-6}\,\sigma + 1.918 \ln(\sigma/1000)$
$\ln p = 31.0 - 67310/T + 330.6 \times 10^{-6}\,\sigma - 1885.0 \times 10^{-12}\,\sigma^2$
$\ln \dot{\varepsilon}_m = 43.69 - 106400/T + 294.0 \times 10^{-6}\,\sigma + 2.596 \ln(\sigma/1000)$

This creep law is valid for Annealed 316 SS over a temperature range from 800°F to 1300°F. The equation is similar to that given for C_6 = 10.0 and is also described in *1.*.

To use equation, input C_1 = 1.0 and C_6 = 14.0. Temperatures should be in °R (or °F with T_{offset} = 460). Time should be in hours. Constants are only valid for English units (pounds and inches). Valid temperature range: 800° - 1300°F. Maximum stress allowed for e_c calculation: 45,000 psi; minimum stress: 0.0 psi. If $T + T_{offset} < 1160$, no creep is computed.

2.5.13.2.10. Primary Explicit Creep Equation for C6 = 15

General Material Rational Polynomial:

$$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$$

where:

$$\varepsilon_c + \frac{cpt}{1+pt} + \dot{\varepsilon}_m t$$

$\dot{\varepsilon}_m = C_2\, 10^{C_3 \sigma}\, \sigma^{C_4}$ (C_2 must not be negative)

$c = C_7 \dot{\varepsilon}_m^{C_8}\, \sigma^{C_9}$

$p = C_{10} \dot{\varepsilon}_m^{C_{11}}\, \sigma^{C_{12}}$

This rational polynomial creep equation is a generalized form of the standard rational polynomial equations given as C_6 = 9.0, 10.0, and 11.0 (C_4 = 1.0 and 2.0). This equation reduces to the standard equations for isothermal cases. The hardening rules for load reversal described for the C_6 = 9.0 standard Rational Polynomial creep equation are also available.

2.5.13.2.11. Primary Explicit Creep Equation for C6 = 100

A user-defined creep equation is used. See the *Guide to ANSYS User Programmable Features* for more information.

2.5.13.2.12. Secondary Explicit Creep Equation for C12 = 0

$$\dot{\varepsilon}_{cr} = C_7 e^{\sigma/C_8}\, e^{-C_{10}/T}$$

where:

σ = equivalent stress
T = temperature (absolute). The offset temperature (from **TOFFST**), is internally added to all temperatures for convenience.
t = time
e = natural logarithm base

2.5.13.2.13. Secondary Explicit Creep Equation for C12 = 1

$$\dot{\varepsilon}_{cr} = C_7 \sigma^{C_8}\, e^{-C_{10}/T}$$

2.5.13.2.14. Irradiation Induced Explicit Creep Equation for C66 = 5

$$\dot{\varepsilon}_{cr} = C_{55}\, \sigma \dot{\phi} e^{-\phi t_{0.5}/C_{56}} + C_{61} B \sigma \dot{\phi}$$

where:

$$B = FG + C_{63}$$

$$F = \frac{e^{-C_{58}/T}}{C_{59} + C_{60} e^{-C_{57}/T}}$$

$$G = 1 - e^{-\phi t_{0.5}/C_{62}} a$$

σ = equivalent stress
T = temperature (absolute). The offset temperature (from **TOFFST**) is internally added to all temperatures for convenience.
$\Phi_{t_{0.5}}$ = neutron fluence (input on **BF** or **BFE** command)
e = natural logarithm base
t = time

This irradiation induced creep equation is valid for 20% Cold Worked 316 SS over a temperature range from 700° to 1300°F. Constants 56, 57, 58 and 62 must be positive if the B term is included.

See the **TB** command for a listing of the elements that can be used with this material option.

See Creep in the *Structural Analysis Guide* for more information on this material option.

2.5.14. Shape Memory Alloys

This option (SMA) is used to model the superelastic behavior of shape memory alloys. Use this with the MP command to define the elastic behavior in the austenite state. The SMA model can be used with these elements: PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, and SOLSH190.

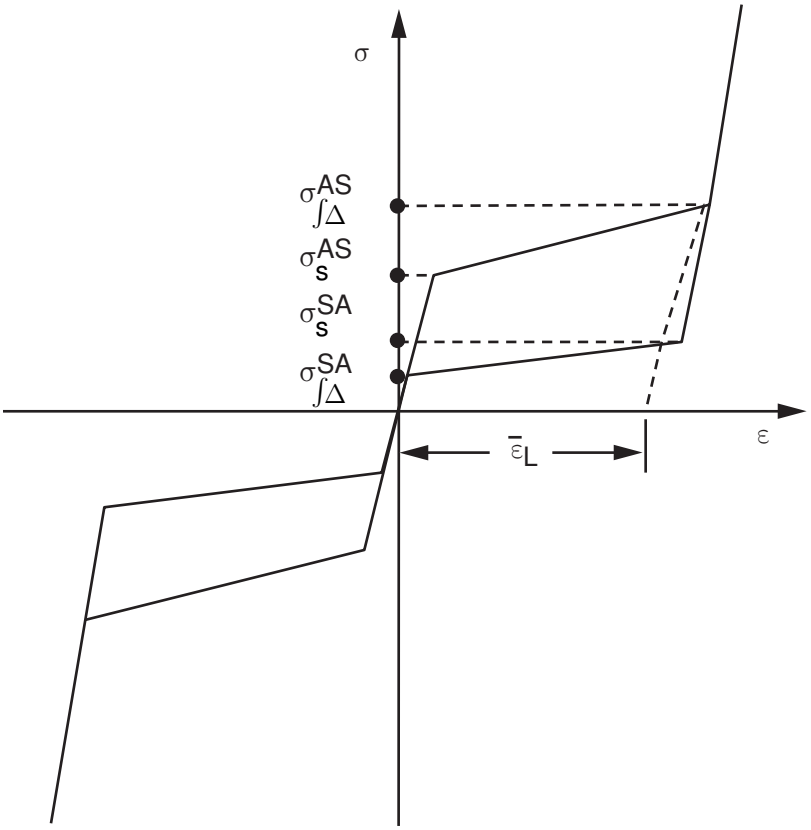
The SMA option is described by six constants that define the stress-strain behavior in loading and unloading for the uniaxial stress-state.

Initialize the data table with **TB**,SMA. For each data set, define the temperature using **TBTEMP**, then define constants C1 through C6 using **TBDATA**. You may define up to six sets of temperature-dependent constants in this manner. See *Section 8.4.1.7: Shape Memory Alloy* or more information and an example.

Table 2.5 Shape Memory Alloy Constants

Constant		Meaning
SIG-SAS (C1)	σ_s^{AS}	Starting stress value for the forward phase transformation
SIG-FAS (C2)	σ_f^{AS}	Final stress value for the forward phase transformation
SIG-SSA (C3)	σ_s^{SA}	Starting stress value for the reverse phase transformation
SIG-FSA (C4)	σ_f^{SA}	Final stress value for the reverse phase transformation
EPSILON (C5)	$\bar{e}_L$	Maximum residual strain
ALPHA (C6)	α	Parameter measuring the difference between material responses in tension and compression

Figure 2.1 Shape Memory Alloy Phases



2.5.15. Swelling Equations

If Table 4.*n*-1 lists "swelling" as a "Special Feature," then the element can model swelling behavior. Swelling is a material enlargement due to neutron bombardment and other effects (see the *Theory Reference for ANSYS and ANSYS Workbench*). The swelling strain rate may be a function of temperature, time, neutron flux level, and stress.

The fluence (which is the flux x time) is input on the **BF** or **BFE** command. A linear stepping function is used to calculate the change in the swelling strain within a load step:

$$\Delta\varepsilon_{sw}=\frac{d\varepsilon_{sw}}{d(\phi t)}(\Delta(\phi t))$$

where Φt is the fluence and the swelling strain rate equation is as defined in subroutine USERSW.

Because of the many empirical swelling equations available, the programming of the actual swelling equation is left to the user. In fact, the equation and the "fluence" input may be totally unrelated to nuclear swelling. See the *Guide to ANSYS User Programmable Features* for user programmable features.

For highly nonlinear swelling strain vs. fluence curves a small fluence step should be used. Note that since fluence (Φt), and not flux (Φ), is input, a constant flux requires that a linearly changing fluence be input if time is changing. Temperatures used in the swelling equations should be based on an absolute scale [**TOFFST**]. Temperature and fluence values are entered with the **BF** or **BFE** command. Swelling calculations for the current substep are based upon the previous substep results.

Initialize the swelling table with **TB**,SWELL. The constants entered on the **TBDATA** commands (6 per command) are:

Constant	Meaning
C1-CN	Constants C_1, C_2, C_3, etc. (as required by the user swelling equations). C_{72} must equal 10.

See the **TB** command for a listing of the elements that can be used with this material option.

See Swelling in the *Structural Analysis Guide* for more information on this material option.

2.5.16. MPC184 Joint Materials

The JOIN material option on the **TB** command allows you to impose linear and nonlinear elastic stiffness and damping behavior or hysteretic friction behavior on the available components of relative motion of an MPC184 joint element. The material behaviors described here apply to all joint elements except the weld, orient, and spherical joints.

Note that the **TB** command may be repeated with the same material ID number to specify both the stiffness and damping behavior.

2.5.16.1. Linear Elastic Stiffness and Damping Behavior

Input the linear stiffness or damping behavior for the relevant components of relative motion of a joint element by specifying the terms as part of a 6 x 6 matrix with data table commands as described below.

The 6 x 6 matrix for linear stiffness or damping behavior is as follows:

$$\begin{bmatrix} D_{11} & & & & & \\ D_{21} & D_{22} & & & & \\ D_{31} & D_{32} & D_{33} & & & \\ D_{41} & D_{42} & D_{43} & D_{44} & & \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

Enter the stiffness or damping coefficient of the matrix in the data table with **TB** set of commands. Initialize the constant table with **TB**,JOIN,,,STIF (for stiffness behavior) or **TB**,JOIN,,,DAMP (for damping behavior). Define the temperature with **TBTEMP**, followed by the relevant constants input with **TBDATA** commands. Matrix terms are linearly interpolated between temperature points. Based on the joint type, the relevant constant specification is as follows:

Joint Element	Constant	Meaning
x-axis Revolute joint	C16	Term D_{44}
z-axis Revolute joint	C21	Term D_{66}
Universal joint	C16, C18, C21	Terms D_{44}, D_{64}, D_{66}
Slot joint	C1	Term D_{11}
Point-in-plane joint	C7, C8, C12	Terms D_{22}, D_{32}, D_{33}
Translational joint	C1	Term D_{11}
x-axis Cylindrical joint	C1, C4, C16	Terms D_{11}, D_{41}, D_{44}
z-axis Cylindrical joint	C12, C15, C21	Terms D_{33}, D_{63}, D_{66}
x-axis Planar joint	C7, C8, C9, C12, C13, C16	Terms $D_{22}, D_{32}, D_{42}, D_{33}, D_{43}, D_{44}$
z-axis Planar joint	C1, C2, C6, C7, C11, C21	Terms $D_{11}, D_{21}, D_{61}, D_{22}, D_{62}, D_{66}$
General joint	Use appropriate entries based on unconstrained DOFs.	

The following example shows how you would define the uncoupled linear elastic stiffness behavior for a universal joint at the two available components of relative motion, with two temperature points:

```
TB,JOIN,1,2,,STIF ! Activate JOIN material model with linear elastic stiffness
TBTEMP,100.0  ! Define first temperature
TBDATA,16,D44  ! Define constant D44 in the local ROTX direction
TBDATA,21,D66  ! Define constant D66 in the local ROTZ direction
TBTEMP,200.0  ! Define second temperature
TBDATA,16,D44  ! Define constant D44 in the local ROTX direction.
TBDATA,21,D66  ! Define constant D66 in the local ROTZ direction.
```

2.5.16.2. Nonlinear Elastic Stiffness and Damping Behavior

You can specify nonlinear elastic stiffness as a displacement (rotation) versus force (moment) curve using the **TB**,JOIN command with a suitable *TB0PT* setting. Use the **TBPT** command to specify the data points. The values may be temperature dependent. You can specify nonlinear damping behavior in a similar manner by supplying velocity versus damping force (or moment). The appropriate *TB0PT* labels for each joint element type are shown in the tables below. See *JOIN Specifications* in the **TB** command for a description of each *TB0PT* label.

Nonlinear Stiffness Behavior	
Joint Element	***TB0PT* on TB command**
x-axis Revolute joint	JNSA, JNS4
z-axis Revolute joint	JNSA, JNS6
Universal joint	JNSA, JNS4, and JNS6
Slot joint	JNSA and JNS1
Point-in-plane joint	JNSA, JNS2, and JNS3
Translational joint	JNSA and JNS1
x-axis Cylindrical joint	JNSA, JNS1, and JNS4
z-axis Cylindrical joint	JNSA, JNS3, and JNS6
x-axis Planar joint	JNSA, JNS2, JNS3, and JNS4
z-axis Planar joint	JNSA, JNS1, JNS2, and JNS6

Nonlinear Stiffness Behavior	
Joint Element	***TB0PT* on TB command**
General joint	Use appropriate entries based on unconstrained degrees of freedom

Nonlinear Damping Behavior	
Joint Element	***TB0PT* on TB command**
x-axis Revolute joint	JNDA, JND4
z-axis Revolute joint	JNDA, JND6
Universal joint	JNDA, JND4, and JND6
Slot joint	JNDA and JND1
Point-in-plane joint	JNDA, JND2, and JND3
Translational joint	JNDA and JND1
x-axis Cylindrical joint	JNDA, JND1, and JND4
z-axis Cylindrical joint	JNDA, JND3, and JND6
x-axis Planar joint	JNDA, JND2, JND3, and JND4
z-axis Planar joint	JNDA, JND1, JND2, and JND6
General joint	Use appropriate entries based on unconstrained degrees of freedom

The following example illustrates the specification of nonlinear stiffness behavior for a revolute joint that has only one available component of relative motion (the rotation around the axis of revolution). Two temperature points are specified.

```
TB,JOIN,1,2,2,JNS4
TBTEMP,100.
TBPT,,rotation_value_1,moment_value_1
TBPT,,rotation_value_2,moment_value_2
TBTEMP,200.0
TBPT,,rotation_value_1,moment_value_1
TBPT,,rotation_value_2,moment_value_2
```

2.5.16.3. Hysteretic Frictional Behavior

You can specify hysteretic frictional behavior as a relative displacement (rotation) versus frictional force (moment) curve using the **TB**,JOIN command with a suitable *TB0PT* setting. Use the **TBPT** command to specify the data points. The values may be temperature dependent. Only the upper half of the X-Y plane values is necessary. The curve is then reflected onto the lower half of the X-Y plane. The appropriate *TB0PT* labels for each joint element type are shown in the tables below. See *JOIN Specifications* in the **TB** command for a description of each *TB0PT* label.

Hysteretic Frictional Behavior	
Joint Element	***TB0PT* on TB command**
x-axis Revolute joint	JNFA, JNF4
z-axis Revolute joint	JNFA, JNF6
Universal joint	JNFA, JNF4, and JNF6
Slot joint	JNFA and JNF1
Point-in-plane joint	JNFA, JNF2, and JNF3
Translational joint	JNFA and JNF1
x-axis Cylindrical joint	JNFA, JNF1, and JNF4
z-axis Cylindrical joint	JNFA, JNF3, and JNF6

Hysteretic Frictional Behavior	
Joint Element	***TB0PT* on TB command**
x-axis Planar joint	JNFA, JNF2, JNF3, and JNF4
z-axis Planar joint	JNFA, JNF1, JNF2, and JNF6
General joint	Use appropriate entries based on unconstrained degrees of freedom

You may specify a stick stiffness value using *TB0PT* = FRIC on the **TB**,JOIN command. The stick stiffness value is used to model the elastic behavior inside the two bounding curves specified. Input the stick stiffness for the relevant components of relative motion of a joint element by specifying the terms as part of a 6 x 6 matrix with data table commands as described below.

The 6 x 6 matrix for stick stiffness is as follows:

$$\begin{bmatrix} D_{11} & & & & & \\ D_{21} & D_{22} & & & & \\ D_{31} & D_{32} & D_{33} & & & \\ D_{41} & D_{42} & D_{43} & D_{44} & & \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

The relevant stick stiffness values are based on the joint type as follows:

Joint Element	Constant	Meaning
x-axis Revolute joint	C16	Terms D_{44}
z-axis Revolute joint	C21	Term D_{66}
Universal joint	C16, C18, C21	Terms D_{44}, D_{64}, D_{66}
Slot joint	C1	Terms D_{11}
Point-in-plane joint	C7, C8, C12	Terms D_{22}, D_{32}, D_{33}
Translational joint	C1	Terms D_{11}
x-axis Cylindrical joint	C1, C4, C16	Terms D_{11}, D_{41}, D_{44}
z-axis Cylindrical joint	C12, C15, C21	Terms D_{33}, D_{63}, D_{66}
x-axis Planar joint	C7, C8, C9, C12, C13, C16	Terms D_{22}, D_{32}, D_{42}, D_{33}, D_{43}, D_{44}
z-axis Planar joint	C1, C2, C6, C7, C11, C21	Terms D_{11}, D_{21}, D_{61}, D_{22}, D_{62}, D_{66}
General joint	Depends on the unconstrained relative DOFs.	

The following example illustrates the specification of the hysteretic frictional behavior for an x-axis revolute joint.

Specify the stick stiffness value as follows:

```
TB,JOIN,1,,,FRIC
TBDATA,16,D44
```

Specify the displacement (rotation) versus force (moment) curve as follows:

```
TB,JOIN,1,,2,JNF4
TBPT,,rotation_value_1,moment_value_1
TBPT,,rotation_value_2,moment_value_2
```

Note that the moment values must be positive in the above specification. If the stick stiffness value is not specified, it is computed by default as 100 times the first force value specified on the hysteretic curve.

2.5.17. Contact Friction

Contact friction is a material property that is used with contact elements CONTAC12, CONTAC52, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, and CONTA178. It is specified through the coefficient of friction, MU. Contact friction may be isotropic or orthotropic.

2.5.17.1. Isotropic Friction

Isotropic friction is applicable to 2-D and 3-D contact and is available for all contact elements. Use the **TB**,FRIC command with *TB0PT* = ISO to define isotropic friction, and specify the coefficient of friction MU on the **TBDATA** command. This is the recommended method for defining isotropic friction. To define a temperature dependent coefficient of friction, use the **TBTEMP** command as shown below:

```
TB,FRIC,1,2,,ISO     ! Activate isotropic friction model
TBTEMP,100.0         ! Define first temperature
TBDATA,1,MU          ! Define coefficient of friction at temp 100.0
TBTEMP,200.0         ! Define second temperature
TBDATA,1,MU          ! Define coefficient of friction at temp 200.0
```

Alternatively, you can use MU on the **MP** command to specify the isotropic friction. Use the **MPTEMP** command to define MU as a function of temperature. See *Section 2.4: Linear Material Properties* for details.

2.5.17.2. Orthotropic Friction

The orthotropic friction model uses two different coefficients of friction in two principal directions (see *Section 14.174.3: Frictional Model* in the *Theory Reference for ANSYS and ANSYS Workbench* for details). It is applicable only to 3-D contact and is available for elements CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177. Use the **TB**,FRIC command with *TB0PT* = ORTHO to define orthotropic friction, and specify the coefficients of friction, MU1 and MU2, on the **TBDATA** command. To define a temperature dependent coefficient of friction, use the **TBTEMP** command as shown below:

```
TB,FRIC,1,2,,ORTHO   ! Activate orthotropic friction model
TBTEMP,100.0         ! Define first temperature
TBDATA,1,MU1,MU2     ! Define coefficients of friction at temp 100.0
TBTEMP,200.0         ! Define second temperature
TBDATA,1,MU1,MU2     ! Define coefficients of friction at temp 200.0
```

2.5.18. Cohesive Zone Materials

Cohesive zone materials can be used with interface elements INTER202, INTER203, INTER204 and INTER205, as well as contact elements CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177. For theoretical background on cohesive zone materials, see *Section 4.11: Cohesive Zone Material Model* in the *Theory Reference for ANSYS and ANSYS Workbench*.

2.5.18.1. Cohesive Zone Materials for Interface Elements

The interface elements allow cohesive zone materials to be used for simulating interface delamination and other fracture phenomenon. Use the **TB**,CZM command with *TB0PT* = EXPO to define exponential material behavior, and specify the following material constants using the **TBDATA** command.

Constant	Symbol	Meaning
C1	σ_{max}	maximum normal traction at the interface
C2	δ_n	normal separation across the interface where the maximum normal traction is attained
C3	δ_t	shear separation where the maximum shear traction is attained

To define a temperature dependent material, use the **TBTEMP** command as shown below:

```
TB,CZM,1,2,,EXPO     ! Activate exponential material model
TBTEMP,100.0      ! Define first temperature
TBDATA,1, max, n, t  ! Define material constants at temp 100.0
TBTEMP,200.0      ! Define second temperature
TBDATA,1, max, n, t  ! Define material constants at temp 200.0
```

2.5.18.2. Cohesive Zone Materials for Contact Elements

To model interface delamination, also known as debonding, the contact elements support a cohesive zone material model with bilinear behavior. This model allows two ways to specify material data.

Bilinear Material Behavior with Tractions and Separation Distances

Use the **TB**,CZM command with *TB0PT* = CBDD to define bilinear material behavior with tractions and separation distances, and specify the following material constants using the **TBDATA** command.

Constant	Symbol	Meaning
C1	σ_{max}	maximum normal contact stress
C2	u_n^c	contact gap at the completion of debonding
C3	τ_{max}	maximum equivalent tangential contact stress
C4	u_t^c	tangential slip at the completion of debonding
C5	η	artificial damping coefficient
C6	β	flag for tangential slip under compressive normal contact stress

To define a temperature dependent material, use the **TBTEMP** command as shown below:

```
TB,CZM,1,2,,CBDD     ! Activate bilinear material model with tractions
                     ! and separation distances
TBTEMP,100.0         ! Define first temperature

TBDATA,1, max, u_n^c , max, u_t^c  , ,   ! Define material constants at temp 100.0
TBTEMP,200.0         ! Define second temperature

TBDATA,1, max, u_n^c , max, u_t^c  , ,   ! Define material constants at temp 200.0
```

Bilinear Material Behavior with Tractions and Critical Fracture Energies

Use the **TB**,CZM command with *TB0PT* = CBDE to define bilinear material behavior with tractions and critical fracture energies, and specify the following material constants using the **TBDATA** command.

Constant	Symbol	Meaning
C1	σ_{max}	maximum normal contact stress
C2	G_{cn}	critical fracture energy for normal separation
C3	τ_{max}	maximum equivalent tangential contact stress
C4	G_{ct}	critical fracture energy for tangential slip
C5	η	artificial damping coefficient
C6	β	flag for tangential slip under compressive normal contact stress

To define a temperature dependent material, use the **TBTEMP** command as shown below:

```
TB,CZM,1,2,,CBDE                ! Activate bilinear material model with
                                ! tractions and facture energies
TBTEMP,100.0                    ! Define first temperature
TBDATA,1, max,Gcn, max,Gct, ,   ! Define material constants at temp 100.0
TBTEMP,200.0                    ! Define second temperature
TBDATA,1, max,Gcn, max,Gct, ,   ! Define material constants at temp 200.0
```

2.6. Material Model Combinations

You can combine several of the material model options discussed in this chapter to simulate various material behaviors. *Table 2.6, "Material Model Combination Possibilities"* presents the model options you can combine along with the associated **TB** command labels, and links to sample input listings located under Material Model Combinations in the *Structural Analysis Guide*.

Table 2.6 Material Model Combination Possibilities

Model	With ...	Combination Type	Command, Label	Link to Example
Plasticity	Combined Hardening	Bilinear	**TB**,BISO + **TB**,CHAB	BISO and CHAB Example
Plasticity	Combined Hardening	Multilinear	**TB**,MISO + **TB**,CHAB	MISO and CHAB Example
Plasticity	Combined Hardening	Multilinear	**TB**, PLAS,,,,MISO + **TB**,CHAB	PLAS (Multilinear Isotropic Hardening) and CHAB Example
Plasticity	Combined Hardening	Nonlinear	**TB**,NLISO + **TB**,CHAB	NLISO and CHAB Example
Plasticity	Combined Hardening	Multilinear	**TB**,PLAS,,,,MISO + **TB**,EDP	PLAS (MISO) and EDP Example
Plasticity	Combined Hardening	Multilinear	**TB**,MISO+ **TB**,EDP	MISO and EDP Example
Viscoplasticity	Isotropic Hardening	Bilinear	**TB**,PLAS,,,,BISO + **TB**,EDP	BISO and RATE Example
Viscoplasticity	Isotropic Hardening	Multilinear	**TB**,MISO + **TB**,RATE	MISO and RATE Example
Viscoplasticity	Isotropic Hardening	Multilinear	**TB**,PLAS,,,,MISO + **TB**,RATE	PLAS (MISO) and RATE Example
Viscoplasticity	Isotropic Hardening	Nonlinear	**TB**,NLISO + **TB**,RATE	NLISO and RATE Example
Gurson Plasticity	Isotropic Hardening	Bilinear	**TB**,GURS + **TB**,BISO	GURSON and BISO Example
Gurson Plasticity	Isotropic Hardening	Multilinear	**TB**,GURS + **TB**,MISO	GURSON and MISO Example
Gurson Plasticity	Isotropic Hardening	Multilinear	**TB**,GURS + **TB**,PLAS,,,MISO	GURSON and PLAS (MISO) Example
Gurson Plasticity	Isotropic Hardening	Nonlinear	**TB**,GURS + **TB**,NLISO	GURSON and NLISO Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Bilinear	**TB**,BISO + **TB**,CREEP	BISO and CREEP Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Multilinear	**TB**,MISO + **TB**,CREEP	MISO and CREEP Example

Model	With ...	Combination Type	Command, Label	Link to Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Multilinear	**TB**PLAS,,,,MISO + **TB**,CREEP	PLAS (MISO) and CREEP Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Nonlinear	**TB**,NLISO + **TB**,CREEP	NLISO and CREEP Example
Plasticity and Creep (Implicit)	Kinematic Hardening	Bilinear	**TB**,BKIN + **TB**,CREEP	BKIN and CREEP Example
Anisotropic Plasticity	Isotropic Hardening	Bilinear	**TB**,HILL + **TB**,BISO	HILL and BISO Example
Anisotropic Plasticity	Isotropic Hardening	Multilinear	**TB**,HILL + **TB**,MISO	HILL and MISO Example
Anisotropic Plasticity	Isotropic Hardening	Multilinear	**TB**,HILL + **TB**PLAS,,,,MISO	HILL and PLAS (MISO) Example
Anisotropic Plasticity	Isotropic Hardening	Nonlinear	**TB**,HILL + **TB**,NLSIO	HILL and NLISO Example
Anisotropic Plasticity	Kinematic Hardening	Bilinear	**TB**,HILL + **TB**,BKIN	HILL and BKIN Example
Anisotropic Plasticity	Kinematic Hardening	Multilinear	**TB**,HILL + **TB**,MKIN	HILL and MKIN Example,
Anisotropic Plasticity	Kinematic Hardening	Multilinear	**TB**,HILL + **TB**,/ KINH	HILL and KINH Example
Anisotropic Plasticity	Kinematic Hardening	Multilinear	**TB**,HILL + **TB**PLAS,,,, KINH	HILL and PLAS (KINH) Example
Anisotropic Plasticity	Kinematic Hardening	Chaboche	**TB**,HILL + **TB**,CHAB	HILL and CHAB Example
Anisotropic Plasticity	Combined Hardening	Bilinear Isotropic and Chaboche	**TB**,HILL + **TB**,BISO + **TB**,CHAB	HILL and BISO and CHAB Example
Anisotropic Plasticity	Combined Hardening	Multilinear Isotropic and Chaboche	**TB**,HILL + **TB**,MISO + **TB**,CHAB	HILL and MISO and CHAB Example
Anisotropic Plasticity	Combined Hardening	Multilinear Isotropic and Chaboche	**TB**,HILL + **TB**,PLAS,,,,MISO + **TB**,CHAB	HILL and PLAS (MISO) and CHAB Example
Anisotropic Plasticity	Combined Hardening	Nonlinear Isotropic and Chaboche	**TB**,HILL + **TB**,NLISO + **TB**,CHAB	HILL and NLISO and CHAB Example
Anisotropic Viscoplasticity	Isotropic Hardening	Bilinear	**TB**,HILL + **TB**,RATE + **TB**,BISO	HILL and RATE and BISO Example
Anisotropic Viscoplasticity	Isotropic Hardening	Multilinear	**TB**,HILL + **TB**,RATE + **TB**,MISO	HILL and RATE and MISO Example
Anisotropic Viscoplasticity	Isotropic Hardening	Nonlinear	**TB**,HILL + **TB**,RATE + **TB**,NLISO	HILL and RATE and NLISO Example
Anisotropic Creep (Implicit)			**TB**,HILL + **TB**,CREEP	HILL and CREEP Example

Model	With ...	Combination Type	Command, Label	Link to Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Bilinear	**TB**,HILL + **TB**,CREEP + **TB**,BISO	HILL and CREEP and BISO Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Multilinear	**TB**,HILL + **TB**,CREEP + **TB**,MISO	HILL and CREEP and MISO Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Multilinear	**TB**,HILL + **TB**,CREEP + **TB**,MISO	HILL and CREEP and PLAS (MISO) Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Nonlinear	**TB**,HILL + **TB**,CREEP + **TB**,NLISO	HILL and CREEP and NLISO Example
Anisotropic Creep and Plasticity (Implicit)	Kinematic Hardening	Bilinear	**TB**,HILL +	HILL and CREEP and BKIN Example
Hyperelasticity and Viscoelasticity (Implicit)	Finite Strain Viscoelasticity	Nonlinear	**TB**,HYPER + **TB**,VISCO	HYPERELASTICITY and VISCOELASTICITY Example

Presented below are cross-reference links to other sections in this chapter, and to other locations in the documentation that provide descriptions of the *individual* material model options represented in the table above.

- Bilinear Isotropic Hardening [**TB**,BISO] - *Section 2.5.2.4: Bilinear Isotropic Hardening* [*1*].
- Bilinear Kinematic Hardening [**TB**,BKIN] - *Section 2.5.2.1: Bilinear Kinematic Hardening* [*1*].
- Chaboche Nonlinear Kinematic Hardening [**TB**,CHAB] - *Section 2.5.2.3: Nonlinear Kinematic Hardening* [*1*].
- Creep (Implicit) [**TB**,CREEP] - *Section 2.5.13: Creep Equations*; Creep in the *Structural Analysis Guide*.
- Hill Anisotropy [**TB**,HILL] - *Section 2.5.2.8: Hill's Anisotropy* [*1*].
- Multilinear Isotropic Hardening [**TB**,MISO] - *Section 2.5.2.5: Multilinear Isotropic Hardening* [*1*].
- Multilinear Kinematic Hardening [**TB**,MKIN or KINH] - *Section 2.5.2.2: Multilinear Kinematic Hardening* [*1*].
- Nonlinear Isotropic Hardening [**TB**,NLISO] - *Section 2.5.2.6: Nonlinear Isotropic Hardening* [*1*].
- Rate-Dependent Plasticity [**TB**,RATE] - *Section 2.5.11: Rate-Dependent Plastic (Viscoplastic) Materials*; Viscoplasticity in the *Structural Analysis Guide*.

1. Further information on this option is available under Plastic Material Options in the *Structural Analysis Guide*.

2.7. Explicit Dynamics Materials

Material properties used in explicit dynamic analyses (*ANSYS LS-DYNA User's Guide* program) differ somewhat from those used in ANSYS implicit analyses. (Those used in ANSYS implicit analyses are discussed in *Section 2.4: Linear Material Properties* and *Section 2.5: Data Tables - Implicit Analysis*.) Most explicit dynamics material models require data table input. A data table is a series of constants that are interpreted when they are used. Data tables are always associated with a material number and are most often used to define nonlinear material data (e.g., stress-strain curves). The form of the data table (referred to as the **TB** table) depends on the material model being defined.

For a complete description of all explicit dynamics material models, including detailed data table input, see Material Models in the *ANSYS LS-DYNA User's Guide*.

2.8. Node and Element Loads

Loadings are defined to be of two types: nodal and element. Nodal loads are defined at the nodes and are not directly related to the elements. These nodal loads are associated with the degrees of freedom at the node and are typically entered with the **D** and **F** commands (such as nodal displacement constraints and nodal force loads). Element loads are surface loads, body loads, and inertia loads. Element loads are always associated with a particular element (even if the input is at the nodes). Certain elements may also have "flags."

Flags are not actually loads, but are used to indicate that a certain type of calculation is to be performed. For example, when the FSI (fluid-structure interaction) flag is turned on, a specified face of an acoustic element is treated as an interface between a fluid portion and a structural portion of the model. Similarly, MXWF and MVDI are flags used to trigger magnetic force (Maxwell surface) and Jacobian force (virtual displacement) calculations, respectively, in certain magnetics elements. Details of these flags are discussed under the applicable elements in *Chapter 4: Element Library*.

Flags are associated either with a surface (FSI and MXWF) and are applied as surface loads (below), or with an element (MVDI) and are applied as body loads (below). For the FSI and MXWF flags, values have no meaning - these flags are simply turned on by specifying their label on the appropriate command. For the MVDI flag, its value (which can range from zero to one) is specified, along with the label, on the appropriate command. Flags are always step-applied (i.e., the **KBC** command does not affect them).

Surface loads (pressures for structural elements, convections for thermal element, etc.) may be input in a nodal format or an element format. For example, surface loading may be applied to an element face or, for convenience, to the face nodes of an element (which are then processed like face input). Nodal input of surface loads also allows a more general entry of tapered values. Surface loads are typically input with the **SF** and **SFE** commands. Some elements allow multiple types of surface loads (as shown with the load labels listed under "Surface Loads" in the input table for each element type). Also, some elements allow multiple loads on a single element face (as indicated with the load numbers after the load labels). Load numbers are shown on the element figures (within circles) and point in the direction of positive load to the face upon which the load acts. A surface load applied on the edge of a shell element is on a per unit length basis, not per unit area.

Surface loads are designated by a label and a key. The label indicates the type of surface load and the key indicates where on the element the load acts. For example, for element type PLANE42, the surface load list of "Pressure: face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)" indicates that pressure loads are available on 4 faces: the line from node J to node I defines the element's face 1 (identified on surface load commands with key = 1), and K-J (key = 2), L-K (key = 3), and IL (key = 4). Likewise, for thermal element type PLANE55, the surface load list shows that convections and heat fluxes can be applied to the 4 faces of the element by using surface load commands.

The surface load can be defined on element faces with the **SFE** command by using key (i.e., LKEY), the load label (Lab), and the load value. The **SF** command can be used to define surface loads by using nodes to identify element faces. The CONV load label requires two values, the first value being the film coefficient and the second being the bulk temperature.

A tapered surface load, which allows different values to be defined at the nodes of an element, may be entered with the **SFE** command. Tapered loads are input in the same order that the face nodes are listed. For example, for element type PLANE42 with load label PRES and key = 1, the pressures are input in the node J to I order. For element type SOLID45, which has a surface load list of "Pressures: face 1 (J-I-L-K), etc.," the corresponding pressures are input in the node J, I, L, K order.

Table 2.7, "Surface Loads Available in Each Discipline" shows surface loads available in each discipline and their corresponding ANSYS labels.

Table 2.7 Surface Loads Available in Each Discipline

Discipline	Surface Load	ANSYS Label
Structural	Pressure	PRES[*1*]
Thermal	Convection, Heat Flux, Infinite Surface	CONV, HFLUX, INF
Magnetic	Maxwell Surface, Infinite Surface	MXWF, INF
Electric	Maxwell Surface, Surface Charge Density, Infinite Surface, Temperature	MXWF, CHRGS, INF, TEMP
Fluid	Fluid-Structure Interface, Impedance	FSI, IMPD
All	Superelement Load Vector	SELV

1. Not to be confused with the PRES degree of freedom

Body loads (temperatures for structural elements, heat generation rates for thermal elements, etc.) may be input in a nodal format or an element format. For some structural elements, the temperature does not contribute to the element load vector but is only used for material property evaluation. For thermal elements using the diagonalized specified heat matrix option in a transient analyses, a spatially varying heat generation rate is averaged over the element. Heat generation rates are input per unit volume unless otherwise noted with the element. The element format is usually in terms of the element nodes but may be in terms of fictitious corner points as described for each element. Corner point numbers are shown on the element figures where applicable. Either the nodal or the element loading format may be used for an element, with the element format taking precedence. Nodal body loads are internally converted to element body loads. Body loads are typically entered with the **BF**, **BFE**, and **BFUNIF** commands. See also *Section 2.1.7: Body Loads* for additional details.

Table 2.8, "Body Loads Available in Each Discipline" shows all body loads available in each discipline and their corresponding ANSYS labels.

Table 2.8 Body Loads Available in Each Discipline

Discipline	Body Load	ANSYS Label
Structural	Temperature, Fluence	TEMP[*1*], FLUE
Thermal	Heat Generation Rate	HGEN
Magnetic	Temperature, Current Density, Virtual Displacement, Voltage Drop	TEMP[*1*], JS, MVDI, VLTG
Electric	Temperature, Charge Density	TEMP[*1*], CHRGD
Fluid[*1*]	Heat Generation Rate, Force Density	HGEN, FORC

1. Not to be confused with the TEMP degree of freedom

Inertial loads (gravity, spinning, etc.), are applicable to all elements with structural DOFs and having mass (i.e., elements having mass as an input real constant or having a density (DENS) material property). Inertia loads are typically entered with the **ACEL** and **OMEGA** commands.

Initial stresses can be set as constant or read in from a file for the following element types: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, SHELL208, SHELL209, and SHELL281. The **INISTATE** command allows you to set constant initial stress for selected elements and, optionally, only for specified materials. This command also allows you to read in a file specifying the initial stresses. The stresses specified in the input file can be applied to the element centroids or element integration points, and can be applied to the same points for all selected elements or can be applied differently for each element. The stresses can also be a written to an external file. See the **INISTATE** command, and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide* for more information on the new **INISTATE** capability.

2.9. Triangle, Prism and Tetrahedral Elements

Degenerated elements are elements whose characteristic face shape is quadrilateral, but is modeled with at least one triangular face. For example, PLANE42 triangles, SOLID45 wedges, and SOLID45 tetrahedra are all degenerated shapes.

Degenerated elements are often used for modeling transition regions between fine and coarse meshes, for modeling irregular and warped surfaces, etc. Degenerated elements formed from quadrilateral and brick elements *without midside nodes* are much less accurate than those formed from elements with midside nodes and should not be used in high stress gradient regions. If used elsewhere, they should be used with caution.

An exception where triangular shell elements are preferred is for severely skewed or warped elements. Quadrilateral shaped elements should not be skewed such that the included angle between two adjacent faces is outside the range of 90° ± 45° for non-midside-node elements or 90° ± 60° for midside-node elements. Warping occurs when the 4 nodes of a quadrilateral shell element (or solid element face) are not in the same plane, either at input or during large deflection. Warping is measured by the relative angle between the normals to the face at the nodes. A flat face (no warping) has all normals parallel (zero relative angle). A warning message is output if warping is beyond a small, but tolerable value. If warping is excessive, the problem will abort. See the *Theory Reference for ANSYS and ANSYS Workbench* for element warping details and other element checking details. Triangular (or prism) elements should be used in place of a quadrilateral (or brick) element with large warping.

When using triangular elements in a rectangular array of nodes, best results are obtained from an element pattern having alternating diagonal directions. Also, for shell elements, since the element coordinate system is relative to the I-J line, the stress results are most easily interpreted if the I-J lines of the elements are all parallel.

Degenerated triangular 2-D solid and shell elements may be formed from 4-node quadrilateral elements by defining duplicate node numbers for the third and fourth (K and L) node locations. The node pattern then becomes I, J, K, K. If the L node is not input, it defaults to node K. If extra shape functions are included in the element, they are automatically suppressed (degenerating the element to a lower order). Element loads specified on a nodal basis should have the same loads specified at the duplicate node locations. When forming a degenerated triangular element by repeating node numbers, the face numbering remains the same. Face 3, however, condenses to a point. The centroid location printed for a degenerated triangular element is usually at the geometric centroid of the element. Elements should be oriented with alternating diagonals, if possible.

Degenerated triangular prism elements may be formed from 8-node 3-D solid elements by defining duplicate node numbers for the third and fourth (K and L) and the seventh and eighth (O and P) node locations. The node pattern then becomes I, J, K, K, M, N, O, O. When forming a degenerated prism element by repeating node numbers, the face numbering remains the same. Face 4, however, condenses to a line. The centroid location printed for a degenerated element is not at the geometric centroid but is at an average nodal location. The integration points are proportionately rearranged within the element. Elements should be oriented with alternating diagonals, if possible. If extra shape functions are included in the element, they are partially suppressed. Element loads should have the same loads specified at the duplicate node locations.

A degenerated tetrahedral element may be formed from a triangular prism element by a further condensation of face 6 to a point. The input node pattern should be I, J, K, K, M, M, M, M. If extra shape functions are included in the element, they are automatically suppressed. Element nodal loads should have the same loads specified at the duplicate node locations.

Warning: Surface stress (or convection heat flow) printout (see *Section 2.2.2.5: Surface Solution*) should not be requested on a condensed face. Also, pressures (or convection conditions) should not be defined on a condensed face.

2.10. Shell Elements

Shell elements are a special class of elements that are designed to efficiently model thin structures. They take advantage of the fact that the only shear on the free surfaces is in-plane. Normals to the shell middle surface stay straight, but not necessarily normal. As a result, the in-plane strain variation through the thickness cannot be more complex than linear.

The assumption of linear in-plane strain variation through the thickness is definitely not valid at the edges of layered composite shell elements that have different material properties at each layer. For accurate stresses in this area, you should use submodeling.

There are no hard rules as to when is it valid to use shell elements. But if the structure acts like a shell, then you may use shell elements. The program does not check to see if the element thickness exceeds its width (or many times its width) since such an element may be part of a fine mesh of a larger model that acts as a shell. If the initial shape of the model is curved, then the radius/thickness ratio is important since the strain distribution through the thickness will depart from linear as the ratio decreases. With the exception of SHELL61 and SHELL63, all shell elements allow shear deformation. This is important for relatively thick shells.

The element coordinate system for all shell elements has the z-axis normal to the plane. The element x-axis is in the plane, with its orientation determined by one of the following: the **ESYS** command, side I-J of the element, or real constants.

Various shell element types tolerate a different degree of warping before their results become questionable (see *Section 13.7.13: Warping Factor* in the *Theory Reference for ANSYS and ANSYS Workbench*). Four-node shell elements that do not have all their nodes in the same plane are considered to be warped. Eight-node shell elements can accept a much greater degree of warping, but unlike other midside-node elements, their midside nodes cannot be dropped.

The in-plane rotational (drill) stiffness is added at the nodes for solution stability, as shell elements do not have a true in-plane stiffness. Consequently, you should never expect the in-plane rotational stiffness to carry a load.

Nodes are normally located on the center plane of the element. You can offset nodes from the center plane using one of the following: the **SECOFFSET** command, an element KEYOPT, or a rigid link (MPC184) that connects a middle surface node to an out-of-plane node. You must use node offsets with care when modeling initially curved structures with either flat or curved elements. For curved elements, an increased mesh density in the circumferential direction may improve the results.

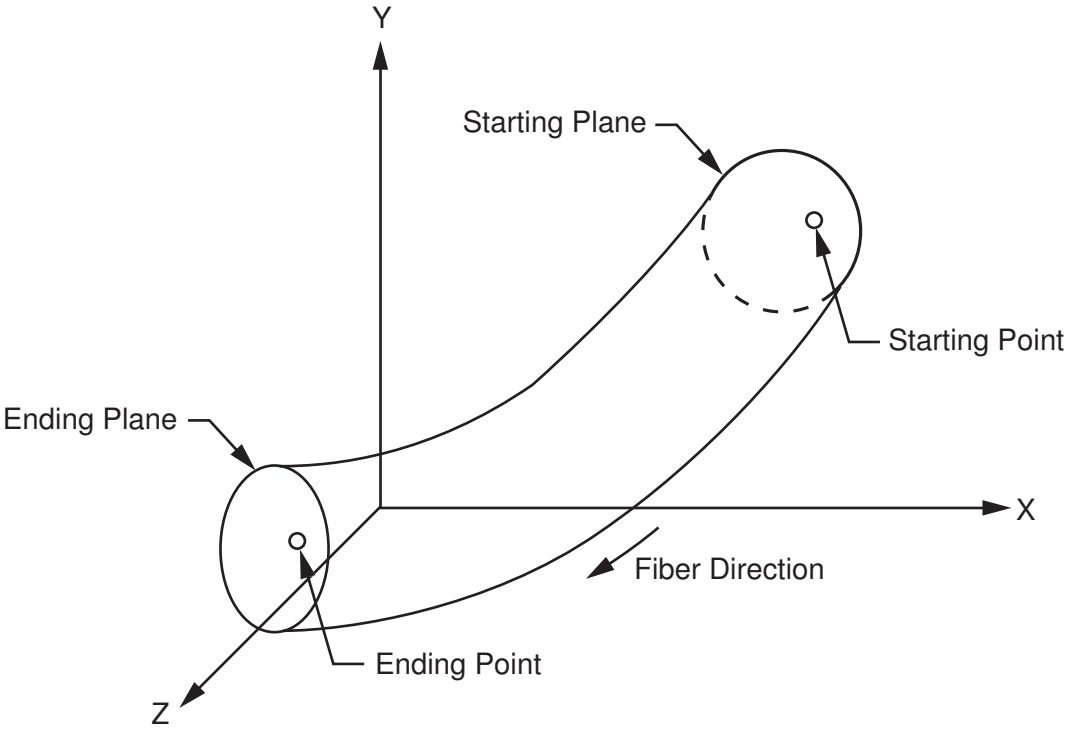
2.11. Generalized Plane Strain Option of 18x Solid Elements

The generalized plane strain option is a feature developed for PLANE182 and PLANE183. The generalized plane strain feature assumes a finite deformation domain length in the Z direction, as opposed to the infinite value assumed for standard plane strain. Generalized plane strain, therefore, will give more practical results for deformation problems where the Z-direction dimension is not long enough. It will also give users a more efficient way to simulate certain 3-D deformations using 2-D element options.

The deformation domain or structure is formed by extruding a plane area along a curve with a constant curvature, as shown in *Figure 2.2, "Generalized Plane Strain Deformation"*. The extruding begins at the starting (or reference) plane and stops at the ending plane. The curve direction along the extrusion path is called the fiber direction. The starting and ending planes must be perpendicular to this fiber direction at the beginning and ending intersections. If the boundary conditions and loads in the fiber direction do not change over the course of the curve, and if the starting plane and ending plane remain perpendicular to the fiber direction during deformation, then the amount of deformation of all cross sections will be identical throughout the curve, and will not vary at any curve position in the fiber direction. Therefore, any deformation can be represented by the deformation on the starting plane, and the 3-D deformation can be simulated by solving the deformation problem on the starting

plane. The existing plane strain and axisymmetric options will be particular cases of the generalized plane strain option.

Figure 2.2 Generalized Plane Strain Deformation



All inputs and outputs are in the global Cartesian coordinate system. The starting plane must be the X-Y plane, and must be meshed. The applied nodal force on the starting plane is the total force along the fiber length. The geometry in the fiber direction is specified by the rotation about X and Y of the ending plane and the fiber length passing through a user-specified point on the starting plane called the starting or reference point. The starting point creates an ending point on the ending plane through the extrusion process. The boundary conditions and loads in the fiber direction are specified by applying displacements or forces at the ending point. This ending point can be different from regular nodes, in that it is designated by the same X - Y coordinates that are fixed in plane during deformation.

The generalized plane strain option introduces three new degrees of freedom for each element. Two internal nodes will be created automatically at the solution stage for the generalized plane strain option to carry the extra three DOF's. Users can apply boundary conditions and loads and check the results of the fiber length and rotation angle changes, and reaction forces, using the commands **GSBDATA**, **GSGDATA**, **GSSOL**, and **GSLIST**. The results of the fiber length change, rotation angle change, and reaction forces can also be viewed through **OUTPR**.

The fiber length change is positive when the fiber length increases. The sign of the rotation angle or angle change is determined by how the fiber length changes when the coordinates of the ending point change. If the fiber length decreases when the X coordinate of the ending point increases, the rotation angle about Y is positive. If the fiber length increases when the Y coordinate of the ending point increases, the rotation angle about X is positive.

In Eigenvalue analyses, such as Eigen buckling and modal analysis, the generalized plane strain option usually reports fewer Eigenvalues and Eigenvectors than you would obtain in a 3-D analysis. Because it reports only homogenous deformation in the fiber direction, generalized plane strain employs only three DOFs to account for these deformations. The same 3-D analysis would incorporate many more DOFs in the fiber direction.

Because the mass matrix terms relating to DOFs in the fiber direction are approximated for modal and transient analyses, you cannot use the lumped mass matrix for these types of analyses, and the solution may be slightly different from regular 3-D simulations when any of the three designated DOFs is not restrained.

2.12. Axisymmetric Elements

An axisymmetric structure may be represented by a plane (X,Y) finite-element model. The use of an axisymmetric model greatly reduces the modeling and analysis time compared to that of an equivalent 3-D model. A special class of ANSYS axisymmetric elements (called harmonic elements: PLANE25, SHELL61, PLANE75, PLANE78, FLUID81, and PLANE83) allow a nonaxisymmetric load and are discussed in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

All axisymmetric elements are modeled on a 360° basis. Hence, all input and output nodal heat flows, forces, moments, fluid flows, current flows, electric charges, magnetic fluxes, and magnetic current segments must be input in this manner. Similarly, input real constants representing volumes, convection areas, thermal capacitances, heat generations, spring constants, and damping coefficients must also be input in on a 360° basis.

Unless otherwise stated, the model must be defined in the Z = 0.0 plane. The global Cartesian Y-axis is assumed to be the axis of symmetry. Further, the model is developed only in the +X quadrants. Hence, the radial direction is in the +X direction.

The boundary conditions are described in terms of the structural elements. The forces (FX, FY, etc.) and displacements (UX, UY, etc.) for the structural elements are input and output in the nodal coordinate system. All nodes along the y-axis centerline (at x = 0.0) should have the radial displacements (UX if not rotated) specified as zero, unless a pinhole effect is desired. At least one value of UY should be specified or constrained to prevent rigid body motions. Torsion, while axisymmetric, is available only for a few element types. If an element type allows torsion, all UZ degrees of freedom should be set to 0.0 on the centerline, and one node with a positive X coordinate must also have a specified or constrained value of UZ. Pressures and temperatures may be applied directly. Acceleration, if any, is usually input only in the axial (Y) direction. Similarly, angular velocity, if any, is usually input only about the Y axis. See *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*, Case A for an expanded discussion.

2.13. Axisymmetric Elements with Nonaxisymmetric Loads

An axisymmetric structure (defined with the axial direction along the global Y axis and the radial direction parallel to the global X axis) may be represented by a plane (X,Y) finite-element model. The use of an axisymmetric model greatly reduces the modeling and analysis time compared to that of an equivalent 3-D model. A special class of ANSYS axisymmetric elements (called harmonic elements) allows a nonaxisymmetric load. For these elements (PLANE25, SHELL61, PLANE75, PLANE78, FLUID81, and PLANE83) , the load is defined as a series of harmonic functions (Fourier series). For example, a load F is given by:

$$F(\theta) = A_0 + A_1 \cos\theta + B_1 \sin\theta + A_2 \cos 2\theta + B_2 \sin 2\theta + A_3 \cos 3\theta + B_3 \sin 3\theta + ...$$

Each term of the above series must be defined as a separate load step. A term is defined by the load coefficient (A^{ℓ} or B^{ℓ}), the number of harmonic waves (ℓ), and the symmetry condition ($\cos^{\ell}\theta$ or $\sin^{\ell}\theta$). The number of harmonic waves, or the mode number, is input with the **MODE** command. Note that $\ell = 0$ represents the axisymmetric term (A_0). θ is the circumferential coordinate implied in the model. The load coefficient is determined from the standard ANSYS boundary condition input (i.e., displacements, forces, pressures, etc.). Input values for temperature, displacement, and pressure should be the peak value. The input value for force and heat flow should be a number equal to the peak value per unit length times the circumference. The symmetry condition is determined from the *ISYM* value also input on the **MODE** command. The description of the element given in *Chapter 4: Element Library* and in the appropriate sections of the *Theory Reference for ANSYS and ANSYS Workbench* should be reviewed to see which deformation shape corresponds to the symmetry conditions.

Results of the analysis are written to the results file. The deflections and stresses are output at the peak value of the sinusoidal function. The results may be scaled and summed at various circumferential (θ) locations with POST1. This may be done by storing results data at the desired θ location using the *ANGLE* argument of the **SET** command. A load case may be defined with **LCWRITE**. Repeat for each set of results, then combine or scale the load cases as desired with **LCOPER**. Stress (and temperature) contour displays and distorted shape displays of the combined results can also be made.

Caution should be used if the harmonic elements are mixed with other, nonharmonic elements. The harmonic elements should not be used in nonlinear analyses, such as large deflection and/or contact analyses.

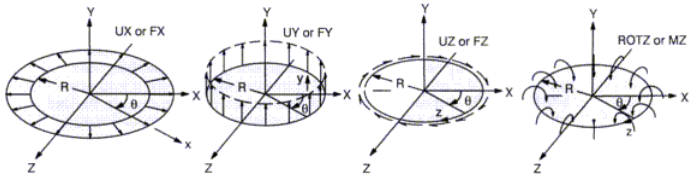
The element matrices for harmonic elements are dependent upon the number of harmonic waves *(MODE*) and the symmetry condition (*ISYM*). For this reason, neither the element matrices nor the triangularized matrix is reused in succeeding substeps if the *MODE* and *ISYM* parameters are changed. In addition, a superelement generated with particular *MODE* and *ISYM* values must have the same values in the "use" pass.

For stress stiffened (prestressed) structures, the ANSYS program uses only the stress state of the most recent previous *MODE* = 0 load case, regardless of the current value of *MODE*.

Loading Cases - The following cases are provided to aid the user in obtaining a physical understanding of the *MODE* parameter and the symmetric (*ISYM*=1) and antisymmetric (*ISYM*=-1) loading conditions. The loading cases are described in terms of the structural elements. The forces (FX, FY, etc.) and displacements (UX, UY, etc.) for the structural elements are input and output in the nodal coordinate system. In all cases illustrated, it is assumed that the nodal coordinate system is parallel to the global Cartesian coordinate system. The loading description may be extended to any number of modes. The harmonic thermal elements (PLANE75 and PLANE78) are treated the same as PLANE25 and PLANE83, respectively, with the following substitutions: UY to TEMP, and FY to HEAT. The effects of UX, UZ, ROTZ, FX, FZ and MZ are all ignored for thermal elements.

Case A: (*MODE* = 0, *ISYM* not used) - This is the case of axisymmetric loading (similar to the axisymmetric option of PLANE42, etc.) except that torsional effects are included. *Figure 2.3, "Axisymmetric Radial, Axial, Torsion and Moment Loadings"* shows the various axisymmetric loadings. Pressures and temperatures may be applied directly. Acceleration, if any, is usually input only in the axial (Y) direction. Similarly, angular velocity, if any, is usually input only about the Y axis.

Figure 2.3 Axisymmetric Radial, Axial, Torsion and Moment Loadings



The total force (F) acting in the axial direction due to an axial input force (FY) is:

$$F = \int_0^{2\pi} (\text{force per unit length}) * (\text{increment length})$$

$$F = \int_0^{2\pi} (FY / 2\pi R) * (R d\theta) = FY$$

where FY is on a full 360° basis.

The total applied moment (M) due to a tangential input force (FZ) acting about the global axis is:

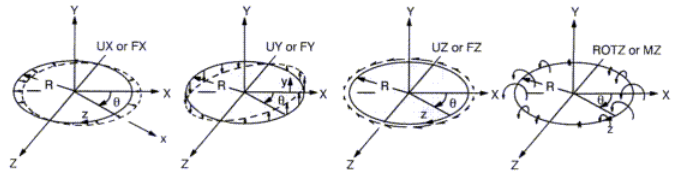
$$M = \int_0^{2\pi} (\text{force per unit length})*(\text{lever arm})*(\text{increment length})$$

$$M = \int_0^{2\pi} (-FZ / 2\pi R) * (R) * (Rd\theta) = -R * FZ$$

where FZ is on a full 360° basis. Calculated reaction forces are also on a full 360° basis and the above expressions may be used to find the total force. Nodes at the centerline (X = 0.0) should have UX and UZ (and ROTZ, for SHELL61) specified as zero, unless a pinhole effect is desired. At least one value of UY should be specified or constrained to prevent rigid body motions. Also, one node with a nonzero, positive X coordinate must have a specified or constrained value of UZ if applicable. When Case A defines the stress state used in stress stiffened analyses, torsional stress is not allowed.

Case B: (*MODE* = 1, *ISYM*=1) - An example of this case is the bending of a pipe. *Figure 2.4, "Bending and Shear Loading (ISYM = 1)"* shows the corresponding forces or displacements on a nodal circle. All functions are based on sin θ or cos θ. The input and output values of UX, FX, etc., represent the peak values of the displacements or forces. The peak values of UX, UY, FX and FY (and ROTZ and MZ for SHELL61) occur at $\theta = 0°$, whereas the peak values of UZ and FZ occur at $\theta = 90°$. Pressures and temperatures are applied directly as their peak values at $\theta = 0°$. The thermal load vector is computed from T_{peak}, where T_{peak} is the input element or nodal temperature. The reference temperature for thermal strain calculations [**TREF**] is internally set to zero in the thermal strain calculation for the harmonic elements if *MODE* > 0. Gravity (g) acting in the global X direction should be input [**ACEL**] as *ACELX* = g, *ACELY* = 0.0, and *ACELZ* = -g. The peak values of $\sigma_x, \sigma_y, \sigma_z$ and σ_{xy} occur at $\theta = 0°$, whereas the peak values of σ_{yz} and σ_{xz} occur at 90 °.

Figure 2.4 Bending and Shear Loading (ISYM = 1)



The total applied force in the global X direction (F) due to both an input radial force (FX) and a tangential force (FZ) is:

$$F = \int_0^{2\pi} (\text{force per unit length})*(\text{directional cosine})*(\text{increment length})$$

$$F = \int_0^{2\pi} ((FX(\cos\theta) / 2\pi R) * (\cos\theta) + (FZ(\sin\theta / 2\pi R) * (-\sin\theta)) * (Rd\theta)$$

$$F = (FX - FZ) / 2$$

where FX and FZ are the peak forces on a full 360° basis. Calculated reaction forces are also the peak values on a full 360° basis and the above expression may be used to find the total force. These net forces are independent of radius so that they may be applied at any radius (including X = 0.0) for the same net effect.

An applied moment (M) due to an axial input force (FY) for this case can be computed as follows:

$$M = \int_0^{2\pi} (\text{force per unit length})*(\text{lever arm})*(\text{increment length})$$

$$M = \int_0^{2\pi} (FY(\cos\theta)/2\,\pi R) * (R\cos\theta) * (Rd\theta) = (FY)\ R/2$$

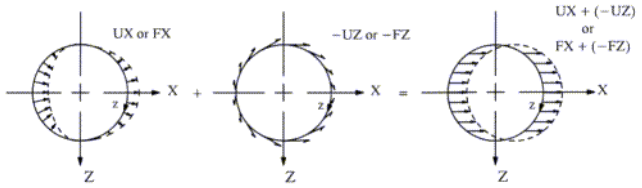
An additional applied moment (M) is generated based on the input moment (MZ):

$$M = \int_0^{2\pi} (\text{force per unit length}) * (\text{directional cosine}) * (\text{increment length})$$

$$M = \int_0^{2\pi} (MZ(\cos\theta)/2vR) * (\cos\theta)(Rd\theta) = (MZ)/2$$

If it is desired to impose a uniform lateral displacement (or force) on the cross section of a cylindrical structure in the global X direction, equal magnitudes of UX and UZ (or FX and FZ) may be combined as shown in *Figure 2.5, "Uniform Lateral Loadings"*.

Figure 2.5 Uniform Lateral Loadings

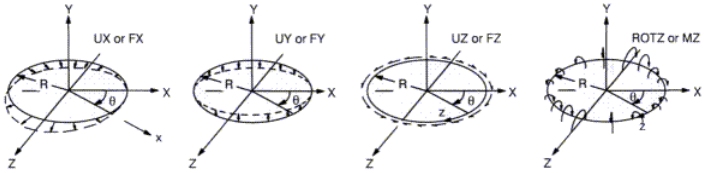


When UX and UZ are input in this manner, the nodal circle moves in an uniform manner. When FX and FZ are input in this manner, a uniform load is applied about the circumference, but the resulting UX and UZ will not, in general, be the same magnitude. If it is desired to have the nodal circle moving in a rigid manner, it can be done by using constraint equations [**CE**] so that UX = -UZ.

Node points on the centerline (X = 0.0) should have UY specified as zero. Further, UX must equal -UZ at all points along the centerline, which may be enforced with constraint equations. In practice, however, it seems necessary to do this only for the harmonic fluid element, FLUID81, since this element has no static shear stiffness. To prevent rigid body motions, at least one value of UX or UZ, as well as one value of UY (not at the centerline), or ROTZ, should be specified or constrained in some manner. For SHELL61, if plane sections (Y = constant) are to remain plane, ROTZ should be related to UY by means of constraint equations at the loaded nodes.

Case C: (*MODE* = 1, *ISYM* = -1) - This case (shown in *Figure 2.6, "Bending and Shear Loading (ISYM = -1)"*) represents a pipe bending in a direction 90° to that described in Case B.

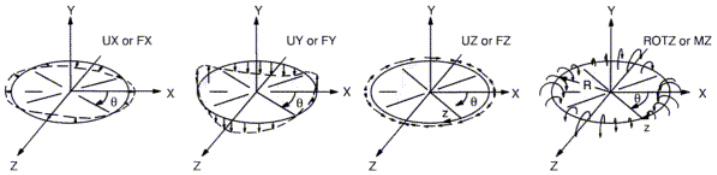
Figure 2.6 Bending and Shear Loading (*ISYM* = -1)



The same description applying to Case B applies also to Case C, except that the negative signs on UZ, FZ, and the direction cosine are changed to positive signs. Also, the location of the peak values of various quantities are switched between the 0° and 90° locations.

Case D: (*MODE* = 2, *ISYM* = 1) - The displacement and force loadings associated with this case are shown in *Figure 2.7, "Displacement and Force Loading Associated with MODE = 2 and ISYM = 1"*. All functions are based on sin 2 θ and cos 2 θ.

Figure 2.7 Displacement and Force Loading Associated with *MODE* = 2 and *ISYM* = 1



Additional Cases: There is no programmed limit to the value of *MODE*. Additional cases may be defined by the user.

2.14. Shear Deflection

Shear deflection effects are often significant in the lateral deflection of short beams. The significance decreases as the ratio of the radius of gyration of the beam cross-section to the beam length becomes small compared to unity. Shear deflection effects are activated in the stiffness matrices of ANSYS beam elements by including a nonzero shear deflection constant (SHEAR_) in the real constant list for that element type.

The shear deflection constant is defined as the ratio of the actual beam cross-sectional area to the effective area resisting shear deformation. The shear constant should be equal to or greater than zero. The element shear stiffness decreases with increasing values of the shear deflection constant. A zero shear deflection constant may be used to neglect shear deflection. Shear deflection constants for several common sections are as follows: rectangle (6/5), solid circle (10/9), hollow (thin-walled) circle (2), hollow (thin-walled) square (12/5). Shear deflection constants for other cross-sections can be found in structural handbooks.

2.15. Geometric Nonlinearities

Geometric nonlinearities refer to the nonlinearities in the structure or component due to the changing geometry as it deflects. That is, the stiffness [K] is a function of the displacements {u}. The stiffness changes because the shape changes and/or the material rotates. The program can account for five types of geometric nonlinearities:

1. *Large strain* assumes that the strains are no longer infinitesimal (they are finite). Shape changes (e.g., area, thickness, etc.) are also taken into account. Deflections and rotations may be arbitrarily large.
2. *Large rotation* assumes that the rotations are large but the mechanical strains (those that cause stresses) are evaluated using linearized expressions. The structure is assumed not to change shape except for rigid body motions. The elements of this class refer to the original configuration.
3. *Stress stiffening* assumes that both strains and rotations are small. A first order approximation to the rotations is used to capture some nonlinear rotation effects.
4. *Spin softening* also assumes that both strains and rotations are small. This option accounts for the radial motion of a body's structural mass as it is subjected to an angular velocity. Hence it is a type of large deflection but small rotation approximation.
5. *Pressure load stiffness* accounts for the change of stiffness caused by the follower load effect of a rotating pressure load. In a large deflection run, this can affect the convergence rate.

All elements support the spin softening capability, while only some of the elements support the other options. *Table 2.9, "Elements Having Nonlinear Geometric Capability"* lists the elements that have large strain, large deflection, stress stiffening capability, and/or pressure load stiffness. Explicit Dynamics elements (160 to 167) are not included in this table.

Table 2.9 Elements Having Nonlinear Geometric Capability

Element Name - Description	NLGEOM=ON	Stress Stiffening	Pressure Load Stiffness
LINK1 - 2-D Spar	LR	AN	-
BEAM3 - 2-D Elastic Beam	LR	AN	-
BEAM4 - 3-D Elastic Beam	LR	AN	-
SOLID5 - 3-D Coupled-Field Solid	LR[1]	AN[2]	-
COMBIN7 - Revolute Joint	LR	x	-
LINK8 - 3-D Spar	LR	AN	-
LINK10 - Tension-only or Compression-only Spar	LR	x	-
LINK11 - Linear Actuator	LR	x	-
PLANE13 - 2-D Coupled-Field Solid	LS	AN[2]	-
COMBIN14 - Spring-Damper	LR	x	-
PIPE16 - Elastic Straight Pipe	LR	x	-
PIPE17 - Elastic Pipe Tee	LR	x	-
PIPE18 - Elastic Curved Pipe (Elbow)	LR	-	-
PIPE20 - Plastic Straight Pipe	LR	x	-
MASS21 - Structural Mass	LR	-	-
BEAM23 - 2-D Plastic Beam	LR	AN	-
BEAM24 - 3-D Thin-walled Beam	LR	AN	-
PLANE25 - 4-Node Axisymmetric-Harmonic Structural Solid	-	x	-
SHELL28 - Shear/Twist Panel	-	AN	-
COMBIN39 - Nonlinear Spring	LR	x	-
SHELL41 - Membrane Shell	LR	AN	-
PLANE42 - 2-D Structural Solid	LS	AN	x
SHELL43 - Plastic Large Strain Shell	LS	AN	-
BEAM44 - 3-D Tapered Unsymmetric Beam	LR	AN	-
SOLID45 - 3-D Structural Solid	LS	AN	x
SOLID46 - 3-D Layered Structural Solid	LR	AN	x
MATRIX50 - Superelement	LR	-	-
BEAM54 - 2-D Elastic Tapered Unsymmetric Beam	LR	AN	-
PIPE59 - Immersed Pipe or Cable	LR	x	-
PIPE60 - Plastic Curved Pipe (Elbow)	LR	-	-
SHELL61 - Axisymmetric-Harmonic Structural Shell	-	x	-
SOLID62 - 3-D Magneto-Structural Solid	LS	x	-
SHELL63 - Elastic Shell	LR	AN	-
SOLID65 - 3-D Reinforced Concrete Solid	LS	AN	x
PLANE82 - 2-D 8-Node Structural Solid	LS	AN	x
PLANE83 - 8-Node Axisymmetric-Harmonic Structural Solid	-	x	-
VISCO88 - 2-D 8-Node Viscoelastic Solid	LS	AN	x
VISCO89 - 3-D 20-Node Viscoelastic Solid	LS	AN	x
SHELL91 - Nonlinear Layered Structural Shell	LS	AN	-
SOLID92 - 3-D 10-Node Tetrahedral Structural Solid	LS	AN	x

Element Name - Description	NLGEOM=ON	Stress Stiffening	Pressure Load Stiffness
SHELL93 - 8-Node Structural Shell	LS	AN	-
SOLID95 - 3-D 20-Node Structural Solid	LS	AN	x
SOLID98 - Tetrahedral Coupled-Field Solid	LR[1]	AN[2]	-
SHELL99 - Linear Layered Structural Shell	LR	AN	-
VISCO106 - 2-D Large Strain Solid	LS	AN	-
VISCO107 - 3-D Large Strain Solid	LS	AN	-
VISCO108 - 2-D 8-Node Large Strain Solid	LS	AN	-
SURF153 - 2-D Structural Surface Effect	SC	x	x
SURF154 - 3-D Structural Surface Effect	SC	x	x
TARGE169 - 2-D Target Segment	SC	-	-
TARGE170 - 3-D Target Segment	SC	-	-
CONTA171 - 2-D Surface-to-Surface Contact	SC	-	-
CONTA172 - 2-D 3-Node Surface-to-Surface Contact	SC	-	-
CONTA173 - 3-D Surface-to-Surface Contact	SC	-	-
CONTA174 - 3-D 8-Node Surface-to-Surface Contact	SC	-	-
CONTA175 - 2-D/3-D Point-to-Surface and Edge-to-Surface Contact	SC	-	-
CONTA176 - 3-D Line-to-Line Contact	SC	-	-
CONTA177 - 3-D Line-to-Surface Contact	SC	-	-
LINK180 - 3-D Finite Strain Spar	LS	AB	-
SHELL181 - Finite Strain Shell	LS	AB	x[3]
PLANE182 - 2-D Structural Solid	LS	AB	x
PLANE183 - 2-D 8-Node Structural Solid	LS	AB	x
SOLID185 - 3-D 8-Node Structural Solid	LS	AB	x
SOLID186 - 3-D 20-Node Structural Solid	LS	AB	x
SOLID187 - 3-D 10-Node Tetrahedral Structural Solid	LS	AB	x
BEAM188 - 3-D Finite Strain Beam	LS	AB	x
BEAM189 - 3-D Finite Strain Beam	LS	AB	x
SOLSH190 - 3-D 8–Node Structural Solid-Shell	LS	AB	x
SOLID191 - 3-D 20-Node Layered Structural Solid	-	x	x
SHELL208 - 2-Node Finite Strain Axi. Shell	LS	AB	x
SHELL209 - 3-Node Finite Strain Axi. Shell	LS	AB	x
SHELL281 - 8-Node Finite Strain Shell	LS	AB	x[3]

Codes associated with NLGEOM = ON:

LS = large strain element

LR = Element that can do a rigid body rotation. The NLGEOM = ON option provides only a rigid body rotation. Strains, if any, are linear.

LR[1] = Same as LR, and applies to both structural and piezoelectric analysis.

SC = Surface or contact element. The element follows the underlying element.

Codes associated with stress stiffening:

x = Has option of computing stress stiffness matrix

AN = If NLGEOM = ON, stress stiffening is automatically included. However, the element is not capable of linear buckling using **ANTYPE**,BUCKLE.

AN[2] = Same as AN, applies to structural analyses only.

AB = If NLGEOM = ON, stress stiffening is automatically included, and the element is also capable of linear buckling using **ANTYPE**,BUCKLE.

Code associated with pressure load stiffness:

x = Has option of computing symmetric or unsymmetric pressure load stiffness matrix using **SOLCONTROL**,,,INCP. Symmetry or unsymmetry is handled on the **NROPT** command.

x[3] = Same as x, but does not apply to edge loading.

2.16. Mixed u-P Formulation Elements

Mixed u-P elements use both displacement and hydrostatic pressure as primary unknown variables. These elements include the 18x elements PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, and SOLSH190 with KEYOPT(6) > 0.

2.16.1. Element Technologies

Incompressible material behavior may lead to some difficulties in numerical simulation, such as volumetric locking, inaccuracy of solution, checkerboard pattern of stress distributions, or, occasionally, divergence. Mixed u-P elements are intended to overcome these problems.

Lagrange multiplier based mixed u-P elements (18x solid elements with KEYOPT(6) > 0) can also be used to overcome incompressible material problems. They are designed to model material behavior with high incompressibility such as fully or nearly incompressible hyperelastic materials and nearly incompressible elastoplastic materials (high Poisson ratio or undergoing large plastic strain). Lagrange multipliers extend the internal virtual work so that the volume constraint equation is included explicitly. This introduces hydrostatic pressure as a new independent variable. Unlike the hyperelastic elements, the hydrostatic pressure variables are not condensed on the element level, but are solved at the global level. See the *Theory Reference for ANSYS and ANSYS Workbench* for further details.

The mixed u-P formulation of the 18x solid elements offers you more choices in handling incompressible material behavior. You can combine the mixed u-P formulation with other element technologies such as the $\bar{B}$ method (also known as the selective reduced integration method), the uniform reduced integration method, and the enhanced strain formulation method. Furthermore, the mixed u-P formulation is associated with hyperelastic models, such as Mooney-Rivlin, Neo-Hookean, Ogden, Arruda-Boyce, polynomial form, and user-defined.

2.16.2. 18x Mixed u-P Elements

The number of independent hydrostatic pressure DOFs depends on the element type, element technology, and the value of KEYOPT(6), as shown in *Table 2.10, "Number of Independent Pressure DOFs in One Element"*.

Table 2.10 Number of Independent Pressure DOFs in One Element

Element	Basic Element Technology	KEYOPT(6)	Number of Pressure DOFs	Interpolation Function
PLANE182	$\bar{B}$ method (selective reduced integration) *or* uniform reduced integration	1	1	Constant
PLANE182	Enhanced strain formulation	1	3	Linear
PLANE183	Uniform reduced integration (KEYOPT(1) = 0)	1	3	Linear
PLANE183	3-point integration (KEYOPT(1) = 1)	1	3	Linear
SOLID185	$\bar{B}$ method (selective reduced integration) *or* uniform reduced integration	1	1	Constant
SOLID185	Enhanced strain formulation	1	4	Linear
SOLID186	Uniform reduced integration	1	4	Linear
SOLID186	Full integration	1	4	Linear
SOLID187	4-point integration	1	1	Constant
SOLID187	4-point integration	2	4	Linear
SOLSH190	Enhanced strain formulation	1	4	Linear

The hydrostatic pressure has an interpolation function one order lower than the one for volumetric strain in elements PLANE182, SOLID185, and SOLSH190 with enhanced strain formulation method, PLANE183, SOLID186, and SOLID187 with constant pressure. Therefore, elastic strain only agrees with stress in an element on the average instead of point-wise.

The volume constraint equation is checked for each element of a nonlinear analysis. The number of elements in which the constraint equation is not satisfied is reported in the output file. The default tolerance for the volumetric compatibility or volume constraint check is 1.0×10^{-5}. You can change this value using the **SOLCONTROL** command. For more details, see the *Theory Reference for ANSYS and ANSYS Workbench*.

The mixed u-P formulation is not needed in plane stress; the incompressibility condition is assumed, and the thickness is adjusted based on this incompressible assumption. For 2-D elements PLANE182 and PLANE183, using the mixed u-P formulation with either of the plane stress options [KEYOPT(3) = 0 or 3], ANSYS will automatically reset any KEYOPT(6) setting to 0 for pure displacement formulation.

2.16.3. Applications of Mixed u-P Formulations

Incompressible material behavior can be divided into two categories: fully incompressible materials and nearly incompressible materials. Typical fully incompressible materials are hyperelastic materials. You must choose the 18x elements with mixed u-P formulation to model this material behavior. For element SOLID187, you should set KEYOPT(6) = 1.

Nearly incompressible materials include hyperelastic materials and elastoplastic materials. The following 18x elements with mixed u-P formulation are available for nearly incompressible hyperelastic materials: PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, and SOLSH190. These have several material models available, including Mooney-Rivlin, Neo-Hookean, Polynomial, Gent, Arruda-Boyce, Ogden, and user-defined potential. See **TB**,HYPER for more details.

The best choice of these options varies from problem to problem. The general guidelines are:

- For material behavior with very small compressibility, use a 18x element with mixed u-P formulation.

- For mid compressibility, you can use the PLANE182/SOLID185 with $\bar{B}$ (most efficient), or the hyperelastic element (keep in mind that you are limited to only the Mooney-Rivlin material model), or use the 18x element with mixed u-P formulation.
- When deformation is highly confined, using a 18x element with mixed u-P formulation is recommended.

Nearly incompressible elastoplastic materials are materials with Poisson's ratio close to 0.5, or elastoplastic materials undergoing very large plastic deformation. For such cases, especially when the deformation is close to being fully incompressible, the mixed u-P formulation of the 18x elements is more robust, yielding better performance. However, you should try pure displacement formulation [KEYOPT(6) = 0] first since the extra pressure DOFs are involved in mixed formulation. If you are using mixed formulation with element SOLID187, it is recommended that you use KEYOPT(6) = 2.

2.16.4. Overconstrained Models and No Unique Solution

You should avoid overconstrained models when you are using elements with the mixed u-P formulation. An overconstrained model means that globally or locally, the number of displacement DOFs without any prescribed values, *Nd*, is less than the number of hydrostatic pressure DOFs, *Np*. *Np* can be calculated by the number of elements times the number of pressure DOFs within each element (see *Table 2.10, "Number of Independent Pressure DOFs in One Element"* for the specific numbers used with the 18x elements). If different type elements are included, *Np* is the summation of the products over each type of mixed formulation element. The overconstrained problem can be overcome by increasing the number of nodes without any displacement boundary conditions, or by refining the mesh. The optimal values for *Np* and *Nd* are *Nd*/*Np* = 2 for 2-D problems and *Nd*/*Np* = 3 for 3-D problems. See the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

In addition to overconstrained models, when the mixed u-P formulation of the 18x plane and solid elements is applied to fully incompressible hyperelastic materials, you must also avoid the "no unique solution" situation.

No unique solution occurs if all boundary nodes have prescribed displacements in each direction and the material is fully incompressible. Since hydrostatic pressure is independent from deformation, it should be determined by the force/pressure boundary condition. If no force/pressure boundary condition is applied, the hydrostatic pressure cannot be determined, or the solution is not unique. In this case, any stress field in equilibrium would be still in equilibrium and not cause any further deformation by adding an arbitrary value of hydrostatic pressure. This problem can be solved simply by having at least one node on the boundary in at least one direction without a prescribed displacement boundary condition. This direction cannot be the tangential direction of the boundary at this node. That means the solution is the particular one with zero force at the node in that direction where no displacement is prescribed.

2.17. Legacy vs. Current Element Technologies

If you are a long-time ANSYS user, your models may still employ some legacy element types. As technology advances, however, ANSYS continues to develop robust new element types *and* enhance certain existing elements with greater capabilities.

ANSYS recommends using a current element type in place of one or more legacy elements in your analysis wherever possible. Current element types are more feature-rich and use more advanced technologies than legacy elements. For example, support for the following capabilities is available only in current-technology elements:

- A vast array of material constitutive options (such as anisotropic hyperelasticity, cast iron plasticity, enhanced Drucker-Prager plasticity, hill plasticity, hyperelasticity, shape memory alloy, plasticity, rate-dependent plasticity, viscoelasticity, and others)
- Association of a single element with several material constitutive options (such as a combination of hyperelasticity and viscoelasticity with Prony series expansion)

- A curve-fitting tool (**TBFT**) for calibrating material parameters through experimental data, which currently supports creep, hyperelasticity, and viscoelasticity.
- Fracture mechanics parameter calculation (**CINT**), which uses the domain-integration approach to calculate the J-Integrals for both linear and elastoplastic material behavior at a designated tip location (2-D) or at specific location along the crack front (3-D) through a structure component (available in such elements as PLANE182, PLANE183, SOLID185, SOLID186, and SOLID187)
- ANSYS Variational Technology for optimization analysis, which quickly evaluates response surfaces for various type of applications (such as structural and frequency sweep analysis)
- Initial state conditions (**INISTATE**) incorporating the effects of initial stress that may exist in the structure due to previous loading (available in such elements as LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF265, and SHELL281)
- Custom user-defined material models created via the `UserMat` routine
- 3-D smeared reinforcing, provided by elements such as REINF265 when used with 3-D solid and shell elements (referred to as the base elements) to achieve the effect of extra reinforcement to those elements
- Nonlinear stabilization, a viscous-based algorithm for improving convergence behavior when instabilities are expected (available in such elements as LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, and SHELL281)
- Manual rezoning for solid elements (such as PLANE182 and PLANE183)
- Control of the element technologies (**ETCONTROL**) used in element formulation (for applicable elements such as SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209, and SHELL281)
- A layered section option (via **SECDATA** and other section commands) for shell and solid elements (such as SHELL181, SHELL281, SOLID185, SOLID186, SOLSH190, SHELL208, and SHELL209)

The following table lists legacy element types along with suggested current element types. In come cases, an element KEYOPT setting, degree-of-freedom (DOF) constraint, or ANSYS command may be necessary to closely reproduce the behavior of a given legacy element.

Legacy Element Type [*1*]	**Suggested Current Element Type [*2*]**	**Suggested Setting(s) to Approximate Legacy Element Behavior [*3*]**	
		Element KEY-OPT	**Comments**
BEAM3	BEAM188	KEYOPT(3) = 2	Constrain UZ, ROTX, and ROTY to simulate 2-D behavior. Issue **SECTYPE**,,BEAM.
BEAM4		KEYOPT(3) = 2	May require a finer mesh. Issue **SECTYPE**,,BEAM.
BEAM23		KEYOPT(3) = 2	Constrain UZ, ROTX, and ROTY to simulate 2-D behavior. Issue **SECTYPE**,,BEAM.
BEAM24		KEYOPT(3) = 2	Issue **SECTYPE**,,BEAM.
BEAM44		KEYOPT(3) = 2	Issue **SECTYPE**,,TAPER.
BEAM54		KEYOPT(3) = 2	Constrain UZ, ROTX, and ROTY to simulate 2-D behavior.

Legacy Element Type [*1*]	Suggested Current Element Type [*2*]	Suggested Setting(s) to Approximate Legacy Element Behavior [*3*]	
		Element KEY-OPT	Comments
			Issue **SECTYPE**,,TAPER.
COMBIN7	MPC184	KEYOPT(1) = 6	MPC184 uses the Lagrange multiplier method.
CONTAC12	CONTA178	---	Constrain UZ to simulate 2-D behavior.
CONTAC52		---	Constrain UZ to simulate 2-D behavior.
FLUID141	ANSYS CFX-Flo	---	---
FLUID142		---	---
INFIN9	INFIN110	---	---
INFIN47	INFIN111	---	---
LINK1	LINK180	---	---
LINK8		---	---
PLANE2	PLANE183	KEYOPT(1) = 1	---
PLANE13	PLANE223	---	---
PLANE67		KEYOPT(1) = 110	---
PLANE42	PLANE182	KEYOPT(1) = 3	---
PLANE82	PLANE183	---	---
SHELL41	SHELL181	KEYOPT(1) = 1, KEYOPT(3) = 2	---
SHELL43		KEYOPT(3) = 2	---
SHELL63		KEYOPT(3) = 2	May require a finer mesh.
SHELL57	SHELL131	KEYOPT(3) = 2	Issue **SECTYPE**,,SHELL.
SHELL91	SHELL281	---	Issue **SECTYPE**,,SHELL.
SHELL93		---	---
SHELL99		---	Issue **SECTYPE**,,SHELL.
SOLID5	SOLID226	---	---
SOLID69		KEYOPT(1) = 110	---
SOLID45	SOLID185	KEYOPT(2) = 3	---
SOLID46		KEYOPT(3) = 1	Issue **SECTYPE**,,SHELL.
SOLID92	SOLID187	---	---
SOLID95	SOLID186	KEYOPT(2) = 1 KEYOPT(2) = 0 for nonlinear analysis	---
SOLID191		KEYOPT(3) = 1	Issue **SECTYPE**,,SHELL.
SOLID98	SOLID227	---	---
VISCO88	PLANE183	---	Issue **TB**,PRONY and **TB**,SHIFT.
VISCO89	SOLID186	KEYOPT(2) = 1 KEYOPT(2) = 0 for nonlinear analysis	Issue **TB**,PRONY and **TB**,SHIFT.

1. ANSYS may undocument certain legacy elements in future releases after considering element redundancy and consistency issues; however, the table does not imply that all legacy elements listed are targets for such action.
2. The table is not a definitive listing of legacy-to-current element equivalents in terms of either formulation or use of shape functions. For example, a suggested current element may require a more refined mesh in some cases, or may require adaptation via appropriate constraints for specific 2-D analyses.
3. While a given KEYOPT setting can allow you to approximate the behavior of a *legacy* element, it may not be the most desirable KEYOPT for the *current* element. For structural-only analyses, try the **ETCONTROL** command for element and KEYOPT recommendations. For more information, see *Section 2.18: Automatic Selection of Element Technologies*.

2.18. Automatic Selection of Element Technologies

With the variety of technologies available in many elements, choosing the best settings to solve your problem efficiently can be challenging. Particularly in the newer structural and solid elements for stress analysis, an element might offer two to four technologies.

ANSYS can offer suggestions (or reset the technology settings) to help you achieve the best solution via the **ETCONTROL** command. The command helps you to select the appropriate element technologies by considering the stress state, the options set on the element type, and the associated constitutive models of the element type. It works for all current-technology elements such as SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209.

The materials associated with each element type are detected at the solution stage and are classified as:

- Elastoplastic materials, including:
  - Linear elastic materials with Poisson's ratio <= 0.49
  - Linear elastic materials with Poisson's ratio > 0.49
  - Anisotropic materials
  - Elastoplastic materials (other nonlinear materials except hyperelastic materials)
- Hyperelastic materials, including:
  - Nearly incompressible hyperelastic materials
  - Fully incompressible hyperelastic materials

In practical application, one element type could be associated with more than one constitutive model, for example, linear elastic and elastoplastic materials. For this case, the suggestions/resettings are based on elastoplastic materials, which need more complicated element technology. The list of constitutive models above is organized in order of complexity, with the bottom ones more difficult to solve numerically. When an element type is associated with multiple constitutive models, the more complex one is used in the automatic selection of the element technology.

When using **ETCONTROL** the suggestions given or settings changed are described in the tables below.

Table 2.11 Recommendation Criteria for Element Technology (Linear Material)

Linear Materials Only			
Element	**Stress State / Option**	**Poisson's ratio (ν) <= 0.49**	**Poisson's ratio (ν) > 0.49 (or anisotropic materials)**
SHELL181	Default	KEYOPT(3) = 2 for higher accuracy of membrane stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 2 for all cases	
	Membrane-only option (KEYOPT(1) = 1)	KEYOPT(3) = 2 for higher accuracy of stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 0 for all cases	
PLANE182	Plane stress	KEYOPT(1) = 2	
	Not plane stress	KEYOPT(1) = 3	KEYOPT(1) = 2
PLANE183	Plane stress	No change. (Default element settings are best.)	
	Not plane stress	No change. (Default element settings are best.)	
SOLID185		KEYOPT(2) = 3	KEYOPT(2) = 2
SOLID186		KEYOPT(2) = 0	
SOLID187		No change. (Default element settings are best.)	
BEAM188		KEYOPT(3) = 2, KEYOPT(1) = 1[*1*]	
BEAM189		KEYOPT(1) = 1[*1*]	
SHELL208		KEYOPT(8) = 2	
SHELL209		KEYOPT(8) = 2	
SHELL281		No change. (Default element settings are best.)	

1. Only when the beam section is not circular, otherwise KEYOPT(1) = 0

Table 2.12 Recommendation Criteria for Element Technology (Nonlinear Materials)

Nonlinear Materials			
Element	**Stress State / Option**	**Elastoplastic materials (may also have hyperelastic materials)**	**Hyperelastic materials only**
SHELL181	Default	KEYOPT(3) = 2 for higher accuracy of membrane stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 2 for all cases	KEYOPT(3) = 0, KEYOPT(8) = 0
	Membrane-only option (KEYOPT(1) = 1)	KEYOPT(3) = 2 for higher accuracy of stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 0 for all cases	KEYOPT(3) = 0, KEYOPT(8) = 0
PLANE182	Plane stress	KEYOPT(1) = 2	KEYOPT(1) = 0
	Not plane stress	KEYOPT(1) = 2[*1*]	KEYOPT(1) = 0[*1*]
PLANE183	Plane stress	No change. (Default element settings are best.)	
	Not plane stress	no change (default element settings are best)[*1*]	
SOLID185		KEYOPT(2) = 2[*1*]	KEYOPT(2) = 0[*1*]
SOLID186		KEYOPT(2) = 0[*1*]	
SOLID187		No change. (Default element settings are best.)[*1*]	
BEAM188		KEYOPT(3) = 2, KEYOPT(1) = 1[*2*]	

Nonlinear Materials			
Element	**Stress State / Option**	**Elastoplastic materials (may also have hyperelastic materials)**	**Hyperelastic materials only**
BEAM189		KEYOPT(1) = 1[*2*]	
SHELL208		KEYOPT(8) = 2	KEYOPT(8) = 0
SHELL209		KEYOPT(8) = 2	KEYOPT(8) = 0
SHELL281		No change. (Default element settings are best.)	

1. If even only one of your hyperelastic materials is fully incompressible, then KEYOPT(6) = 1 must also be used.
2. Only when the beam section is not circular, otherwise KEYOPT(1) = 0.

For the solid elements listed above, **ETCONTROL** provides suggestions or resets the KEYOPTs for the element technology settings (i.e., KEYOPT(1) for PLANE182, KEYOPT(2) for SOLID185 and SOLID186). The Mixed u-P formulation KEYOPT(6) is reset when necessary; that is, when fully incompressible hyperelastic materials are associated with non-plane stress states.

For BEAM188 and BEAM189, KEYOPT(1) is always suggested for non-circular cross section beams and should be reset when **ETCONTROL**,SET,.. is issued. However, the KEYOPT setting changes the number of DOFs at each node, so it must be set before you issue any **D**, **DK**, **DA**, and similar commands. Because the **ETCONTROL** reset is done at the beginning of the solution stage, ANSYS displays a message stating that you should change KEYOPT(1) to 1, but it does not make the change automatically.

For SHELL181, the setting for KEYOPT(3) depends on your problem and your purpose if the element type is associated with any non-hyperelastic materials. In this case, ANSYS cannot reset it automatically, but it displays a message with the recommended setting (even if you used **ETCONTROL**,SET,..). You should reset this manually if the defined criteria matches your problem.

If an element type is defined but not associated with any material, no suggestions or resets are done for that element type. The stress states and options are mentioned only when they affect the suggestions or settings. The suggestions are available in output.

All nonlinear stress/strain data should be input as true-stress/true-(logarithmic-) strain. Accordingly, all output data is also indicated as true-stress/true-strain. For small strains, the true-stress/true-strain data and engineering-stress/engineering-strain data are essentially identical.

Chapter 3: Element Characteristics

The ANSYS program has a large library of element types. Some of the characteristics of the element types, and their groupings, are described in this chapter to make element type selection easier. The detailed element type descriptions are given in *Chapter 4: Element Library*.

The following element characteristic topics are available:

3.1. Element Classifications
3.2. Pictorial Summary
3.3. GUI-Inaccessible Elements

The ANSYS element library consists of almost 200 different element formulations or types. An element type is identified by a name (8 characters maximum), such as BEAM3, consisting of a group label (BEAM) and a unique identifying number (3). The element descriptions in *Chapter 4: Element Library* are arranged in order of these identification numbers. The element is selected from the library for use in the analysis by inputting its name on the element type command [**ET**].

2-D versus 3-D Models

ANSYS models may be either 2-D or 3-D depending upon the element types used. A 2-D model must be defined in an X-Y plane. They are easier to set up, and run faster than equivalent 3-D models. Axisymmetric models are also considered to be 2-D.

If any 3-D element type (such as BEAM4) is included in the element type [**ET**] set, the model becomes 3-D. Some element types (such as COMBIN14) may be 2-D or 3-D, depending upon the KEYOPT value selected. Other element types (such as COMBIN40) have no influence in determining the model dimensions. A 2-D element type may be used (with caution) in 3-D models.

Element Characteristic Shape

In general, four shapes are possible: point, line, area, or volume. A point element is typically defined by one node, e.g., a mass element. A line element is typically represented by a line or arc connecting two or three nodes. Examples are beams, spars, pipes, and axisymmetric shells. An area element has a triangular or quadrilateral shape and may be a 2-D solid element or a shell element. A volume element has a tetrahedral or brick shape and is usually a 3-D solid element.

Degrees of Freedom and Discipline

The degrees of freedom of the element determine the discipline for which the element is applicable: structural, thermal, fluid, electric, magnetic, or coupled-field. The element type should be chosen such that the degrees of freedom are sufficient to characterize the model's response. Including unnecessary degrees of freedom increases the solution memory requirements and running time. Similarly, selecting element types with unnecessary features, such as using an element type with plastic capability in an elastic solution, also unnecessarily increases the analysis run time.

User Elements

You may also create your own element type and use it in an analysis as a *user element*. User elements and other user programmable features (UPFs) are described in the *Guide to ANSYS User Programmable Features*.

3.1. Element Classifications

Table 3.1 List of Elements by Classification

Classification		Elements
Structural Point		MASS21
Structural Line	**2-D**	LINK1
	3-D	LINK8 , LINK10, LINK11, LINK180
Structural Beam	**2-D**	BEAM3, BEAM23, BEAM54
	3-D	BEAM4, BEAM24, BEAM44, BEAM188, BEAM189
Structural Solid	**2-D**	PLANE25, PLANE42, PLANE82, PLANE83, PLANE145, PLANE146, PLANE182, PLANE183
	3-D	SOLID45, SOLID65, SOLID92, SOLID95, SOLID147, SOLID148, SOLID185, SOLID186, SOLID187
Structural Shell	**2-D**	SHELL61, SHELL208, SHELL209
	3-D	SHELL28, SHELL41, SHELL43, SHELL63, SHELL93, SHELL150, SHELL181, SHELL281
Structural Solid Shell	**3-D**	SOLSH190
Structural Pipe		PIPE16, PIPE17, PIPE18, PIPE20, PIPE59, PIPE60
Structural Interface		INTER192, INTER193, INTER194, INTER195, INTER202, INTER203, INTER204, INTER205
Structural Multipoint Constraint Elements		MPC184, MPC184-Link/Beam, MPC184-Slider, MPC184-Revolute, MPC184-Universal, MPC184-Slot, MPC184-Point, MPC184-Trans, MPC184-Cylin, MPC184-Planar, MPC184-Weld, MPC184-Orient, MPC184-Spherical, MPC184-General
Structural Layered Composite		SOLID46, SHELL91, SHELL99, SOLID185 Layered Solid, SOLID186 Layered Solid, SOLSH190, SOLID191
Explicit Dynamics		LINK160, BEAM161, PLANE162, SHELL163, SOLID164, COMBI165, MASS166, LINK167, SOLID168
Visco Solid		VISCO88, VISCO89, VISCO106, VISCO107, VISCO108
Thermal Point		MASS71
Thermal Line		LINK31, LINK32, LINK33, LINK34
Thermal Solid	**2-D**	PLANE35, PLANE55, PLANE75, PLANE77, PLANE78
	3-D	SOLID70, SOLID87, SOLID90
Thermal Shell		SHELL57, SHELL131, SHELL132
Thermal Electric		PLANE67, LINK68, SOLID69, SHELL157
Fluid		FLUID29, FLUID30, FLUID38, FLUID79, FLUID80, FLUID81, FLUID116, FLUID129, FLUID130, FLUID136, FLUID138, FLUID139, FLUID141, FLUID142
Magnetic Electric		PLANE53, SOLID96, SOLID97, INTER115, SOLID117, HF118, HF119, HF120, PLANE121, SOLID122, SOLID123, SOLID127, SOLID128, PLANE230, SOLID231, SOLID232
Electric Circuit		SOURC36, CIRCU94, CIRCU124, CIRCU125
Electromechanical		TRANS109, TRANS126
Coupled-Field		SOLID5, PLANE13, SOLID62, SOLID98, ROM144, PLANE223, SOLID226, SOLID227

Classification		Elements
Contact		CONTAC12, CONTAC52, TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, CONTA178
Combination		COMBIN7, COMBIN14, COMBIN37, COMBIN39, COMBIN40, COMBI214, PRETS179
Matrix		MATRIX27, MATRIX50
Infinite		INFIN9, INFIN47, INFIN110, INFIN111
Surface		SURF151, SURF152, SURF153, SURF154, SURF156, SURF251, SURF252
Follower Load		FOLLW201
Meshing		MESH200
Reinforcing		REINF265

3.2. Pictorial Summary

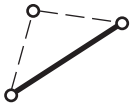
The following tables contain numerically listed elements by group name with graphic pictorial, element description, and product availability.

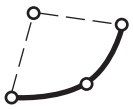
BEAM Elements	CIRCU Elements	COMBIN Elements	CONTAC Elements
FLUID Elements	FOLLW Elements	HF Elements	INFIN Elements
INTER Elements	LINK Elements	MASS Elements	MATRIX Elements
MESH Elements	MPC Elements	PIPE Elements	PLANE Elements
PRETS Elements	REINF Elements	ROM Elements	SHELL Elements
SOLID Elements	SOLSH Elements	SOURC Elements	SURF Elements
TARGE Elements	TRANS Elements	VISCO Elements	

The codes represent each of the products in the ANSYS suite of products:

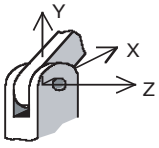
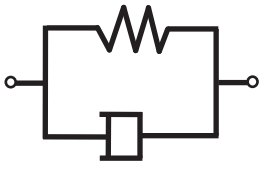
Code	Product	Code	Product
MP	ANSYS Multiphysics	EM	ANSYS Emag - Low Frequency
ME	ANSYS Mechanical	EH	ANSYS Emag - High Frequency
ST	ANSYS Structural	FL	ANSYS FLOTRAN
PR	ANSYS Professional - Nonlinear Thermal	DS	ANSYS DesignSpace
PRN	ANSYS Professional - Nonlinear Structural	DSS	ANSYS DesignSpace - Structural
DY	ANSYS LS-DYNA	PP	ANSYS PrepPost
VT	ANSYS DesignXplorer		

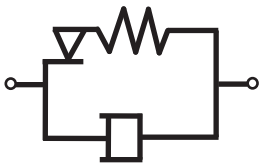
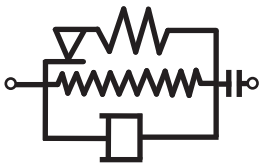
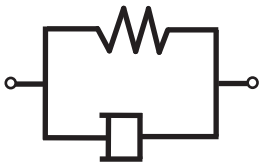
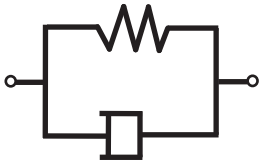
BEAM Elements	Product Codes
BEAM3 Structural 2-D Elastic Beam 2 nodes 2-D space DOF: UX, UY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
BEAM4 Structural 3-D Elastic Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

BEAM Elements	Product Codes
	
BEAM23 Structural 2-D Plastic Beam 2 nodes 2-D space DOF: UX, UY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
BEAM24 Structural 3-D Thin-Walled Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
BEAM44 Structural 3-D Elastic Tapered Unsymmetric Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
BEAM54 Structural 2-D Elastic Tapered Unsymmetric Beam 2 nodes 2-D space DOF: UX, UY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
BEAM161 Explicit Dynamics 3-D Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, AX, AY, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>
BEAM188 Structural 3-D Linear Finite Strain Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
BEAM189 Structural 3-D Quadratic Finite Strain Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	MP ME ST PR PRN DS DSS <> <> <> <> PP VT

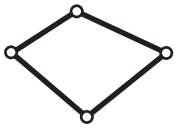
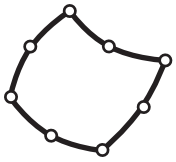
BEAM Elements	Product Codes
	

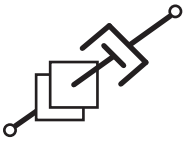
CIRCU Elements	Product Codes
CIRCU94 Coupled-Field Piezoelectric Circuit 2 or 3 nodes 2-D space DOF: VOLT, CURR 	MP <> <> <> <> <> <> <> <> <> PP <>
CIRCU124 Magnetic Electric Circuit 2-6 nodes 3-D space DOF: VOLT, CURR, EMF 	MP <> <> <> <> <> <> <> EM <> <> PP <>
CIRCU125 Magnetic Electric Diode 2 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>

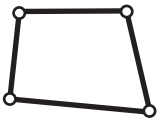
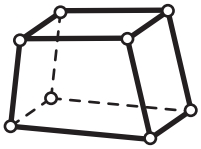
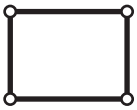
COMBIN Elements	Product Codes
COMBIN7 Combination Revolute Joint 5 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
COMBIN14 Combination Spring-Damper 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS <> <> <> <> <> PP VT
COMBIN37 Combination Control 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP	MP ME ST <> <> <> <> <> <> <> <> PP <>

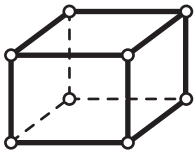
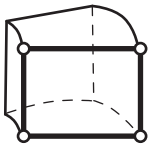
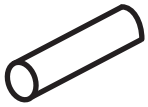
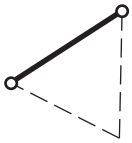
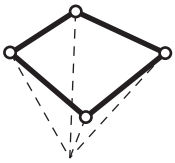
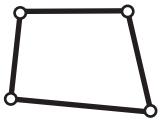
COMBIN Elements	Product Codes
	
COMBIN39 Combination Nonlinear Spring 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP 	MP ME ST <> <> <> <> <> <> <> <> PP <>
COMBIN40 Combination 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
COMBI165 Explicit Dynamics Spring-Damper 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, VZ, AX, AV, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>
COMBI214 Combination Spring-Damper Bearing 2 nodes 2-D space DOF: UX, UY, UZ, 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

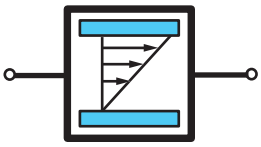
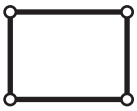
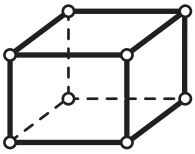
CONTAC Elements	Product Codes
CONTAC12 2-D Point-to-Point Contact 2 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN <> <> <> <> <> <> PP <>

CONTAC Elements	Product Codes
CONTAC52 3-D Point-to-Point Contact 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
CONTA171 2-D 2-Node Surface-to-Surface Contact 2 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>
CONTA172 2-D 3-Node Surface-to-Surface Contact 3 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>
CONTA173 3-D 4-Node Surface-to-Surface Contact 4 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG 	MP ME ST PR PRN DS <> <> <> <> <> PP <>
CONTA174 3-D 8-Node Surface-to-Surface Contact 8 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>
CONTA175 2-D/3-D Node-to-Surface Contact 1 node 2-D/3-D space DOF: UX, UY, UZ, TEMP, VOLT, AX, MAG 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>
CONTA176 3-D Line-to-Line Contact 3 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>
CONTA177	MP ME ST PR PRN <> <> <> <> <> <> PP <>

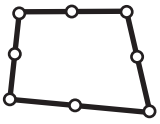
CONTAC Elements	Product Codes
3-D Line-to-Surface Contact 3 nodes 3-D space DOF: UX, UY, UZ 	
CONTA178 3-D Node-to-Node Contact 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>

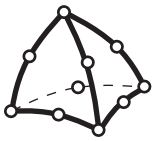
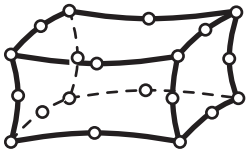
FLUID Elements	Product Codes
FLUID29 2-D Acoustic Fluid 4 nodes 2-D space DOF: UX, UY, PRES 	MP ME <> <> <> <> <> <> <> <> <> PP <>
FLUID30 3-D Acoustic Fluid 8 nodes 3-D space DOF: UX, UY, UZ, PRES 	MP ME <> <> <> <> <> <> <> <> <> PP <>
FLUID38 Dynamic Fluid Coupling 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
FLUID79 2-D Contained Fluid 4 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>
FLUID80 3-D Contained Fluid	MP ME ST <> <> <> <> <> <> <> <> PP <>

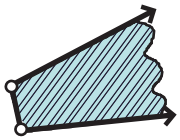
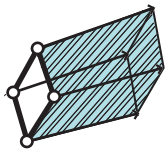
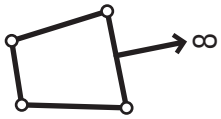
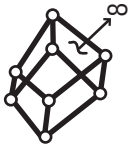
FLUID Elements	Product Codes
8 nodes 3-D space DOF: UX, UY, UZ 	
FLUID81 Axisymmetric-Harmonic Contained Fluid 4 nodes 2-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> PP <>
FLUID116 Coupled Thermal-Fluid Pipe 2 nodes 3-D space DOF: PRES, TEMP 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
FLUID129 2-D Infinite Acoustic 2 nodes 2-D space DOF: PRES 	MP ME <> <> <> <> <> <> <> <> <> PP <>
FLUID130 3-D Infinite Acoustic 4 nodes 3-D space DOF: PRES 	MP ME <> <> <> <> <> <> <> <> <> PP <>
FLUID136 3-D Squeeze Film Fluid 4, 8 nodes 3-D space DOF: PRES 	MP ME <> <> <> <> <> <> <> <> <> PP <>
FLUID138 3-D Viscous Fluid Link 2 nodes 3-D space	MP ME <> <> <> <> <> <> <> <> <> PP <>

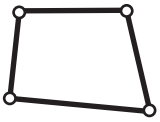
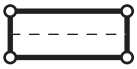
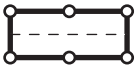
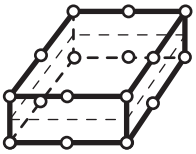
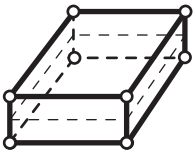
FLUID Elements	Product Codes
DOF: PRES 	
FLUID139 3-D Slide Film Fluid 2, 32 nodes 3-D space DOF: UX, UY, UZ 	MP ME <> <> <> <> <> <> <> <> PP <>
FLUID141 2-D Fluid-Thermal 4 nodes 2-D space DOF: VX, VY, VZ, PRES, TEMP, ENKE, ENDS 	MP <> <> <> <> <> <> FL <> <> <> PP <>
FLUID142 3-D Fluid-Thermal 8 nodes 3-D space DOF: VX, VY, VZ, PRES, TEMP, ENKE, ENDS 	MP <> <> <> <> <> <> FL <> <> <> PP <>

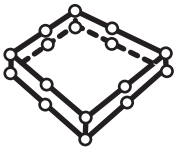
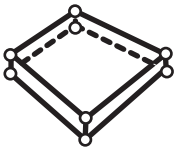
FOLLW Elements	Product Codes
FOLLW201 3-D Follower Load 1 node 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

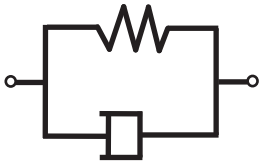
HF Elements	Product Codes
HF118 2-D High-Frequency Magnetic Electric Quadrilateral Solid 8 nodes 2-D space DOF: AX 	MP <> <> <> <> <> <> <> <> EH <> PP <>
HF119 3-D High-Frequency Magnetic Electric Tetrahedral Solid 10 nodes 3-D space	MP <> <> <> <> <> <> <> <> EH <> PP <>

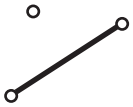
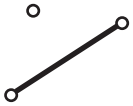
HF Elements	Product Codes
DOF: AX 	
HF120 3-D High-Frequency Magnetic Electric Brick Solid 20 nodes 3-D space DOF: AX 	MP <> <> <> <> <> <> <> EH <> PP <>

INFIN Elements	Product Codes
INFIN9 2-D Infinite Boundary 2 nodes 2-D space DOF: AZ, TEMP 	MP ME <> <> <> <> <> <> EM <> <> PP <>
INFIN47 3-D Infinite Boundary 4 nodes 3-D space DOF: MAG, TEMP 	MP ME <> <> <> <> <> <> EM <> <> PP <>
INFIN110 2-D Infinite Solid 4 or 8 nodes 2-D space DOF: AZ, VOLT, TEMP 	MP ME <> <> <> <> <> <> EM <> <> PP <>
INFIN111 3-D Infinite Solid 8 or 20 nodes 3-D space DOF: MAG, AX, AY, AZ, VOLT, TEMP 	MP ME <> <> <> <> <> <> EM <> <> PP <>

INTER Elements	Product Codes
INTER115 3-D Magnetic Electric Interface 4 nodes 3-D space DOF: AX, AY, AZ, MAG 	MP <> <> <> <> <> <> <> EM <> <> PP <>
INTER192 Structural 2-D Interface 4-Node Gasket 4 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER193 Structural 2-D Interface 6-Node Gasket 6 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER194 Structural 3-D Interface 16-Node Gasket 16 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER195 Structural 3-D Interface 8-Node Gasket 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER202 Structural 2-D Interface 4-Node Cohesive Zone 4 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER203 Structural 2-D Interface 6-Node Cohesive Zone 6 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>

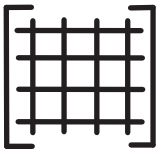
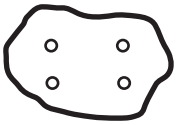
INTER Elements	Product Codes
INTER204 Structural 3-D Interface 16-Node Cohesive Zone 16 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
INTER205 Structural 3-D Interface 8-Node Cohesive Zone 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>

LINK Elements	Product Codes
LINK1 Structural 2-D Spar (or Truss) 2 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>
LINK8 Structural 3-D Spar (or Truss) 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
LINK10 Structural 3-D Tension-only or Compression-only Spar 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
LINK11 Structural 3-D Linear Actuator 2 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
LINK31	MP ME <> PR PRN <> <> <> <> <> <> PP <>

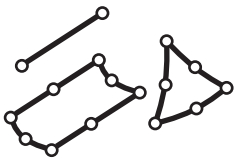
LINK Elements	Product Codes
Radiation Link 2 nodes 3-D space DOF: TEMP 	
LINK32 Thermal 2-D Conduction Bar 2 nodes 2-D space DOF: TEMP 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
LINK33 Thermal 3-D Conduction Bar 2 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP <>
LINK34 Convection Link 2 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
LINK68 Coupled Thermal-Electric Line 2 nodes 3-D space DOF: TEMP, VOLT 	MP ME <> PR PRN <> <> <> EM <> <> PP <>
LINK160 Explicit 3-D Spar (or Truss) 3 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>
LINK167 Explicit Tension-Only Spar 3 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>
LINK180 Structural 3-D Finite Strain Spar (or Truss)	MP ME ST PR PRN <> <> <> <> <> <> PP VT

LINK Elements	Product Codes
2 nodes 3-D space DOF: UX, UY, UZ 	

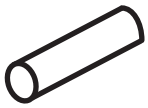
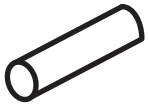
MASS Elements	Product Codes
MASS21 Structural Mass 1 node 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ ●	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
MASS71 Thermal Mass 1 node 3-D space DOF: TEMP ●	MP ME <> PR PRN DS <> <> <> <> <> PP <>
MASS166 Explicit 3-D Structural Mass 1 node 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ ●	<> <> <> <> <> <> <> <> <> <> DY <> <>

MATRIX Elements	Product Codes
MATRIX27 Stiffness, Damping, or Mass Matrix 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
MATRIX50 Superelement (or Substructure) 2-D or 3-D space DOF: Determined from included element types 	MP ME ST PR PRN <> <> <> <> <> <> PP <>

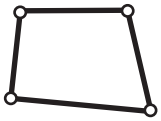
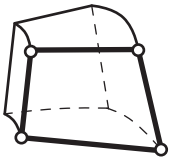
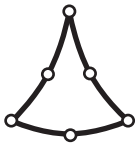
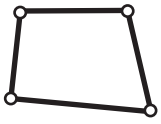
MESH Elements	Product Codes
MESH200 Meshing Facet 2-20 nodes 2-D/3-D space DOF: None KEYOPT Dependent	MP ME ST PR PRN <> <> <> EM EH DY PP <>

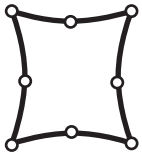
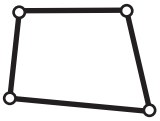
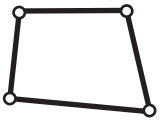
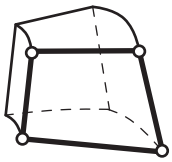
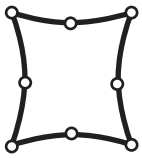
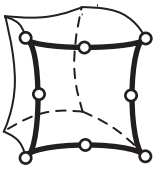
MESH Elements	Product Codes
	

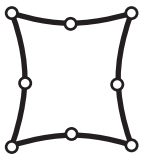
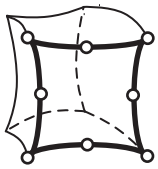
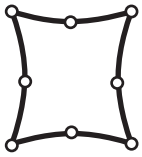
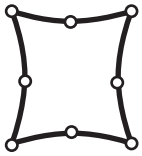
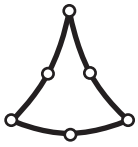
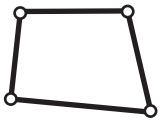
MPC Elements	Product Codes
MPC184 Structural Multipoint Constraint 2 or 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ KEYOPT Dependent 	MP ME ST PR PRN <> <> <> <> <> <> PP <>

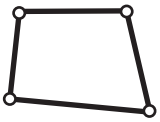
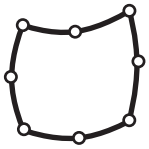
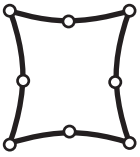
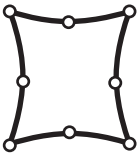
PIPE Elements	Product Codes
PIPE16 Structural Elastic Straight Pipe 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
PIPE17 Structural Elastic Pipe Tee 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
PIPE18 Structural Elastic Curved Pipe (Elbow) 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
PIPE20 Structural Plastic Straight Pipe 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
PIPE59 Structural Immersed Pipe or Cable 2 nodes 3-D space	MP ME ST <> <> <> <> <> <> <> <> PP <>

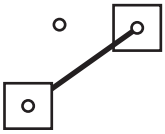
PIPE Elements	Product Codes
DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	
PIPE60 Structural Plastic Curved Pipe (Elbow) 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>

PLANE Elements	Product Codes
PLANE13 2-D Coupled-Field Solid 4 nodes 2-D space DOF: TEMP, AZ, UX, UY, VOLT 	MP ME <> <> <> <> <> <> EM <> <> PP <>
PLANE25 Axisymmetric-Harmonic 4-Node Structural Solid 4 nodes 2-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
PLANE35 2-D 6-Node Triangular Thermal Solid 6 nodes 2-D space DOF: TEMP 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
PLANE42 2-D Structural Solid 4 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>
PLANE53 2-D 8-Node Magnetic Solid 8 nodes 2-D space DOF: VOLT, AZ, CURR, EMF	MP <> <> <> <> <> <> <> EM <> <> PP <>

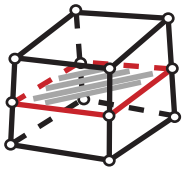
PLANE Elements	Product Codes
	
PLANE55 2-D Thermal Solid 4 nodes 2-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP <>
PLANE67 2-D Coupled Thermal-Electric Solid 4 nodes 2-D space DOF: TEMP, VOLT 	MP ME <> PR PRN <> <> <> EM <> <> PP <>
PLANE75 2-D Axisymmetric-Harmonic 4-Node Thermal Solid 4 nodes 2-D space DOF: TEMP 	MP ME <> <> <> <> <> <> <> <> <> PP <>
PLANE77 2-D 8-Node Thermal Solid 8 nodes 2-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP <>
PLANE78 2-D Axisymmetric - Harmonic 8-Node Thermal Solid 8 nodes 2-D space DOF: TEMP 	MP ME <> <> <> <> <> <> <> <> <> PP <>
PLANE82 2-D 8-Node Structural Solid 8 nodes 2-D space DOF: UX, UY	MP ME ST PR PRN DS <> <> <> <> <> PP <>

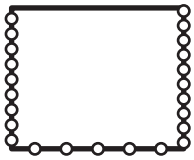
PLANE Elements	Product Codes
	
PLANE83 2-D Axisymmetric - Harmonic 8-Node Structural Solid 8 nodes 2-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> PP <>
PLANE121 2-D 8-Node Electrostatic Solid 8 nodes 2-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>
PLANE145 2-D Quadrilateral Structural Solid p-Element 8 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
PLANE146 2-D Triangular Structural Solid p-Element 6 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
PLANE162 Explicit 2-D Structural Solid 4 nodes 2-D space DOF: UX, UY, VX, VY, AX, AY 	<> <> <> <> <> <> <> <> <> <> DY <> <>
PLANE182 2-D 4-Node Structural Solid	MP ME ST PR PRN DS DSS <> <> <> <> PP VT

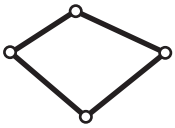
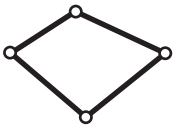
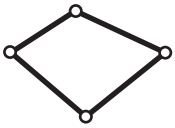
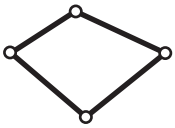
PLANE Elements	Product Codes
4 nodes 2-D space DOF: UX, UY 	
PLANE183 2-D 8-Node Structural Solid 8 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
PLANE223 2-D 8-Node Coupled-Field Solid 8 nodes 2-D space DOF: UX, UY, TEMP, VOLT 	MP <> <> <> <> <> <> <> <> <> <> PP <>
PLANE230 2-D 8-Node Electric Solid 8 nodes 2-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>

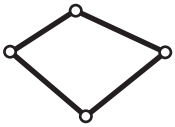
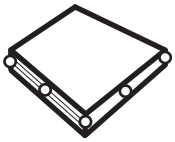
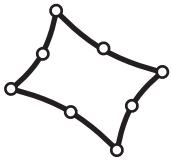
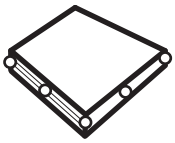
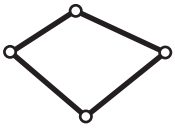
PRETS Elements	Product Codes
PRETS179 2-D/3-D Pretension Combination 3 nodes 2-D/3-D space DOF: UX 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

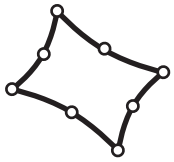
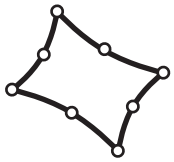
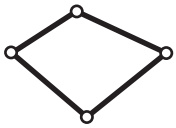
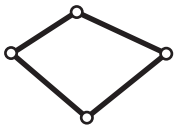
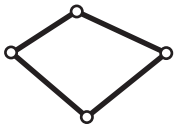
REINF Elements	Product Code
REINF265 3-D Smeared Reinforcing Up to 20 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	MP ME ST PR PRN <> <> <> <> <> <> PP <>

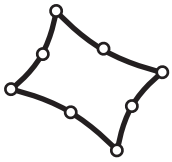
REINF Elements	Product Code
	

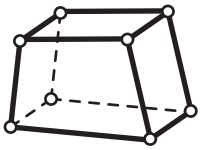
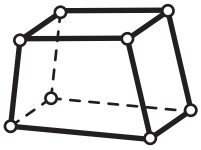
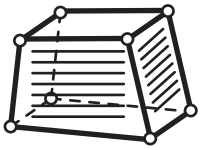
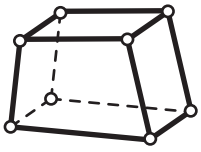
ROM Elements	Product Code
ROM144 Reduced Order Electrostatic-Structural Coupled-Field 20 or 30 nodes 3-D space DOF: EMF, VOLT, UX 	MP <> <> <> <> <> <> <> <> <> PP <>

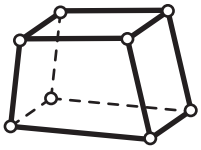
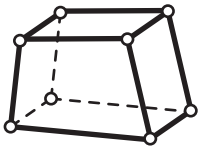
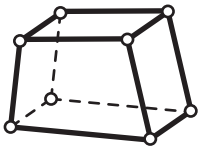
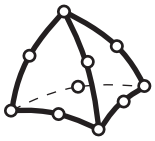
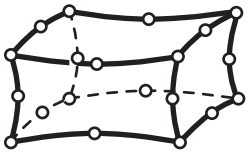
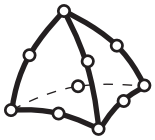
SHELL Elements	Product Codes
SHELL28 Structural 3-D Shear/Twist Panel 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SHELL41 Structural 3-D Membrane Shell 4 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SHELL43 3-D Structural 4-Node Plastic Large Strain Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
SHELL57 Thermal Shell 4 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP <>
SHELL61	MP ME ST <> <> <> <> <> <> <> <> PP <>

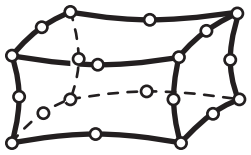
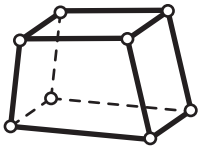
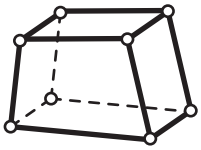
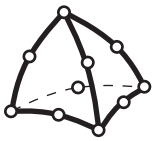
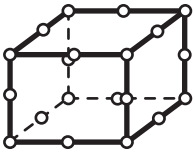
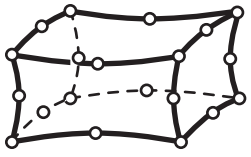
SHELL Elements	Product Codes
2-D Axisymmetric - Harmonic Structural Shell 2 nodes 2-D space DOF: UX, UY, UZ, ROTZ 	
SHELL63 Structural 3-D Elastic Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS <> <> <> <> <> PP <>
SHELL91 3-D Nonlinear Layered Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
SHELL93 3-D 8-Node Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS <> <> <> <> <> PP <>
SHELL99 3-D Linear Layered Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SHELL131 4 Node Layered Thermal Shell 4 nodes 3-D space DOF: TBOT, TE2, TE3, TE4, . . . TTOP 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
SHELL132 8 Node Layered Thermal Shell	MP ME <> PR PRN <> <> <> <> <> <> PP <>

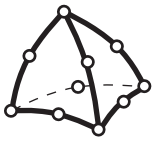
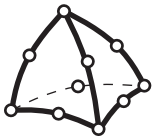
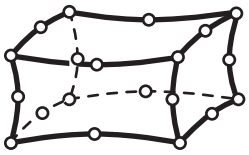
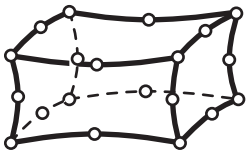
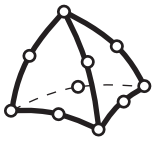
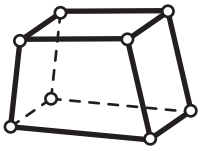
SHELL Elements	Product Codes
8 nodes 3-D space DOF: TBOT, TE2, TE3, TE4, . . . TTOP 	
SHELL150 3-D 8-Node Structural Shell p-Element 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SHELL157 Thermal-Electric Shell 4 nodes 3-D space DOF: TEMP, VOLT 	MP ME <> PR PRN <> <> <> EM <> <> PP <>
SHELL163 Explicit Thin Structural Shell 4 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ, 	<> <> <> <> <> <> <> <> <> <> DY <> <>
SHELL181 3-D Structural 4-Node Finite Strain Layered Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SHELL208 2-D Axisymmetric Structural Shell 2 nodes 2-D space DOF: UX, UY, ROTZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SHELL209 2-D Axisymmetric Structural Shell	MP ME ST PR PRN <> <> <> <> <> <> PP <>

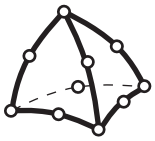
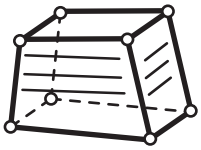
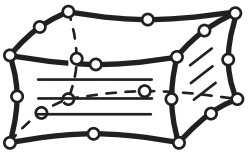
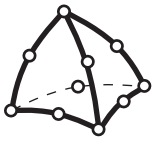
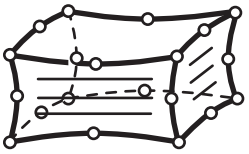
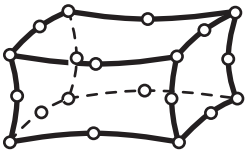
SHELL Elements	Product Codes
3 nodes 2-D space DOF: UX, UY, ROTZ 	
SHELL281 3-D Structural 8-Node Finite Strain Layered Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

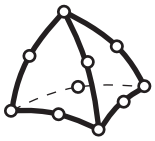
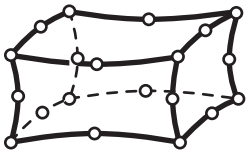
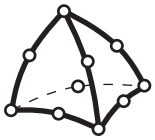
SOLID Elements	Product Codes
SOLID5 3-D Coupled-Field Solid 8 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG 	MP ME <> <> <> <> <> <> EM <> <> PP <>
SOLID45 3-D Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>
SOLID46 3-D 8-Node Layered Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SOLID62 3-D Magneto-Structural Coupled-Field Solid 8 nodes 3-D space DOF: UX, UY, UZ, AX, AY, AZ, VOLT 	MP <> <> <> <> <> <> <> <> <> <> PP <>

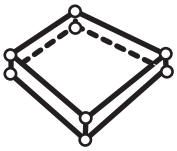
SOLID Elements	Product Codes
SOLID65 3-D Reinforced Concrete Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
SOLID69 3-D Coupled Thermal-Electric Solid 8 nodes 3-D space DOF: TEMP, VOLT 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
SOLID70 3-D Thermal Solid 8 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP VT
SOLID87 3-D 10-Node Tetrahedral Thermal Solid 10 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP VT
SOLID90 3-D 20-Node Thermal Solid 20 nodes 3-D space DOF: TEMP 	MP ME <> PR PRN DS <> <> <> <> <> PP VT
SOLID92 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

SOLID Elements	Product Codes
SOLID95 3-D 20-Node Structural Solid 20 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>
SOLID96 3-D Magnetic Scalar Solid 8 nodes 3-D space DOF: MAG 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID97 3-D Magnetic Solid 8 nodes 3-D space DOF: AX, AY, AZ, VOLT, CURR, EMF 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID98 Tetrahedral Coupled-Field Solid 10 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG 	MP ME <> <> <> <> <> <> EM <> <> PP <>
SOLID117 3-D 20-Node Magnetic Edge 20 nodes 3-D space DOF: AZ 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID122 3-D 20-Node Electrostatic Solid 20 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>

SOLID Elements	Product Codes
SOLID123 3-D 10-Node Tetrahedral Electrostatic Solid 10 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID127 3-D Tetrahedral Electrostatic Solid p-Element 10 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID128 3-D Brick Electrostatic Solid p-Element 20 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID147 3-D Brick Structural Solid p-Element 20 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SOLID148 3-D Tetrahedral Structural Solid p-Element 10 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN <> <> <> <> <> <> PP <>
SOLID164 Explicit 3-D Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>

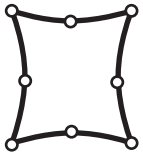
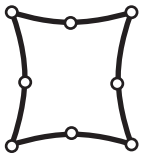
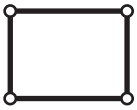
SOLID Elements	Product Codes
SOLID168 Explicit 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ 	<> <> <> <> <> <> <> <> <> <> DY <> <>
SOLID185 3-D 8-Node Structural Solid or Layered Solid 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SOLID186 3-D 20-Node Structural Solid or Layered Solid 20 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SOLID187 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SOLID191 3-D 20-Node Layered Structural Solid 20 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
SOLID226 3-D 20-Node Coupled-Field Solid 20 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT 	MP <> <> <> <> <> <> <> <> <> <> PP <>

SOLID Elements	Product Codes
SOLID227 3-D 10-Node Coupled-Field Solid 10 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT 	MP <> <> <> <> <> <> <> <> <> PP <>
SOLID231 3-D 20-Node Electric Solid 20 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>
SOLID232 3-D 10-Node Tetrahedral Electric Solid 10 nodes 3-D space DOF: VOLT 	MP <> <> <> <> <> <> <> EM <> <> PP <>

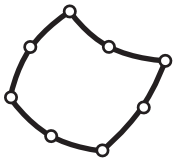
SOLSH Elements	Product Codes
SOLSH190 Structural Layered Solid Shell 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>

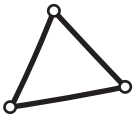
SOURC Elements	Product Codes
SOURC36 Magnetic Electric Current Source 3 nodes 3-D space DOF: None 	MP <> <> <> <> <> <> <> EM <> <> PP <>

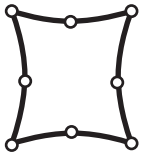
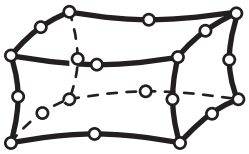
SURF Elements	Product Codes
SURF151 2-D Thermal Surface Effect 2 or 4 nodes 2-D space DOF: TEMP	MP ME ST PR PRN DS <> <> <> <> <> PP <>

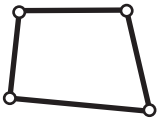
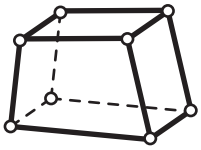
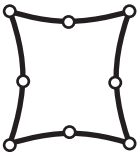
SURF Elements	Product Codes
	
SURF152 3-D Thermal Surface Effect 4 to 9 nodes 3-D space DOF: TEMP 	MP ME ST PR PRN DS <> <> <> <> <> PP VT
SURF153 2-D Structural Surface Effect 2 or 3 nodes 2-D space DOF: UX, UY 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SURF154 3-D Structural Surface Effect 4 to 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP VT
SURF156 3-D Structural Surface Line Load 3 to 4 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST PR PRN DS DSS <> <> <> <> PP <>
SURF251 2-D Radiosity Surface 2 nodes 2-D space 	MP ME <> PR PRN <> <> <> <> <> <> PP <>
SURF252 3-D Thermal Radiosity Surface 3 or 4 nodes, 3-D space 	MP ME <> PR PRN <> <> <> <> <> <> PP <>

TARGE Elements	Product Codes
TARGE169 Contact 2-D Target Segment	MP ME ST PR PRN DS DSS <> EM <> <> PP <>

TARGE Elements	Product Codes
3 nodes 2-D space DOF: UX, UY, ROTZ, TEMP 	
TARGE170 Contact 3-D Target Segment 8 nodes 3-D space DOF: UX, UY, UZ, TEMP 	MP ME ST PR PRN DS DSS <> EM <> <> PP <>

TRANS Elements	Product Codes
TRANS109 2-D Electromechanical Solid 3 nodes 2-D space DOF: UX, UY, VOLT 	MP <> <> <> <> <> <> <> <> <> <> PP <>
TRANS126 Electromechanical Transducer 2 nodes 3-D space DOF: UX-VOLT, UY-VOLT, UZ-VOLT 	MP <> <> <> <> <> <> <> <> <> <> PP <>

VISCO Elements	Product Codes
VISCO88 2-D 8-Node Viscoelastic Solid 8 nodes 2-D space DOF: UX, UY 	MP ME ST <> <> <> <> <> <> <> <> PP <>
VISCO89 3-D 20-Node Viscoelastic Solid 20 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
VISCO106	MP ME ST <> <> <> <> <> <> <> <> PP <>

VISCO Elements	Product Codes
2-D 4-Node Viscoplastic Solid 4 nodes 2-D space DOF: UX, UY, UZ 	
VISCO107 3-D 8-Node Viscoplastic Solid 8 nodes 3-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>
VISCO108 2-D 8-Node Viscoplastic Solid 8 nodes 2-D space DOF: UX, UY, UZ 	MP ME ST <> <> <> <> <> <> <> <> PP <>

3.3. GUI-Inaccessible Elements

These elements are available via the ET family of commands only and are inaccessible from within the ANSYS GUI:

SURF156	3-D Structural Surface Line Load Effect
MPC184-Point	Multipoint Constraint Element: Point-in-plane Joint
MPC184-Trans	Multipoint Constraint Element: Translational Joint
MPC184-Cylin	Multipoint Constraint Element: Cylindrical Joint
MPC184-Planar	Multipoint Constraint Element: Planar Joint
MPC184-Weld	Multipoint Constraint Element: Weld Joint
MPC184-Orient	Multipoint Constraint Element: Orient Joint
MPC184-Spherical	Multipoint Constraint Element: Spherical Joint
MPC184-General	Multipoint Constraint Element: General Joint
INTER202	2-D 4-Node Linear Interface
INTER203	2-D 6-Node Quadratic Interface
INTER204	3-D 16-Node Quadratic Interface
INTER205	3-D 8-Node Linear Interface
SURF251	2-D Radiosity Surface
SURF252	3-D Thermal Radiosity Surface

Part I, Elements Reference

Chapter 4: Element Library

This chapter describes each element, in numerical order. Descriptions common to several elements appear in separate sections of *Chapter 2: General Element Features* and are referenced where applicable. Read *Chapter 1: About This Manual* and *Chapter 2: General Element Features before* reading the element descriptions in *Part I, Elements Reference*.

The details of the element should also be reviewed in the *Theory Reference for ANSYS and ANSYS Workbench*, which explains how the element input items (such as the real constants, material properties, KEYOPT switches, etc.) are used to produce the element output.

LINK1

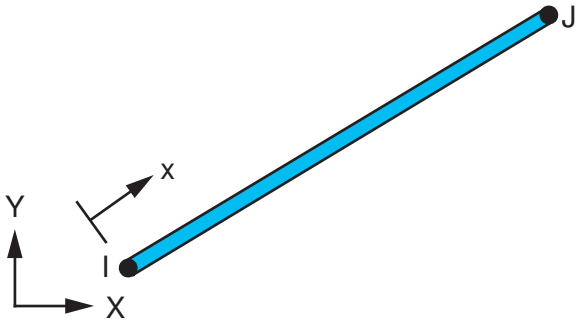
2-D Spar (or Truss)

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

LINK1 Element Description

LINK1 can be used in a variety of engineering applications. Depending upon the application, you can think of the element as a truss, a link, a spring, etc. The 2-D spar element is a uniaxial tension-compression element with two degrees of freedom at each node: translations in the nodal x and y directions. As in a pin-jointed structure, no bending of the element is considered. See LINK1 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See LINK8 for a description of a 3-D spar element.

Figure 1 LINK1 Geometry



LINK1 Input Data

Figure 1, "LINK1 Geometry" shows the geometry, node locations, and the coordinate system for this element. The element is defined by two nodes, the cross-sectional area, an initial strain, and the material properties. The element x-axis is oriented along the length of the element from node I toward node J. The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero-strain length.

Section 2.8: Node and Element Loads describes element loads. You can input temperatures and fluences as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I). Similar defaults occur for fluence except that zero is used instead of TUNIF. You can request a lumped mass matrix formulation, which may be useful for certain analyses such as wave propagation, with the **LUMPM** command.

LINK1 Input Summary summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

LINK1 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY

Real Constants

AREA - Cross-sectional area

ISTRN - Initial strain

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Fluences --
FL(I), FL(J)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPTS
None

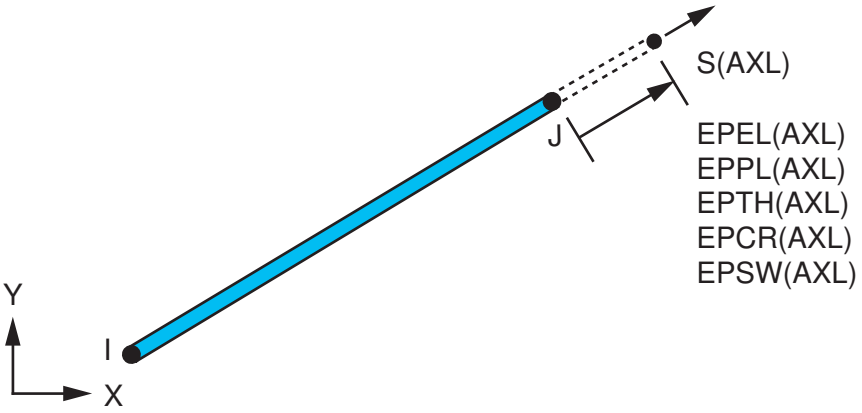
LINK1 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK1 Element Output Definitions"*.

Figure 2, "LINK1 Stress Output" illustrates several items. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 LINK1 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK1 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Element node numbers (I and J)	Y	Y
MAT	Material number for the element	Y	Y
VOLU:	Element volume	-	Y
XC, YC	Location where results are reported	Y	*2*
TEMP	Temperature at nodes I and J	Y	Y
FLUEN	Fluence at nodes I and J	Y	Y
MFORX	Member force in the element coordinate system X direction	Y	Y
SAXL	Axial stress in the element	Y	Y
EPELAXL	Axial elastic strain in the element	Y	Y
EPTHAXL	Axial thermal strain in the element	Y	Y
EPINAXL	Axial initial strain in the element	Y	Y
SEPL	Equivalent stress from the stress-strain curve	*1*	*1*
SRAT	Ratio of trial stress to the stress on yield surface	*1*	*1*
EPEQ	Equivalent plastic strain	*1*	*1*
HPRES	Hydrostatic pressure	*1*	*1*
EPPLAXL	Axial plastic strain	*1*	*1*
EPCRAXL	Axial creep strain	*1*	*1*
EPSWAXL	Axial swelling strain	*1*	*1*

1. Only if the element has a nonlinear material
2. Available only at centroid as a ***GET** item.

The Item and Sequence Number... table lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Output Quantity Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 2 LINK1 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SAXL	LS	1	-	-
EPELAXL	LEPEL	1	-	-
EPTHAXL	LEPTH	1	-	-
EPSWAXL	LEPTH	2	-	-
EPINAXL	LEPTH	3	-	-
EPPLAXL	LEPPL	1	-	-
EPCRAXL	LEPCR	1	-	-
SEPL	NLIN	1	-	-
SRAT	NLIN	2	-	-
HPRES	NLIN	3	-	-
EPEQ	NLIN	4	-	-
MFORX	SMISC	1	-	-
FLUEN	NMISC	-	1	2
TEMP	LBFE	-	1	2

LINK1 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends, of uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The spar must lie in an X-Y plane and must have an area greater than zero.
- The temperature is assumed to vary linearly along the length of the spar.
- The displacement shape function implies a uniform stress in the spar.
- The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.

LINK1 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads cannot be applied.
- The only special features allowed are stress stiffening and large deflections.

BEAM3

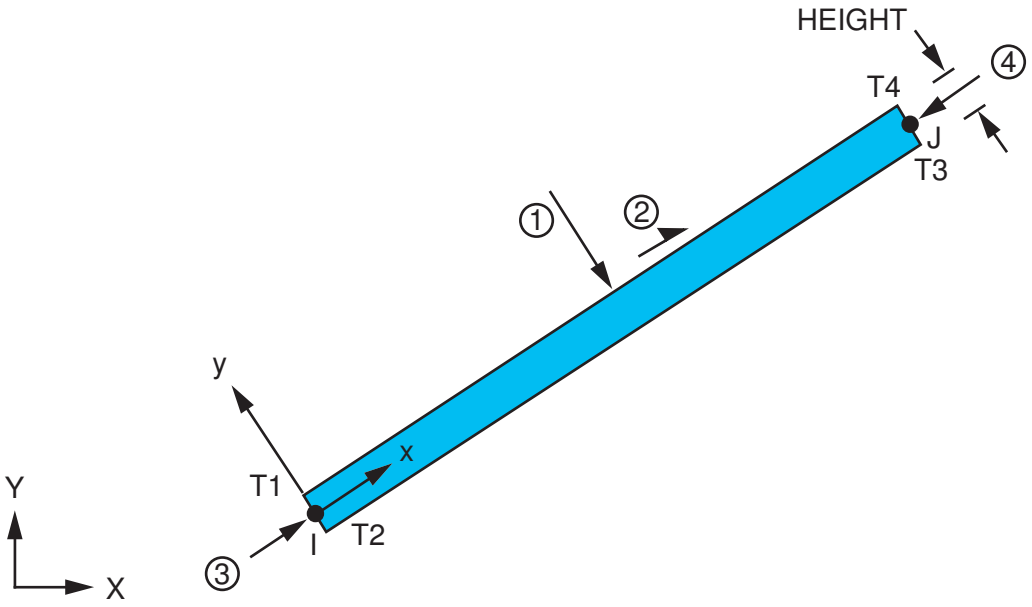
2-D Elastic Beam

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

BEAM3 Element Description

BEAM3 is a uniaxial element with tension, compression, and bending capabilities. The element has three degrees of freedom at each node: translations in the nodal x and y directions and rotation about the nodal z-axis. See BEAM3 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other 2-D beam elements are the plastic beam (BEAM23) and the tapered unsymmetric beam (BEAM54).

Figure 1 BEAM3 Geometry



BEAM3 Input Data

Figure 1, "BEAM3 Geometry" shows the geometry, node locations, and the coordinate system for this element. The element is defined by two nodes, the cross-sectional area, the area moment of inertia, the height, and the material properties. The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L (as defined by the I and J node locations), and the zero strain length. The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.

You can use the element in an axisymmetric analysis if hoop effects are negligible, such as for bolts, slotted cylinders, etc. The area and moment of inertia must be input on a full 360° basis for an axisymmetric analysis. The shear deflection constant (SHEARZ) is optional. You can use a zero value of SHEARZ to neglect shear deflection. See *Section 2.14: Shear Deflection* for details. The shear modulus (GXY) is used only with shear deflection. You can specify an added mass per unit length with the ADDMAS real constant.

Section 2.8: Node and Element Loads describes element loads. You can specify pressures as surface loads on the element faces, shown by the circled numbers in *Figure 1, "BEAM3 Geometry"*. Positive normal pressures act into the element. You specify lateral pressures as a force per unit length. End "pressures" are input as a force. KEYOPT(10) allows tapered lateral pressures to be offset from the nodes. You can specify temperatures as element body loads at the four "corner" locations shown in *Figure 1, "BEAM3 Geometry"*. The first corner temperature T1 defaults

to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(9) is used to request output at intermediate locations. It is based on equilibrium (free body of a portion of the element) considerations and is not valid if:

- stress stiffening is turned on [**SSTIF**,ON]
- more than one component of angular velocity is applied [**OMEGA**]
- *any* angular velocities or accelerations are applied with the **CGOMGA**, **DOMEGA**, or **DCGOMG** commands.

BEAM3 Input Summary summarizes the element input. *Section 2.1: Element Input* contains a general description of element input.

BEAM3 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, ROTZ

Real Constants

AREA - Cross-sectional area
IZZ - Area moment of inertia
HEIGHT - Total beam height
SHEARZ - Shear deflection constant
ISTRN - Initial strain
ADDMAS - Added mass per unit length

Note

SHEARZ goes with the IZZ. If SHEARZ = 0, there is no shear deflection in the element Y direction.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressure --

face 1 (I-J) (-Y normal direction)
face 2 (I-J) (+X tangential direction)
face 3 (I) (+X axial direction)
face 4 (J) (-X axial direction) (use a negative value for loading in the opposite direction)

Body Loads

Temperatures --
T1, T2, T3, T4

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(6)
Member force and moment output:

0 --
No printout of member forces and moments

1 --
Print out member forces and moments in the element coordinate system

KEYOPT(9)
Output at intermediate points between ends I and J:

N --
Output at N intermediate locations (N = 0, 1, 3, 5, 7, 9)

KEYOPT(10)
Load location, used in conjunction with the offset values input on the **SFBEAM** command):

0 --
Offset is in terms of length units

1 --
Offset is in terms of a length ratio (0.0 to 1.0)

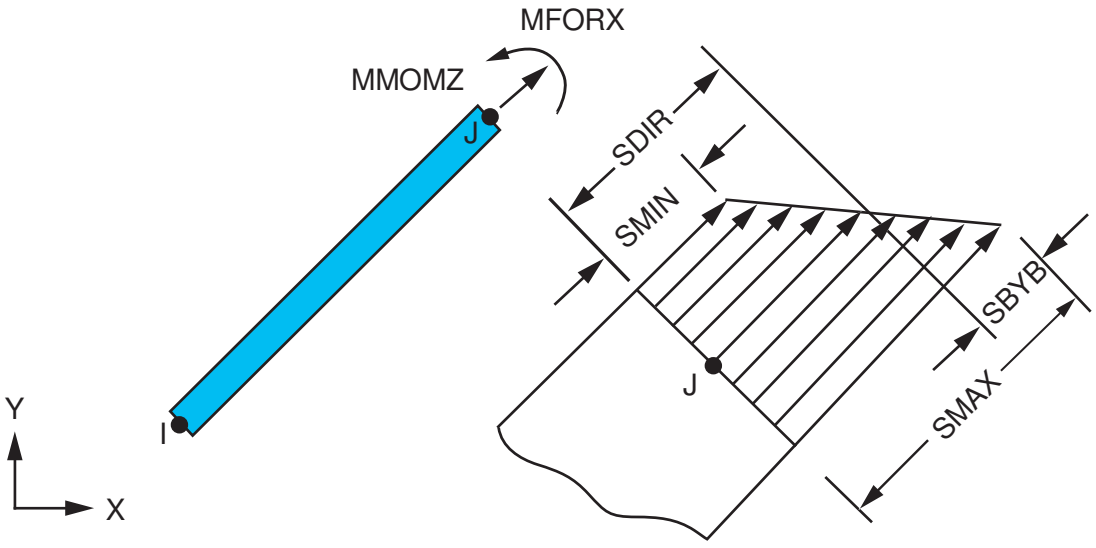
BEAM3 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "BEAM3 Element Output Definitions"*.

Figure 2, "BEAM3 Stress Output" illustrates several items. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 2 BEAM3 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 BEAM3 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Element nodes - I, J	Y	Y
MAT	Element material number	Y	Y
VOLU:	Element volume	N	Y
XC, YC	Location where results are reported	Y	*3*
TEMP	Temperatures T1, T2, T3, T4	Y	Y
PRES	Pressure P1 at nodes I,J; OFFST1 at I,J; P2 at I,J; OFFST2 at I, J; P3 at I; P4 at J	Y	Y
SDIR	Axial direct stress	*1*	*1*
SBYT	Bending stress on the element +Y side of the beam	*1*	*1*
SBYB	Bending stress on the element -Y side of the beam	*1*	*1*
SMAX	Maximum stress (direct stress + bending stress)	*1*	*1*
SMIN	Minimum stress (direct stress - bending stress)	*1*	*1*
EPELDIR	Axial elastic strain at the end	*1*	*1*
EPELBYT	Bending elastic strain on the element +Y side of the beam	*1*	*1*
EPELBYB	Bending elastic strain on the element -Y side of the beam	*1*	*1*
EPTHDIR	Axial thermal strain at the end	*1*	*1*
EPTHBYT	Bending thermal strain on the element +Y side of the beam	*1*	*1*
EPTHBYB	Bending thermal strain on the element -Y side of the beam	*1*	*1*
EPINAXL	Initial axial strain in the element	*1*	*1*
MFOR(X, Y)	Member forces in the element coordinate system X and Y direction	*2*	Y
MMOMZ	Member moment in the element coordinate system Z direction	*2*	Y

1. The item repeats for end I, intermediate locations (see KEYOPT(9)), and end J.
2. If KEYOPT(6) = 1.
3. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. *Table 2, "BEAM3 Item and Sequence Numbers (KEYOPT(9) = 0)"* through *Table 7, "BEAM3 Item and Sequence Numbers (KEYOPT(9) = 9)"* all use the following notation:

Name
output quantity as defined in the *Table 1, "BEAM3 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

IL*N*
sequence number for data at Intermediate Location *N*

Table 2 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 0)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDIR	LS	-	1	4
SBYT	LS	-	2	5
SBYB	LS	-	3	6
EPELDIR	LEPEL	-	1	4
EPELBYT	LEPEL	-	2	5
EPELBYB	LEPEL	-	3	6
EPTHDIR	LEPTH	-	1	4
EPTHBYT	LEPTH	-	2	5
EPTHBYB	LEPTH	-	3	6
EPINAXL	LEPTH	7	-	-
SMAX	NMISC	-	1	3
SMIN	NMISC	-	2	4
MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MMOMZ	SMISC	-	6	12
P1	SMISC	-	13	14
OFFST1	SMISC	-	15	16
P2	SMISC	-	17	18
OFFST2	SMISC	-	19	20
P3	SMISC	-	21	-
P4	SMISC	-	-	22

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 3 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 1)

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	ILI	J
SDIR	LS	-	1	4	7
SBYT	LS	-	2	5	8
SBYB	LS	-	3	6	9
EPELDIR	LEPEL	-	1	4	7

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	ILI	J
EPELBYT	LEPEL	-	2	5	8
EPELBYB	LEPEL	-	3	6	9
EPTHDIR	LEPTH	-	1	4	7
EPTHBYT	LEPTH	-	2	5	8
EPTHBYB	LEPTH	-	3	6	9
EPINAXL	LEPTH	10	-	-	-
SMAX	NMISC	-	1	3	5
SMIN	NMISC	-	2	4	6
MFORX	SMISC	-	1	7	13
MFORY	SMISC	-	2	8	14
MMOMZ	SMISC	-	6	12	18
P1	SMISC	-	19	-	20
OFFST1	SMISC	-	21	-	22
P2	SMISC	-	23	-	24
OFFST2	SMISC	-	25	-	26
P3	SMISC	-	27	-	-
P4	SMISC	-	-	-	28

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 4 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 3)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
SDIR	LS	-	1	4	7	10	13
SBYT	LS	-	2	5	8	11	14
SBYB	LS	-	3	6	9	12	15
EPELDIR	LEPEL	-	1	4	7	10	13
EPELBYT	LEPEL	-	2	5	8	11	14
EPELBYB	LEPEL	-	3	6	9	12	15
EPTHDIR	LEPTH	-	1	4	7	10	13
EPTHBYT	LEPTH	-	2	5	8	11	14
EPTHBYB	LEPTH	-	3	6	9	12	15
EPINAXL	LEPTH	16	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9
SMIN	NMISC	-	2	4	6	8	10
MFORX	SMISC	-	1	7	13	19	25
MFORY	SMISC	-	2	8	14	20	26
MMOMZ	SMISC	-	6	12	18	24	30
P1	SMISC	-	31	-	-	-	32

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
OFFST1	SMISC	-	33	-	-	-	34
P2	SMISC	-	35	-	-	-	36
OFFST2	SMISC	-	37	-	-	-	38
P3	SMISC	-	39	-	-	-	-
P4	SMISC	-	-	-	-	-	40

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 5 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 5)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	I	IL1	IL2	IL3	IL4	IL5	J
SDIR	LS	-	1	4	7	10	13	16	19
SBYT	LS	-	2	5	8	11	14	17	20
SBYB	LS	-	3	6	9	12	15	18	21
EPELDIR	LEPEL	-	1	4	7	10	13	16	19
EPELBYT	LEPEL	-	2	5	8	11	14	17	20
EPELBYB	LEPEL	-	3	6	9	12	15	18	21
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21
EPINAXL	LEPTH	22	-	-	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9	11	13
SMIN	NMISC	-	2	4	6	8	10	12	14
MFORX	SMISC	-	1	7	13	19	25	31	37
MFORY	SMISC	-	2	8	14	20	26	32	38
MMOMZ	SMISC	-	6	12	18	24	30	36	42
P1	SMISC	-	43	-	-	-	-	-	44
OFFST1	SMISC	-	45	-	-	-	-	-	46
P2	SMISC	-	47	-	-	-	-	-	48
OFFST2	SMISC	-	49	-	-	-	-	-	50
P3	SMISC	-	51	-	-	-	-	-	-
P4	SMISC	-	-	-	-	-	-	-	52

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 6 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 7)

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
SDIR	LS	-	1	4	7	10	13	16	19	22	25
SBYT	LS	-	2	5	8	11	14	17	20	23	26
SBYB	LS	-	3	6	9	12	15	18	21	24	27
EPELDIR	LEPEL	-	1	4	7	10	13	16	19	22	25
EPELBYT	LEPEL	-	2	5	8	11	14	17	20	23	26
EPELBYB	LEPEL	-	3	6	9	12	15	18	21	24	27
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19	22	25
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20	23	26
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21	24	27
EPINAXL	LEPTH	28	-	-	-	-	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54
P1	SMISC	-	55	-	-	-	-	-	-	-	56
OFFST1	SMISC	-	57	-	-	-	-	-	-	-	58
P2	SMISC	-	59	-	-	-	-	-	-	-	60
OFFST2	SMISC	-	61	-	-	-	-	-	-	-	62
P3	SMISC	-	63	-	-	-	-	-	-	-	-
P4	SMISC	-		-	-	-	-	-	-	-	64

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 7 BEAM3 Item and Sequence Numbers (KEYOPT(9) = 9)

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SDIR	LS	-	1	4	7	10	13	16	19	22	25	28	31
SBYT	LS	-	2	5	8	11	14	17	20	23	26	29	32
SBYB	LS	-	3	6	9	12	15	18	21	24	27	30	33
EPELDIR	LEPEL	-	1	4	7	10	13	16	19	22	25	28	31
EPELBYT	LEPEL	-	2	5	8	11	14	17	20	23	26	29	32
EPELBYB	LEPEL	-	3	6	9	12	15	18	21	24	27	30	33
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19	22	25	28	31
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20	23	26	29	32
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21	24	27	30	33
EPINAXL	LEPTH	34	-	-	-	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17	19	21
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18	20	22
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49	55	61
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50	56	62
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54	60	66
P1	SMISC	-	67	-	-	-	-	-	-	-	-	-	68
OFFST1	SMISC	-	69	-	-	-	-	-	-	-	-	-	70
P2	SMISC	-	71	-	-	-	-	-	-	-	-	-	72
OFFST2	SMISC	-	73	-	-	-	-	-	-	-	-	-	74
P3	SMISC	-	75	-	-	-	-	-	-	-	-	-	-
P4	SMISC	-	-	-	-	-	-	-	-	-	-	-	76

		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4

BEAM3 Assumptions and Restrictions

- The beam element must lie in an X-Y plane and must not have a zero length or area.
- The beam element can have any cross-sectional shape for which the moment of inertia can be computed. However, the stresses are determined as if the distance from the neutral axis to the extreme fiber is one-half of the height.
- The element height is used only in the bending and thermal stress calculations.
- The applied thermal gradient is assumed linear across the height and along the length.
- The moment of inertia may be zero if large deflections are not used.

BEAM3 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special features allowed are stress stiffening and large deflections.

BEAM4

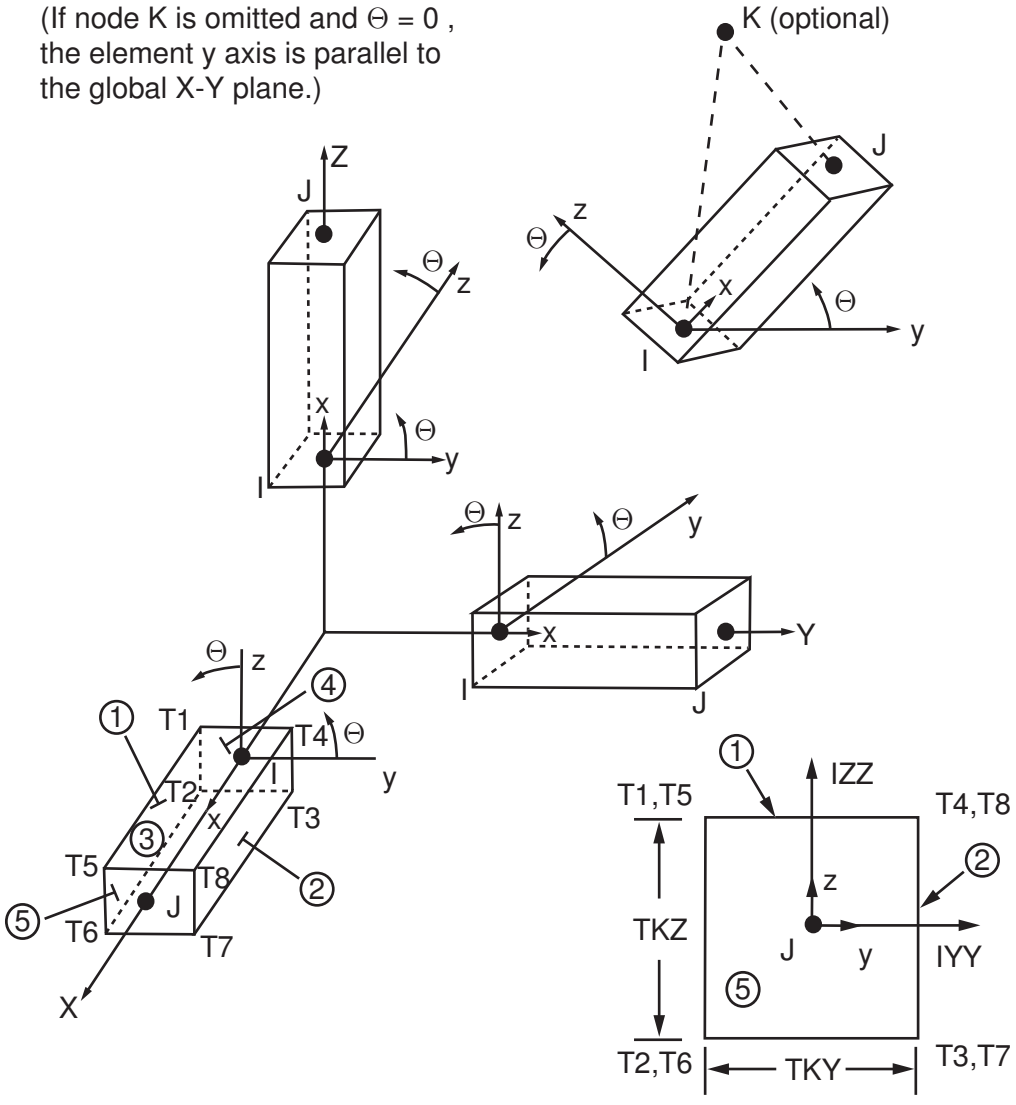
3-D Elastic Beam

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

BEAM4 Element Description

BEAM4 is a uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses. See BEAM4 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A tapered unsymmetrical elastic beam is described in BEAM44 and a 3-D plastic beam in BEAM24.

Figure 1 BEAM4 Geometry



BEAM4 Input Data

The geometry, node locations, and coordinate systems for this element are shown in *Figure 1, "BEAM4 Geometry"*. The element is defined by two or three nodes, the cross-sectional area, two area moments of inertia (IZZ and IYY), two thicknesses (TKY and TKZ), an angle of orientation (θ) about the element x-axis, the torsional moment of inertia (IXX), and the material properties. If IXX is not specified or is equal to 0.0, it is assumed equal to the polar moment of inertia (IYY + IZZ). IXX should be positive and is usually less than the polar moment of inertia. The element torsional stiffness decreases with decreasing values of IXX. An added mass per unit length may be input with the ADDMAS value.

The element x-axis is oriented from node I toward node J. For the two-node option, the default ($\theta = 0°$) orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in *Figure 1, "BEAM4 Geometry"*. For the case where the element is parallel to the global Z axis (or within a 0.01 percent slope of it), the element y axis is oriented parallel to the global Y axis (as shown). For user control of the element orientation about the element x-axis, use the θ angle (THETA) or the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z axes (as shown). If this element is used in a large deflection analysis, it should be noted that the location of the third node (K), or the angle (THETA), is used only to *initially* orient the element. (For information about orientation nodes and beam meshing, see Meshing Your Solid Model in the *Modeling and Meshing Guide*.)

The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero strain length. The shear deflection constants (SHEARZ and SHEARY) are used only if shear deflection is to be included. A zero value of SHEAR_ may be used to neglect shear deflection in a particular direction. See *Section 2.14: Shear Deflection* for details.

KEYOPT(2) is used to activate the consistent tangent stiffness matrix (i.e., a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) in large deflection analyses [**NLGEOM**,ON]. You can often obtain more rapid convergence in a geometrically nonlinear analysis, such as a nonlinear buckling or postbuckling analysis, by activating this option. However, you should not use this option if you are using the element to simulate a rigid link or a group of coupled nodes. The resulting abrupt changes in stiffness within the structure make the consistent tangent stiffness matrix unsuitable for such applications.

KEYOPT(7) is used to compute an unsymmetric gyroscopic damping matrix (often used for rotordynamic analyses). The rotational frequency is input with the SPIN real constant (radians/time, positive in the positive element x direction). The element must be symmetric with this option (e.g., IYY = IZZ and SHEARY = SHEARZ).

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM4 Geometry"*. Positive normal pressures act into the element. Lateral pressures are input as a force per unit length. End "pressures" are input as a force. KEYOPT(10) allows tapered lateral pressures to be offset from the nodes. Temperatures may be input as element body loads at the eight "corner" locations shown in *Figure 1, "BEAM4 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. If only T1 and T4 are input, T2 defaults to T1 and T3 defaults to T4. In both cases, T5 through T8 default to T1 through T4. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(9) is used to request output at intermediate locations. It is based on equilibrium (free body of a portion of the element) considerations and is not valid if:

- stress stiffening is turned on [**SSTIF**,ON]
- more than one component of angular velocity is applied [**OMEGA**]
- *any* angular velocities or accelerations are applied with the **CGOMGA**, **DOMEGA**, or **DCGOMG** commands.

A summary of the element input is given in *BEAM4 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

BEAM4 Input Summary

Nodes
I, J, K (K orientation node is optional)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

AREA, IZZ, IYY, TKZ, TKY, THETA
ISTRN, IXX, SHEARZ, SHEARY, SPIN, ADDMAS
See *Table 1, "BEAM4 Real Constants"* for a description of the real constants.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J) (-Z normal direction)
face 2 (I-J) (-Y normal direction)
face 3 (I-J) (+X tangential direction)
face 4 (I) (+X axial direction)
face 5 (J) (-X axial direction)
(use negative value for opposite loading)

Body Loads

Temperatures --
T1, T2, T3, T4, T5, T6, T7, T8

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(2)
Stress stiffening option:

0 --
Use only the main tangent stiffness matrix when **NLGEOM** is ON. (Stress stiffening effects used in linear buckling or other linear prestressed analyses must be activated separately with **PSTRES**,ON.)

1 --
Use the consistent tangent stiffness matrix (i.e., a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) when **NLGEOM** is ON. (**SSTIF**,ON will be ignored for this element when KEYOPT(2) = 1 is activated.) Note that if **SOLCONTROL** is ON and **NLGEOM** is ON, KEYOPT(2) is automatically set to 1; i.e., the consistent tangent will be used.

2 --
Turn off consistent tangent stiffness matrix (i.e., a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) when **SOLCONTROL** is ON. Sometimes it is necessary to turn off the consistent tangent stiffness matrix if the element is used to simulate rigid bodies by using a very

large real constant number . KEYOPT(2) = 2 is the same as KEYOPT(2) = 0, however, KEYOPT(2) = 0 is controlled by **SOLCONTROL**, ON or OFF, while KEYOPT(2) = 2 is independent of **SOLCONTROL**.

KEYOPT(6)

Member force and moment output:

0 --

No printout of member forces or moments

1 --

Print out member forces and moments in the element coordinate system

KEYOPT(7)

Gyroscopic damping matrix:

0 --

No gyroscopic damping matrix

1 --

Compute gyroscopic damping matrix. Real constant SPIN must be greater than zero. IYY must equal IZZ.

KEYOPT(9)

Output at intermediate points between ends I and J:

N --

Output at N intermediate locations (N = 0, 1, 3, 5, 7, 9)

KEYOPT(10)

Load location, used in conjunction with the offset values input on the **SFBEAM** command):

0 --

Offset is in terms of length units

1 --

Offset is in terms of a length ratio (0.0 to 1.0)

Table 1 BEAM4 Real Constants

No.	Name	Description
1	AREA	Cross-sectional area
2	IZZ	Area moment of inertia
3	IYY	Area moment of inertia
4	TKZ	Thickness along Z axis
5	TKY	Thickness along Y axis
6	THETA	Orientation about X axis
7	ISTRN	Initial strain
8	IXX	Torsional moment of inertia
9	SHEARZ	Shear deflection constant Z [*1*]
10	SHEARY	Shear deflection constant Y [*2*]
11	SPIN	Rotational frequency (required if KEYOPT(7) = 1)
12	ADDMAS	Added mass/unit length

1. SHEARZ goes with IZZ; if SHEARZ = 0, there is no shear deflection in the element Y direction.
2. SHEARY goes with IYY; if SHEARY = 0, there is no shear deflection in the element Z direction.

BEAM4 Output Data

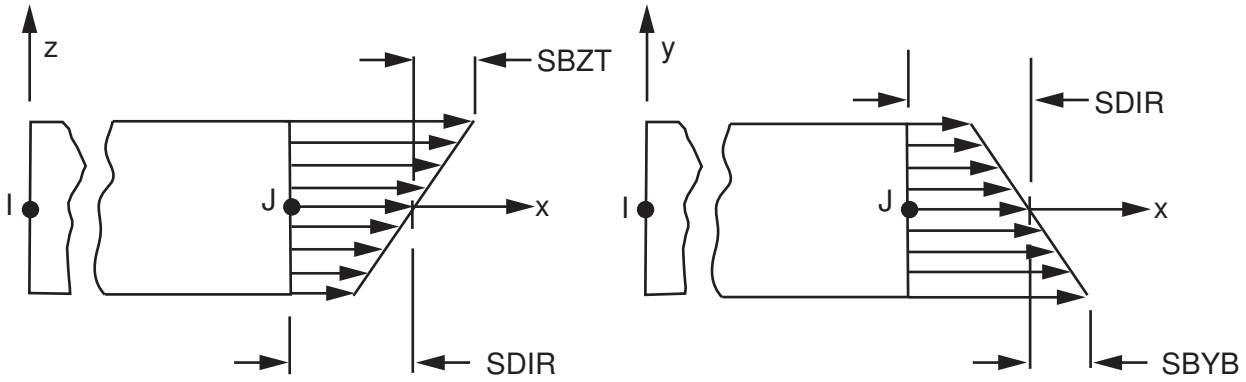
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "BEAM4 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "BEAM4 Stress Output"*.

The maximum stress is computed as the direct stress plus the absolute values of both bending stresses. The minimum stress is the direct stress minus the absolute value of both bending stresses. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 BEAM4 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 BEAM4 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element node number (I and J)	Y	Y
MAT	Material number for the element	Y	Y
VOLU:	Element volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	Temperatures at integration points T1, T2, T3, T4, T5, T6, T7, T8	Y	Y
PRES	Pressure P1 at nodes I, J; OFFST1 at I, J; P2 at I, J; OFFST2 at I, J; P3 at I, J; OFFST3 at I, J; P4 at I; P5 at J	Y	Y
SDIR	Axial direct stress	*1*	*1*
SBYT	Bending stress on the element +Y side of the beam	*1*	*1*
SBYB	Bending stress on the element -Y side of the beam	*1*	*1*
SBZT	Bending stress on the element +Z side of the beam	*1*	*1*

Name	Definition	O	R
SBZB	Bending stress on the element -Z side of the beam	*1*	*1*
SMAX	Maximum stress (direct stress + bending stress)	*1*	*1*
SMIN	Minimum stress (direct stress - bending stress)	*1*	*1*
EPELDIR	Axial elastic strain at the end	*1*	*1*
EPELBYT	Bending elastic strain on the element +Y side of the beam	*1*	*1*
EPELBYB	Bending elastic strain on the element -Y side of the beam	*1*	*1*
EPELBZT	Bending elastic strain on the element +Z side of the beam	*1*	*1*
EPELBZB	Bending elastic strain on the element -Z side of the beam	*1*	*1*
EPTHDIR	Axial thermal strain at the end	*1*	*1*
EPTHBYT	Bending thermal strain on the element +Y side of the beam	*1*	*1*
EPTHBYB	Bending thermal strain on the element -Y side of the beam	*1*	*1*
EPTHBZT	Bending thermal strain on the element +Z side of the beam	*1*	*1*
EPTHBZB	Bending thermal strain on the element -Z side of the beam	*1*	*1*
EPINAXL	Initial axial strain in the element	*1*	*1*
MFOR(X, Y, Z)	Member forces in the element coordinate system X, Y, Z directions	*2*	Y
MMOM(X, Y, Z)	Member moments in the element coordinate system X, Y, Z directions	*2*	Y

1. The item repeats for end I, intermediate locations (see KEYOPT(9)), and end J.
2. If KEYOPT(6) = 1.
3. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "BEAM4 Item and Sequence Numbers (KEYOPT(9) = 0)"* through *Table 8, "BEAM4 Item and Sequence Numbers (KEYOPT(9) = 9)"*:

Name
output quantity as defined in the *Table 2, "BEAM4 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

IL*N*
sequence number for data at Intermediate Location *N*

Table 3 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 0)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDIR	LS	-	1	6
SBYT	LS	-	2	7
SBYB	LS	-	3	8
SBZT	LS	-	4	9
SBZB	LS	-	5	10
EPELDIR	LEPEL	-	1	6
EPELBYT	LEPEL	-	2	7
EPELBYB	LEPEL	-	3	8
EPELBZT	LEPEL	-	4	9
EPELBZB	LEPEL	-	5	10
SMAX	NMISC	-	1	3
SMIN	NMISC	-	2	4
EPTHDIR	LEPTH	-	1	6
EPTHBYT	LEPTH	-	2	7
EPTHBYB	LEPTH	-	3	8
EPTHBZT	LEPTH	-	4	9
EPTHBZB	LEPTH	-	5	10
EPINAXL	LEPTH	11	-	-
MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MFORZ	SMISC	-	3	9
MMOMX	SMISC	-	4	10
MMOMY	SMISC	-	5	11
MMOMZ	SMISC	-	6	12
P1	SMISC	-	13	14
OFFST1	SMISC	-	15	16
P2	SMISC	-	17	18
OFFST2	SMISC	-	19	20
P3	SMISC	-	21	22
OFFST3	SMISC	-	23	24
P4	SMISC	-	25	-
P5	SMISC	-	-	26

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 4 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 1)

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	IL1	J
SDIR	LS	-	1	6	11
SBYT	LS	-	2	7	12
SBYB	LS	-	3	8	13
SBZT	LS	-	4	9	14
SBZB	LS	-	5	10	15
EPELDIR	LEPEL	-	1	6	11
EPELBYT	LEPEL	-	2	7	12
EPELBYB	LEPEL	-	3	8	13
EPELBZT	LEPEL	-	4	9	14
EPELBZB	LEPEL	-	5	10	15
SMAX	NMISC	-	1	3	5
SMIN	NMISC	-	2	4	6
EPTHDIR	LEPTH	-	1	6	11
EPTHBYT	LEPTH	-	2	7	12
EPTHBYB	LEPTH	-	3	8	13
EPTHBZT	LEPTH	-	4	9	14
EPTHBZB	LEPTH	-	5	10	15
EPINAXL	LEPTH	16	-	-	-
MFORX	SMISC	-	1	7	13
MFORY	SMISC	-	2	8	14
MFORZ	SMISC	-	3	9	15
MMOMX	SMISC	-	4	10	16
MMOMY	SMISC	-	5	11	17
MMOMZ	SMISC	-	6	12	18
P1	SMISC	-	19	-	20
OFFST1	SMISC	-	21	-	22
P2	SMISC	-	23	-	24
OFFST2	SMISC	-	25	-	26
P3	SMISC	-	27	-	28
OFFST3	SMISC	-	29	-	30
P4	SMISC	-	31	-	-
P5	SMISC	-	-	-	32

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 5 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 3)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
SDIR	LS	-	1	6	11	16	21
SBYT	LS	-	2	7	12	17	22
SBYB	LS	-	3	8	13	18	23
SBZT	LS	-	4	9	14	19	24
SBZB	LS	-	5	10	15	20	25
EPELDIR	LEPEL	-	1	6	11	16	21
EPELBYT	LEPEL	-	2	7	12	17	22
EPELBYB	LEPEL	-	3	8	13	18	23
EPELBZT	LEPEL	-	4	9	14	19	24
EPELBZB	LEPEL	-	5	10	15	20	25
SMAX	NMISC	-	1	3	5	7	9
SMIN	NMISC	-	2	4	6	8	10
EPTHDIR	LEPTH	-	1	6	11	16	21
EPTHBYT	LEPTH	-	2	7	12	17	22
EPTHBYB	LEPTH	-	3	8	13	18	23
EPTHBZT	LEPTH	-	4	9	14	19	24
EPTHBZB	LEPTH	-	5	10	15	20	25
EPINAXL	LEPTH	26	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25
MFORY	SMISC	-	2	8	14	20	26
MFORZ	SMISC	-	3	9	15	21	27
MMOMX	SMISC	-	4	10	16	22	28
MMOMY	SMISC	-	5	11	17	23	29
MMOMZ	SMISC	-	6	12	18	24	30
P1	SMISC	-	31	-	-	-	32
OFFST1	SMISC	-	33	-	-	-	34
P2	SMISC	-	35	-	-	-	36
OFFST2	SMISC	-	37	-	-	-	38
P3	SMISC	-	39	-	-	-	40
OFFST3	SMISC	-	41	-	-	-	42
P4	SMISC	-	43	-	-	-	
P5	SMISC	-	-	-	-	-	44

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 6 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 5)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	I	IL1	IL2	IL3	IL4	IL5	J
SDIR	LS	-	1	6	11	16	21	26	31
SBYT	LS	-	2	7	12	17	22	27	32
SBYB	LS	-	3	8	13	18	23	28	33
SBZT	LS	-	4	9	14	19	24	29	34
SBZB	LS	-	5	10	15	20	25	30	35
EPELDIR	LEPEL	-	1	6	11	16	21	26	31
EPELBYT	LEPEL	-	2	7	12	17	22	27	32
EPELBYB	LEPEL	-	3	8	13	18	23	28	33
EPELBZT	LEPEL	-	4	9	14	19	24	29	34
EPELBZB	LEPEL	-	5	10	15	20	25	30	35
SMAX	NMISC	-	1	3	5	7	9	11	13
SMIN	NMISC	-	2	4	6	8	10	12	14
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35
EPINAXL	LEPTH	36	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37
MFORY	SMISC	-	2	8	14	20	26	32	38
MFORZ	SMISC	-	3	9	15	21	27	33	39
MMOMX	SMISC	-	4	10	16	22	28	34	40
MMOMY	SMISC	-	5	11	17	23	29	35	41
MMOMZ	SMISC	-	6	12	18	24	30	36	42
P1	SMISC	-	43	-	-	-	-	-	44
OFFST1	SMISC	-	45	-	-	-	-	-	46
P2	SMISC	-	47	-	-	-	-	-	48
OFFST2	SMISC	-	49	-	-	-	-	-	50
P3	SMISC	-	51	-	-	-	-	-	52
OFFST3	SMISC	-	53	-	-	-	-	-	54
P4	SMISC	-	55	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	56

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 7 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 7)

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
SDIR	LS	-	1	6	11	16	21	26	31	36	41
SBYT	LS	-	2	7	12	17	22	27	32	37	42
SBYB	LS	-	3	8	13	18	23	28	33	38	43
SBZT	LS	-	4	9	14	19	24	29	34	39	44
SBZB	LS	-	5	10	15	20	25	30	35	40	45
EPELDIR	LEPEL	-	1	6	11	16	21	26	31	36	41
EPELBYT	LEPEL	-	2	7	12	17	22	27	32	37	42
EPELBYB	LEPEL	-	3	8	13	18	23	28	33	38	43
EPELBZT	LEPEL	-	4	9	14	19	24	29	34	39	44
EPELBZB	LEPEL	-	5	10	15	20	25	30	35	40	45
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31	36	41
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32	37	42
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33	38	43
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34	39	44
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35	40	45
EPINAXL	LEPTH	46	-	-	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50
MFORZ	SMISC	-	3	9	15	21	27	33	39	45	51
MMOMX	SMISC	-	4	10	16	22	28	34	40	46	52
MMOMY	SMISC	-	5	11	17	23	29	35	41	47	53
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54
P1	SMISC	-	55	-	-	-	-	-	-	-	56
OFFST1	SMISC	-	57	-	-	-	-	-	-	-	58
P2	SMISC	-	59	-	-	-	-	-	-	-	60
OFFST2	SMISC	-	61	-	-	-	-	-	-	-	62
P3	SMISC	-	63	-	-	-	-	-	-	-	64
OFFST3	SMISC	-	65	-	-	-	-	-	-	-	66
P4	SMISC	-	67	-	-	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	-	-	68

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 8 BEAM4 Item and Sequence Numbers (KEYOPT(9) = 9)

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SDIR	LS	-	1	6	11	16	21	26	31	36	41	46	51
SBYT	LS	-	2	7	12	17	22	27	32	37	42	47	52
SBYB	LS	-	3	8	13	18	23	28	33	38	43	48	53
SBZT	LS	-	4	9	14	19	24	29	34	39	44	49	54
SBZB	LS	-	5	10	15	20	25	30	35	40	45	50	55
EPELDIR	LEPEL	-	1	6	11	16	21	26	31	36	41	46	51
EPELBYT	LEPEL	-	2	7	12	17	22	27	32	37	42	47	52
EPELBYB	LEPEL	-	3	8	13	18	23	28	33	38	43	48	53
EPELBZT	LEPEL	-	4	9	14	19	24	29	34	39	44	49	54
EPELBZB	LEPEL	-	5	10	15	20	25	30	35	40	45	50	55
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17	19	21
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18	20	22
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31	36	41	46	51
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32	37	42	47	52
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33	38	43	48	53
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34	39	44	49	54
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35	40	45	50	55
EPINAXL	LEPTH	56	-	-	-	-	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49	55	61
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50	56	62
MFORZ	SMISC	-	3	9	15	21	27	33	39	45	51	57	63
MMOMX	SMISC	-	4	10	16	22	28	34	40	46	52	58	64
MMOMY	SMISC	-	5	11	17	23	29	35	41	47	53	59	65
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54	60	66
P1	SMISC	-	67	-	-	-	-	-	-	-	-	-	68
OFFST1	SMISC	-	69	-	-	-	-	-	-	-	-	-	70
P2	SMISC	-	71	-	-	-	-	-	-	-	-	-	72
OFFST2	SMISC	-	73	-	-	-	-	-	-	-	-	-	74
P3	SMISC	-	75	-	-	-	-	-	-	-	-	-	76
OFFST3	SMISC	-	77	-	-	-	-	-	-	-	-	-	78
P4	SMISC	-	79	-	-	-	-	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	-	-	-	-	80

		Pseudo Node							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

BEAM4 Assumptions and Restrictions

- The beam must not have a zero length or area. The moments of inertia, however, may be zero if large deflections are not used.

- The beam can have any cross-sectional shape for which the moments of inertia can be computed. The stresses, however, will be determined as if the distance between the neutral axis and the extreme fiber is one-half of the corresponding thickness.
- The element thicknesses are used only in the bending and thermal stress calculations.
- The applied thermal gradients are assumed to be linear across the thickness in both directions and along the length of the element.
- If you use the consistent tangent stiffness matrix (KEYOPT(2) = 1), take care to use realistic (that is, "to scale") element real constants. This precaution is necessary because the consistent stress-stiffening matrix is based on the calculated stresses in the element. If you use artificially large or small cross-sectional properties, the calculated stresses will become inaccurate, and the stress-stiffening matrix will suffer corresponding inaccuracies. (Certain components of the stress-stiffening matrix could even overshoot to infinity.) Similar difficulties could arise if unrealistic real constants are used in a linear prestressed or linear buckling analysis [**PSTRES**,ON].
- Eigenvalues calculated in a gyroscopic modal analysis can be very sensitive to changes in the initial shift value, leading to potential error in either the real or imaginary (or both) parts of the eigenvalues.

BEAM4 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The SPIN real constant (R11) is not available. Input R11 as a blank.
- The DAMP material property is not allowed.
- KEYOPT(2) can only be set to 0 (default).
- KEYOPT(7) can only be set to 0 (default).
- The only special features allowed are stress stiffening and large deflections.

SOLID5

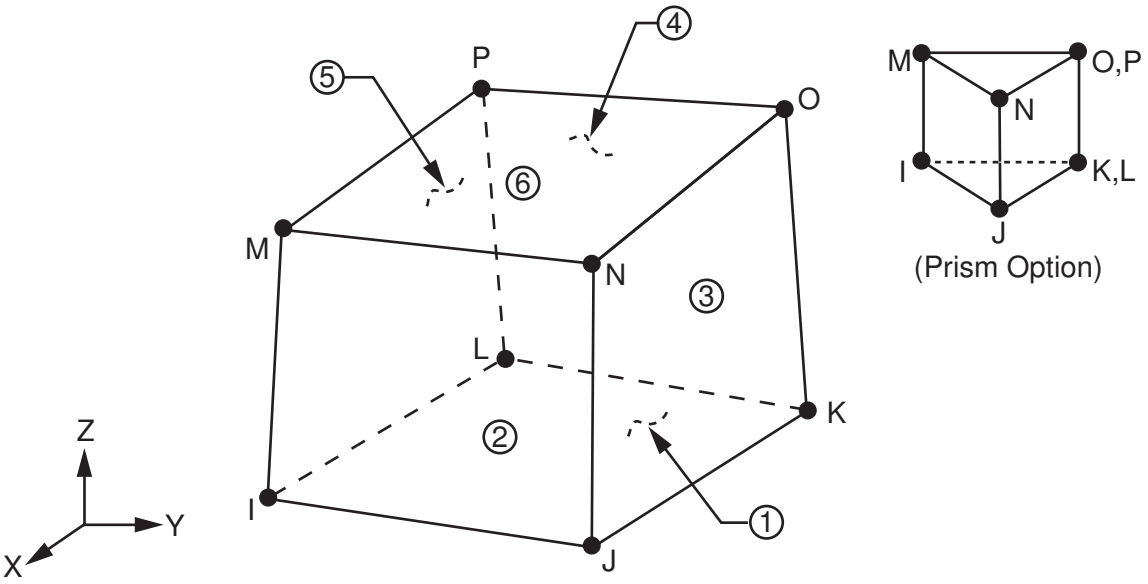
3-D Coupled-Field Solid

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID5 Element Description

SOLID5 has a 3-D magnetic, thermal, electric, piezoelectric, and structural field capability with limited coupling between the fields. The element has eight nodes with up to six degrees of freedom at each node. Scalar potential formulations (reduced RSP, difference DSP, or general GSP) are available for modeling magnetostatic fields in a static analysis. When used in structural and piezoelectric analyses, SOLID5 has large deflection and stress stiffening capabilities. See SOLID5 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Coupled field elements with similar field capabilities are PLANE13, SOLID62, and SOLID98.

Figure 1 SOLID5 Geometry



SOLID5 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID5 Geometry"*. The element is defined by eight nodes and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4 \pi \times 10^{-7}$ Henries/meter. In addition to MUZERO, orthotropic relative permeability is specified through the MURX, MURY, and MURZ material property labels.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Nonlinear magnetic, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (UX, UY, UZ, TEMP, VOLT, MAG) and *VALUE* corresponds to the value (displacements, temperature, voltage, scalar magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (FX, FY, FZ, HEAT, AMPS, FLUX) and *VALUE* corresponds to the value (force, heat flow, current or charge, magnetic flux).

Element loads are described in *Section 2.8: Node and Element Loads*. Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID5 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

The body loads, temperature, heat generation rate and magnetic virtual displacement may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 0,1 or 8), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal values of temperature and heat generation rate default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) is applied in subsequent iterations as heat generation rate.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

Current for the scalar magnetic potential options is defined with the SOURC36 element the command macro RACE, or through electromagnetic coupling. The various types of scalar magnetic potential solution options are defined with the **MAGOPT** command.

A summary of the element input is given in *SOLID5 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID5 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, TEMP, VOLT, MAG if KEYOPT (1) = 0
TEMP, VOLT, MAG if KEYOPT (1) = 1
UX, UY, UZ if KEYOPT (1) = 2
UX, UY, UZ, VOLT if KEYOPT(1) = 3
TEMP if KEYOPT (1) = 8
VOLT if KEYOPT (1) = 9
MAG if KEYOPT (1) = 10

Real Constants

None

Material Properties

EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP, KXX, KYY, KZZ, C,
ENTH, MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ,
MGXX, MGYY, MGZZ, PERX, PERY, PERZ, plus BH, ANEL, and PIEZ data tables (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Magnetic Virtual Displacements --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

Electric Field --
EFX, EFY, EFZ. See *SOLID5 Assumptions and Restrictions*.

Special Features

Requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
Large deflection
Stress stiffening
Birth and death
Adaptive descent

KEYOPT(1)

Element degrees of freedom:

0 --
UX, UY, UZ, TEMP, VOLT, MAG

1 --
TEMP, VOLT, MAG

2 --
UX, UY, UZ

3 --
UX, UY, UZ, VOLT

8 --
TEMP

9 --
VOLT

10 --
MAG

KEYOPT(3)
Extra shapes:

0 --
Include extra shapes

1 --
Do not include extra shapes

KEYOPT(5)
Extra element output:

0 --
Basic element printout

2 --
Nodal stress or magnetic field printout

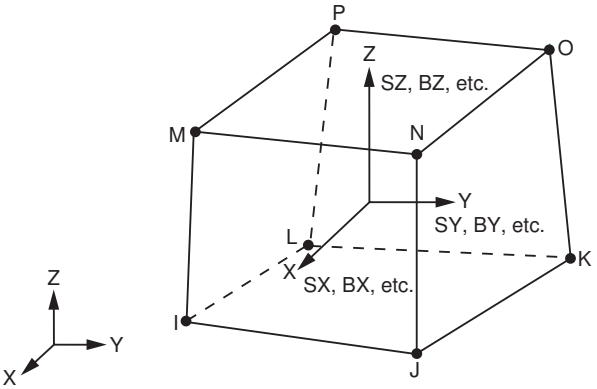
SOLID5 Output Data

The solution output associated with the element is in two forms

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID5 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "SOLID5 Element Output"*. The element stress directions are parallel to the element coordinate system. The reaction forces, heat flow, current, and magnetic flux at the nodes can be printed with the **OUTPR** command. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID5 Element Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID5 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Element nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Element material number	Y	Y
VOLU:	Element volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Input Temperatures: T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
HGEN	Input Heat Generations: HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Component stresses	*1*	*1*
S:1, 2, 3	Principal stresses	*1*	*1*
S:INT	Stress intensity	*1*	*1*
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal elastic strains	*1*	-
EPEL:EQV	Equivalent elastic strains [*4*]	*1*	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strains [*4*]	*1*	*1*
LOC	Output location (X, Y, Z)	*1*	*1*
MUX, MUY, MUZ	Magnetic permeability	*1*	*1*
H: X, Y, Z	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y, Z	Magnetic flux density components	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
FJB	Lorentz magnetic force components (X, Y, Z)	*1*	-
FMX	Maxwell magnetic force components (X, Y, Z)	*1*	-
FVW	Virtual work force components (X, Y, Z)	*1*	*1*
FMAG:X, Y, Z	Combined (FJB or FMX) force components	-	*1*
EF:X, Y, Z	Electric field components (X, Y, Z)	*1*	*1*
EF:SUM	Vector magnitude of EF	*1*	*1*
JS:X, Y, Z	Source current density components	*1*	*1*
JSSUM	Vector magnitude of JS	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
D:X, Y, Z	Electric flux density components	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
UE, UD, UM	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	*1*	*1*
TG:X, Y, Z	Thermal gradient components	*1*	*1*
TG:SUM	Vector magnitude of TG	*1*	*1*
TF:X, Y, Z	Thermal flux components	*1*	*1*

Name	Definition	O	R
TF:SUM	Vector magnitude of TF (heat flow rate/unit cross-section area)	*1*	*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
NODES	Face nodes	*2*	-
HFILM	Film coefficient at each node of face	*2*	-
TBULK	Bulk temperature at each node of face	*2*	-
TAVG	Average face temperature	*2*	*2*
HEAT RATE	Heat flow rate across face by convection	*2*	*2*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*2*	-
HFLUX	Heat flux at each node of face	*2*	-
HFAVG	Average film coefficient of the face	*2*	*2*
TBAVG	Average face bulk temperature	-	*2*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*2*

1. Element solution at the centroid printed out only if calculated (based on input data).
2. Nodal stress or magnetic field solution (only if KEYOPT(5) = 2). The solution results are repeated at each node and only if a surface load is input.
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2, "SOLID5 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. The following notation is used in *Table 2, "SOLID5 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SOLID5 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,...,P
: sequence number for data at nodes I,J,...,P

FC*n*
: sequence number for solution items for element Face *n*

Table 2 SOLID5 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	4	3	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-
P4	SMISC	-	-	-	13	14	-	-	16	15

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P5	SMISC	-	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	-	21	22	23	24
MUX	NMISC	1	-	-	-	-	-	-	-	-
MUY	NMISC	2	-	-	-	-	-	-	-	-
MUZ	NMISC	3	-	-	-	-	-	-	-	-
FVWX	NMISC	4	-	-	-	-	-	-	-	-
FVWY	NMISC	5	-	-	-	-	-	-	-	-
FVWZ	NMISC	6	-	-	-	-	-	-	-	-
FVWSUM	NMISC	7	-	-	-	-	-	-	-	-
UE	NMISC	16	-	-	-	-	-	-	-	-
UD	NMISC	17	-	-	-	-	-	-	-	-
UM	NMISC	18	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	19	25	31	37	43	49
HFAVG	NMISC	20	26	32	38	44	50
TAVG	NMISC	21	27	33	39	45	51
TBAVG	NMISC	22	28	34	40	46	52
HEAT RATE	NMISC	23	29	35	41	47	53
HFLXAVG	NMISC	24	30	36	42	48	54

SOLID5 Assumptions and Restrictions

- When using SOLID5 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfils boundary conditions for the total field.
- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "SOLID5 Geometry"* or may have the planes IJKL and MNOP interchanged.
- A prism shaped element may be formed by defining duplicate node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The difference scalar magnetic potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar and general scalar potential options do not have this restriction.
- At a free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint), the normal component of magnetic flux density (B) is assumed to be zero.
- Temperatures and heat generation rates, if internally calculated, include any user defined heat generation rates.
- The thermal, electrical, magnetic, and structural terms are coupled through an iterative procedure.
- Large deflection capabilities available for KEYOPT(1) = 2 and 3 are not available for KEYOPT(1) = 0.
- Stress stiffening is available for KEYOPT(1) = 0, 2, and 3.

- Do not constrain all VOLT DOFs to the same value in a piezoelectric analysis (KEYOPT(1) = 0 or 3). Perform a pure structural analysis instead (KEYOPT(1) = 2).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.
- If you used the MAG degree of freedom, you cannot restart a job in ANSYS Mechanical using `Jobname.DB` and `Jobname.ESAV` files that were created by ANSYS Multiphysics.

SOLID5 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic capability.
- The MAG degree of freedom is not active.
- KEYOPT(1) cannot be set to 10. If KEYOPT(1) = 0 (default) or 1, the MAG degree of freedom is inactive.
- The magnetic material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags and magnetic virtual displacements body loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are MAG and VOLT.
- KEYOPT(1) settings of 0, 1, 2, 3 and 8 are invalid.
- The only allowable material properties are the magnetic and electric properties (MUZERO through PERZ, plus the BH data table).
- The only applicable surface loads are Maxwell force flags. The only applicable body loads are temperatures (for material property evaluation only) and magnetic virtual displacements.
- The element does not have stress stiffening or birth and death features.
- KEYOPT(3) is not applicable.

COMBIN7

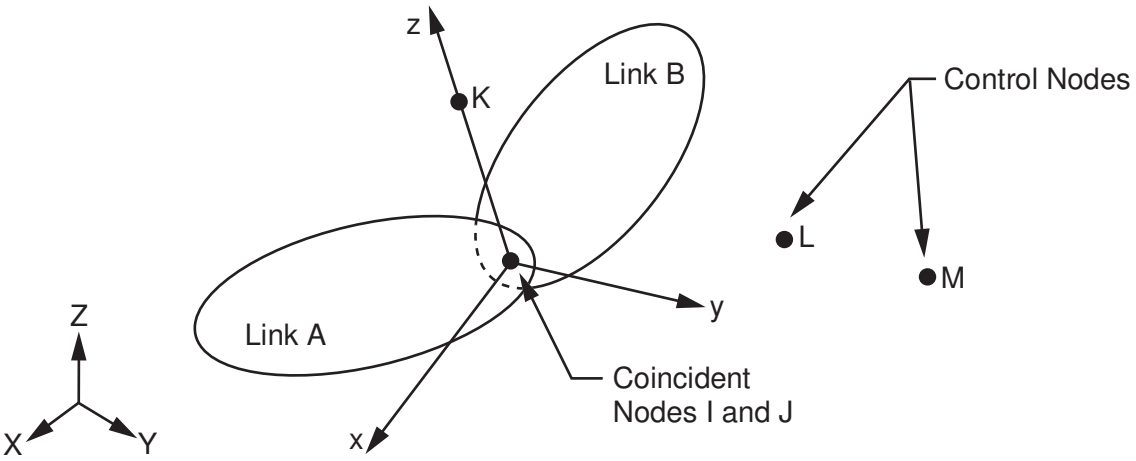
Revolute Joint

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

COMBIN7 Element Description

COMBIN7 is a 3-D pin (or revolute) joint which may be used to connect two or more parts of a model at a common point. Capabilities of this element include joint flexibility (or stiffness), friction, damping, and certain control features. An important feature of this element is a large deflection capability in which a local coordinate system is fixed to and moves with the joint. This element is intended for use in kinetostatic and kinetodynamic analyses. See COMBIN7 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A unidirectional control element having less capability is described in COMBIN37. Similar elements (without remote control capability) are COMBIN14, MASS21, COMBIN39, and COMBIN40.

Figure 1 COMBIN7 Geometry



COMBIN7 Input Data

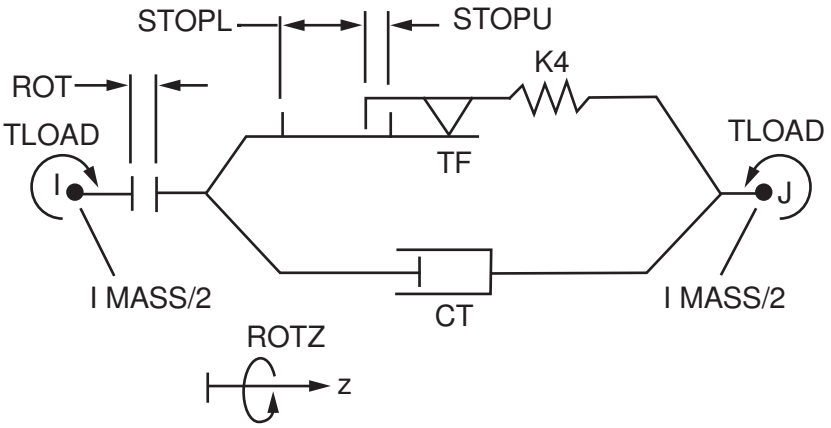
The geometry, node locations, and coordinate systems for this element are shown in *Figure 1, "COMBIN7 Geometry"*. This element is defined in 3-D space by five nodes, these being active nodes (I, J), a node to define the initial revolute axis (K), and control nodes (L, M). The active nodes should be coincident and represent the actual pin joint which connects links A and B. A link can be an individual element or an assemblage of elements. If node K is not defined, then the initial revolute axis is taken to be the Z-direction of the global Cartesian system. The local element coordinate system, when used with large deflection [**NLGEOM**], follows the average translation and rotation of nodes I and J. The element coordinate system x-y-z translates and rotates with the joint, and the orientation of node K is inconsequential after the first iteration. The control nodes' primary aim is to introduce feedback behavior to the element (discussed below).

The active nodes (I, J) are defined to have six degrees of freedom; however, five of these (UX, UY, UZ, ROTX, ROTY) in the local joint system are intended to be constrained with a certain level of flexibility. This level of flexibility is defined by three input stiffnesses: K1 for translational stiffness in the x-y plane, K2 for stiffness in the z direction, and K3 for rotational stiffness about the x and y axes. Joint mass (MASS) and mass moment of inertia (IMASS) input values are evenly distributed between nodes I and J.

The dynamics of the revolute rotation or primary degree of freedom (*Figure 2, "COMBIN7 Real Constants and Dynamic Behavior of Rotation about the Z (Revolute) Axis"*) include friction torque (TF), rotational viscous friction (CT), torsional stiffness (K4), preload torque (TLOAD), interference rotation (ROT), and two differential rotation limits

(STOPL and STOPU). A null value for TF corresponds to zero friction (or free rotation), while a negative value will remove friction capability from the element. Once removed (TF < 0), the joint is locked with stiffness K4. The joint will also become locked with stiffness K4 when a stop is engaged. The upper stop (STOPU) represents the allowed amount of forward rotation (node J rotates away from node I), and the lower stop (STOPL) represents the allowed amount of reverse rotation (node J rotates towards node I). Null values for both stops will remove locking action from the element; i.e., rotation damped only by viscous (CT) and friction (TF) damping torques.

Figure 2 COMBIN7 Real Constants and Dynamic Behavior of Rotation about the Z (Revolute) Axis



The rotational interference (ROT) is intended to correspond to a locally imposed joint rotation if the revolute axis is locked (TF < 0) and stiffness is specified (K4 > 0). A starting status real constant (START) will set the initial behavior of the revolute rotation: START = 0 implies no rotation (locked), START = 1 or -1 implies forward or reverse rotation, respectively. Initial rotation status (START = 1,-1) will be overruled if either START = 1, STOPU = 0, and STOPL $\neq$ 0, or START = -1, STOPL = 0, and STOPU $\neq$ 0.

Consistent units should be used. Units are force/length for K1 and K2 and length*force/radian for K3 and K4. CT uses length*force*time/radian, while TF and TLOAD uses length*force. Force*time^2/length is used with MASS and length*force*time^2/radian is for IMASS. ROT, STOPL, and STOPU use radians.

Feedback control behavior is associated with the control nodes (L, M). The KEYOPT values are used to define the control value (CVAL). KEYOPT(3) selects the degree of freedom for the control nodes, KEYOPT(4) assigns the coordinate system for the selected degree of freedom, and KEYOPT(7) specifies which real constant is to be modified for the subsequent nonlinear analysis. The KEYOPT(1) option assigns to the control value either the value of the degree of freedom, the first or second derivative of the value, the integral of the value, or time.

KEYOPT(2) defines the behavior of the revolute degree of freedom after a stop has been engaged. If KEYOPT(2) = 0, the pin may disengage (or bounce off) the stop. If KEYOPT(2) = 1, the pin axis is locked.

The element can exhibit nonlinear behavior according to the function: $RVMOD = RVAL + C1|CVAL|^{C2} + C3|CVAL|^{C4}$, where RVMOD is the modified value of the input real constant value RVAL (identified by KEYOPT(7)), C1 through C4 are other real constants and give the form of the real constant modification, and CVAL is the control value (see KEYOPT(1)). RVMOD may also be defined by user subroutine USERRC and is accessed by KEYOPT(9) = 1. Examples of CVAL are:

$$CVAL = UX_L - UX_M$$

$$CVAL = d(UZ_L - UZ_M)/dt^2$$

$$CVAL = d^2(ROTZ_L - ROTZ_M)/dt^2$$

$$CVAL = \int_o^t (UY_L - UY_M)dt$$

$$CVAL = t$$

Control values calculated in the current substep are not used until the next substep. Control nodes need not be connected to any other element. If node M is not defined, the control value is based only upon node L.

A summary of the element input is given in *COMBIN7 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBIN7 Input Summary

Nodes
I, J, K, L, M (K, L, M are optional)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

K1, K2, K3, K4, CT, TF
MASS, IMASS, TLOAD, START, STOPL, STOPU
ROT, C1, C2, C3, C4
See *Table 1, "COMBIN7 Real Constants"* for a description of the real constants

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features

Large deflection
Nonlinear (if either stops or friction are specified)
Adaptive descent

KEYOPT(1)
Control Value:

0, 1 --
Control on value (UL-UM) (or UL if M not defined)

2 --
Control on first derivative of value with respect to time

3 --
Control on second derivative of value with respect to time

4 --
Control on integral of value with respect to time

5 --
Control on time value (KEYOPT(3) ignored)

KEYOPT(2)
Behavior when stop is engaged:

0 --
Reverse pin-axis rotation is not prevented when a rotational stop is engaged.

1 --
Pin-axis is locked when a rotational stop is engaged (only after the first substep)

KEYOPT(3)
Degree of freedom for control nodes (L and M):

0, 1 --
UX (Displacement along X axes)

2 --
UY (Displacement along Y axes)

3 --
UZ (Displacement along Z axes)

4 --
ROTX (rotation about X axes)

5 --
ROTY (rotation about Y axes)

6 --
ROTZ (rotation about Z axes

KEYOPT(4)
Control node coordinates:

0 --
Control node degree of freedom is in nodal coordinates

1 --
Control node degree of freedom is in element (moving) coordinates

KEYOPT(7)
Real constant used for RVMOD function (used if C1 or C3 is not equal to zero; see *COMBIN7 Input Data*):

0, 1 --
Use K1 for nonlinear function

2 --
Use K2

3 --
Use K3

4 --
Use K4

5 --
Use CT

6 --
Use TF

7 --
Use MASS

8 --
Use IMASS

9 --
Use TLOAD

10 --
Use START

11 --
Use STOPL

12 --
Use STOPU

13 --
Use ROT

KEYOPT(9)
Method to define nonlinear behavior:

0 --
Use RVMOD expression for real constant modifications

1 --
Real constants modified by user subroutine USERRC (see the *Guide to ANSYS User Programmable Features* for information about user written subroutines)

Table 1 COMBIN7 Real Constants

No.	Name	Description
1	K1	X-Y translational stiffness
2	K2	Z direction stiffness
3	K3	Rotational-X and Rotational-Y stiffness
4	K4	Torsional stiffness
5	CT	Rotational viscous friction
6	TF	Friction torque
7	MASS	Joint mass
8	IMASS	Mass moment of inertia
9	TLOAD	Preload torque
10	START	Starting status
11	STOPL	Lower stop (reverse rotation)
12	STOPU	Upper stop (forward rotation)
13	ROT	Rotational interference
14	C1	First scalar in RVMOD equation
15	C2	First exponential in RVMOD equation
16	C3	Second scalar in RVMOD equation
17	C4	Second exponential in RVMOD equation

COMBIN7 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

- Additional element output as shown in *Table 2, "COMBIN7 Element Output Definitions"*.

It is important to note that element forces and displacements are in the element (moving) coordinate system. The amount of rotational sliding (ROTATE) differs from the total differential rotation (DRZ) about the local revolute axis due to the flexible nature of the joint. STAT and OLDST refer to present and previous statuses, respectively, of the revolute axis. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 COMBIN7 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Active nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
ROTATE	Amount of pin rotational sliding	Y	Y
CVAL	Value (see KEYOPT(1)) of the control nodes	Y	Y
STAT	Element status	*1*	*1*
OLDST	Stat values of the previous time step	*1*	*1*
DUX, DUY, DUZ, DRX, DRY, DRZ	Differential pin displacements and rotations in element coordinates. For example, DUX = UXJ-UXI.	Y	Y
RVMOD	Modified real constant (see *COMBIN7 Input Data*)	Y	Y
FORCE(X, Y, Z)	Spring forces (in element coordinates)	Y	Y
MOMENT(X, Y, Z}	Spring moments (in element coordinates)	Y	Y
RVOLD	Modified real constant of previous time step	Y	Y

1. Element status values:

   0 - No rotation (but no stop engaged)

   1 - Forward rotation

   -1 - Reverse rotation

   2 - Forward stop engaged

   -2 - Reverse stop engaged
2. Available only at centroid as a ***GET** item.

Table 3, "COMBIN7 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "COMBIN7 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "COMBIN7 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 COMBIN7 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCEX	SMISC	1
FORCEY	SMISC	2
FORCEZ	SMISC	3
MOMENTX	SMISC	4
MOMENTY	SMISC	5
MOMENTZ	SMISC	6
STAT	NMISC	1
OLDST	NMISC	2
DUX	NMISC	3
DUY	NMISC	4
DUZ	NMISC	5
DRX	NMISC	6
DRY	NMISC	7
DRZ	NMISC	8
ROTATE	NMISC	9
RVMOD	NMISC	10
CVAL	NMISC	11

COMBIN7 Assumptions and Restrictions

- The joint element is valid only in a structural analysis.
- The active joint nodes (I, J) must be coincident. Node K, if defined, must not be coincident with the active nodes. The control nodes (L, M) may be any active node in the model, including nodes I, J, and K.
- The nonlinear capabilities of the element operate only in static and nonlinear transient dynamic analyses. If used in other analysis types, the element maintains its initial status throughout the analysis. An iterative solution is required when using the nonlinear option.
- The precise nature of the element behavior, whether nonlinearities are present or not, depends on several input items. These include the presence of stops or friction; the selection of a large deflection analysis; and the use of joint control features.
- Stop input values (STOPL, STOPU) must be greater than or equal to zero. For stops to be engaged, a positive torsional stiffness (K4) should be specified. Negative values are ignored. Stop values represent forward and reverse clearances about the revolute axis.
- Revolute friction (TF), when specified, must be positive. A negative friction value removes friction from the element and locks the revolute axis with torsional stiffness K4. A null friction value implies frictionless rotation (unless CT is specified or a stop is engaged).

- The element can not be deactivated with the **EKILL** command.
- The real constants for this element can not be changed from their initial values.
- Only the lumped mass matrix is available.

COMBIN7 Product Restrictions

There are no product-specific restrictions for this element.

LINK8

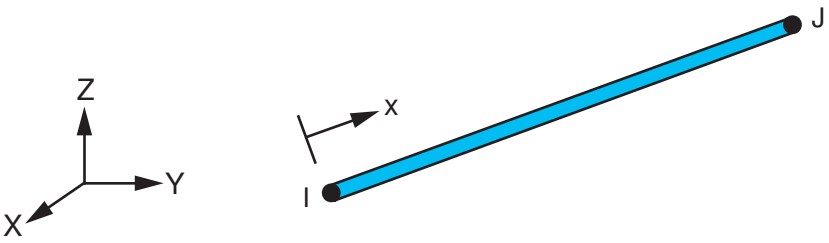
3-D Spar (or Truss)

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

LINK8 Element Description

LINK8 is a spar which may be used in a variety of engineering applications. This element can be used to model trusses, sagging cables, links, springs, etc. The 3-D spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, swelling, stress stiffening, and large deflection capabilities are included. See LINK8 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See LINK10 for a tension-only/compression-only element.

Figure 1 LINK8 Geometry



LINK8 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "LINK8 Geometry"*. The element is defined by two nodes, the cross-sectional area, an initial strain, and the material properties. The element x-axis is oriented along the length of the element from node I toward node J. The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero strain length.

Element loads are described in *Section 2.8: Node and Element Loads*. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature T(J) defaults to T(I). Similar defaults occurs for fluence except that zero is used instead of TUNIF.

A summary of the element input is given in *LINK8 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK8 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

AREA - Cross-sectional area
ISTRN - Initial strain

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Fluences --
FL(I), FL(J)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPTs
None

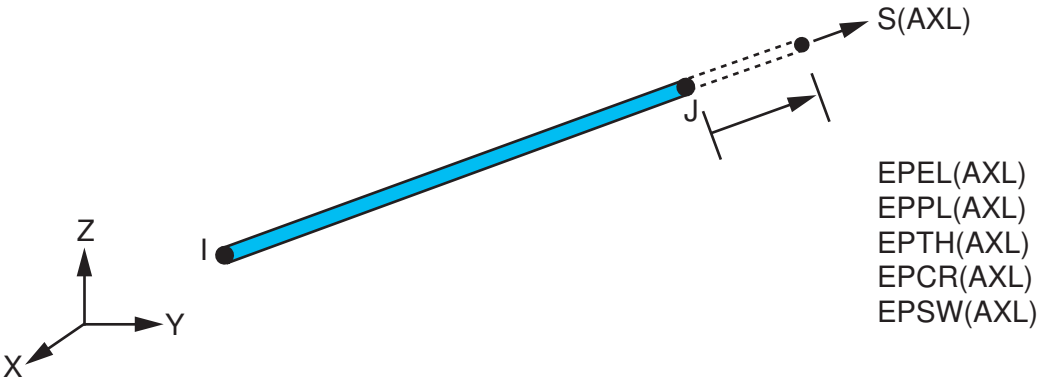
LINK8 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK8 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "LINK8 3-D Spar Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 LINK8 3-D Spar Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK8 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J)	Y	Y
FLUEN	Fluences FL(I), FL(J)	Y	Y
MFORX	Member force in the element coordinate system	Y	Y
SAXL	Axial stress	Y	Y
EPELAXL	Axial elastic strain	Y	Y
EPTHAXL	Axial thermal strain	Y	Y
EPINAXL	Axial initial strain	Y	Y
SEPL	Equivalent stress from stress-strain curve	*1*	*1*
SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
EPEQ	Equivalent plastic strain	*1*	*1*
HPRES	Hydrostatic pressure	*1*	*1*
EPPLAXL	Axial plastic strain	*1*	*1*
EPCRAXL	Axial creep strain	*1*	*1*
EPSWAXL	Axial swelling strain	*1*	*1*

1. Nonlinear solution, only if element has a nonlinear material.
2. Available only at centroid as a ***GET** item.

Table 2, "LINK8 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "LINK8 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "LINK8 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J
: sequence number for data at nodes I and J

Table 2 LINK8 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SAXL	LS	1	-	-
EPELAXL	LEPEL	1	-	-
EPTHAXL	LEPTH	1	-	-
EPSWAXL	LEPTH	2	-	-
EPINAXL	LEPTH	3	-	-
EPPLAXL	LEPPL	1	-	-
EPCRAXL	LEPCR	1	-	-
SEPL	NLIN	1	-	-
SRAT	NLIN	2	-	-
HPRES	NLIN	3	-	-
EPEQ	NLIN	4	-	-
MFORX	SMISC	1	-	-
FLUEN	NMISC	-	1	2
TEMP	LBFE	-	1	2

LINK8 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends, and of uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The area must be greater than zero.
- The temperature is assumed to vary linearly along the length of the spar.
- The displacement shape function implies a uniform stress in the spar.
- The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.

LINK8 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads are not applicable.
- The only special features allowed are stress stiffening and large deflections.

INFIN9

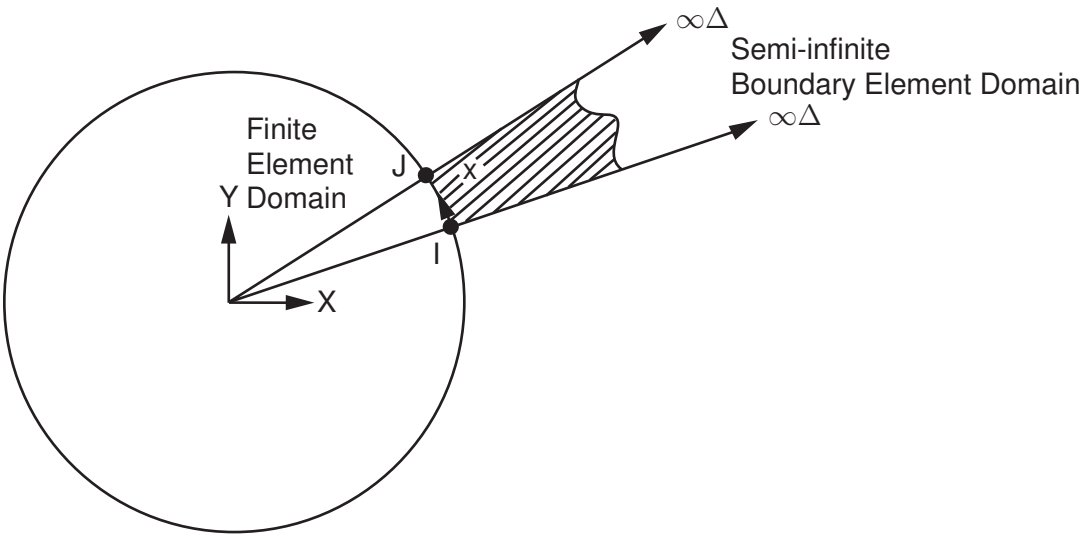
2-D Infinite Boundary

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

INFIN9 Element Description

INFIN9 is used to model an open boundary of a 2-D planar unbounded field problem. The element has two nodes with a magnetic vector potential or temperature degree of freedom at each node. The enclosed element type can be the PLANE13 or PLANE53 magnetic elements, or the PLANE55, PLANE77, and PLANE35 thermal elements. With the magnetic degree of freedom (AZ) the analysis may be linear or nonlinear, static or dynamic. With the thermal degree of freedom steady-state or transient analysis (linear or nonlinear) may be done. See INFIN9 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 INFIN9 Geometry



INFIN9 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "INFIN9 Geometry"*, and a typical application is shown in *Figure 2, "INFIN9 Element Usage"*. The element is defined by two nodes and the material properties. Nonzero material properties must be defined. The element x-axis is oriented along the length of the element from node I toward node J.

The coefficient matrix of this boundary element is, in general, unsymmetric. The matrix is made symmetric by averaging the off-diagonal terms to take advantage of a symmetric solution with a slight decrease in accuracy. KEYOPT(2) can be used to prevent an unsymmetric matrix from being made symmetric.

A summary of the element input is given in *INFIN9 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

INFIN9 Input Summary

Nodes
I, J

Degrees of Freedom

AZ if KEYOPT (1) = 0
TEMP if KEYOPT (1) = 1

Real Constants
None

Material Properties

MUZERO if KEYOPT (1) = 0 (has default value for MKS units or can be set with the **EMUNIT** command)
KXX if KEYOPT (1) = 1

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Element degree of freedom:

0 --
Magnetic option (AZ degree of freedom)

1 --
Thermal option (TEMP degree of freedom)

KEYOPT(2)
Coefficient matrix formation:

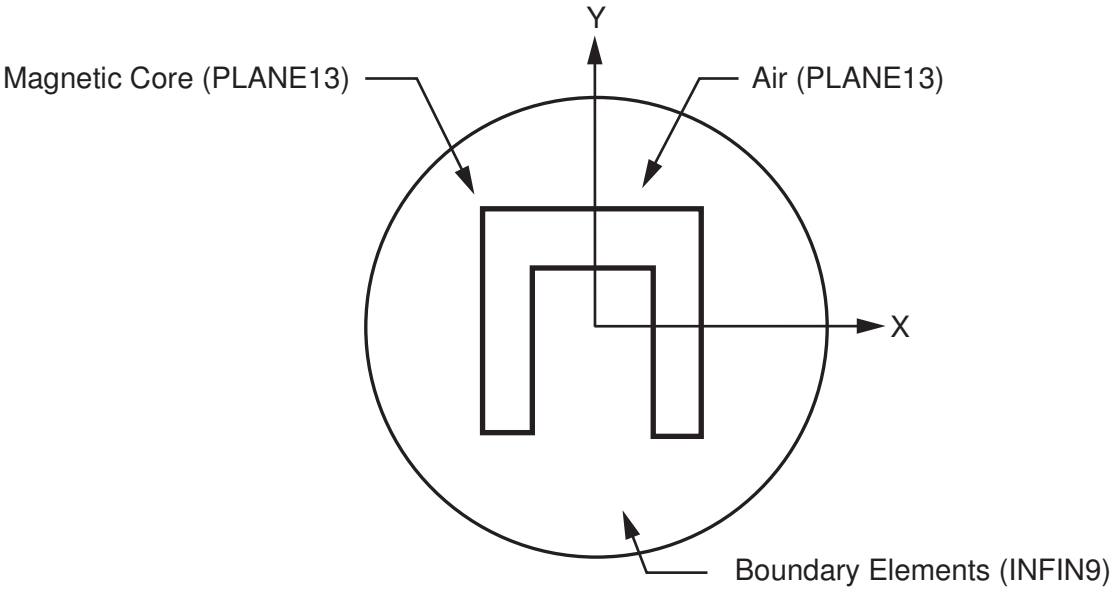
0 --
Make the coefficient matrix symmetric

1 --
Coefficient matrix is used as generated (symmetric or unsymmetric, depending on the problem)

INFIN9 Output Data

The boundary element has no output of its own since it is used only to provide a semi-infinite boundary condition to a model consisting of other elements.

Figure 2 INFIN9 Element Usage



INFIN9 Assumptions and Restrictions

- The boundary element assumes a straight line segment, the length of which must be nonzero.
- The semi-infinite sector is assumed to be bound on three sides by the boundary element, by a semi-infinite radial line from the global coordinate system origin through node I, and by a semi-infinite radial line from the global coordinate system origin through node J. The boundary element should be as normal as possible to the radial lines.
- Acute or wide intersection angles should be avoided by "filling-in" the model with the other elements so that the line of boundary elements around the model is smooth and concave when viewed from the global coordinate system origin.
- The boundary element must lie "against" an enclosed element (that is, share the same nodes).
- The origin of the global coordinate system must be inside the model and as centrally located as possible.
- The line of boundary elements should be located away from the region of interest of the enclosed elements for better accuracy.
- The line of boundary elements need not totally surround the model.
- The exterior semi-infinite domain is assumed to be homogeneous, isotropic, and linear without containing any sources or sinks.
- An axisymmetric option is not available.
- The element cannot be deactivated with the **EKILL** command.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.
- When used in a model with the higher-order elements PLANE53, PLANE77, or PLANE35, the midside nodes of these elements must be removed at the interface with INFIN9 [**EMID**].
- If KEYOPT(2) = 1, the matrices are presumed to be unsymmetric.

INFIN9 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic field capability.
- The AZ degree of freedom is not active.
- KEYOPT(1) defaults to 1 (TEMP) instead of 0 and cannot be changed.
- The material property MUZERO is not allowed.

ANSYS Emag

- This element has only magnetic field capability, and does not have thermal capability.
- The only active degree of freedom is AZ.
- The only allowable material property is MUZERO.
- KEYOPT(1) can only be set to 0 (default).

LINK10

Tension-only or Compression-only Spar

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

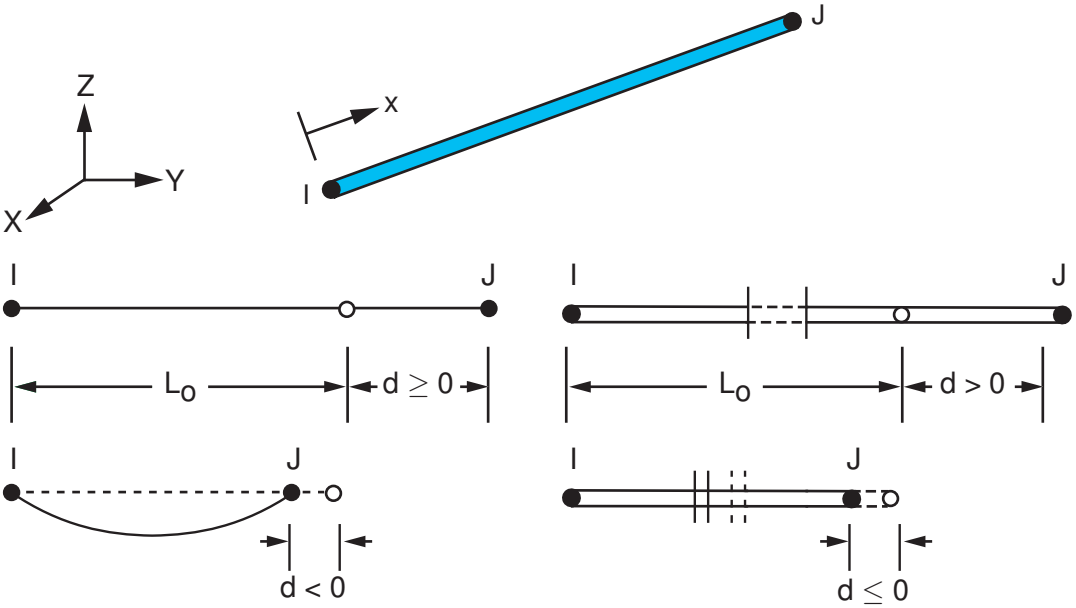
LINK10 Element Description

LINK10 is a 3-D spar element having the unique feature of a bilinear stiffness matrix resulting in a uniaxial tension-only (or compression-only) element. With the tension-only option, the stiffness is removed if the element goes into compression (simulating a slack cable or slack chain condition). This feature is useful for static guy-wire applications where the entire guy wire is modeled with one element. It may also be used in dynamic analyses (with inertia or damping effects) where slack element capability is desired but the motion of the slack elements is not of primary interest. This element is a line version of SHELL41 with KEYOPT(1) = 2, the "cloth" option.

If the purpose of the analysis is to study the motion of the elements (with no slack elements), a similar element which cannot go slack, such as LINK8 or PIPE59, should be used instead. LINK10 should also not be used for static convergence applications where the final solution is known to be a taut structure but a slack condition is possible while iterating to a final converged solution. For this case either a different element should be used or the "slow dynamic" technique should be used if LINK10 is desired.

LINK10 has three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending stiffness is included in either the tension-only (cable) option or the compression-only (gap) option but may be added by superimposing a beam element with very small area on each LINK10 element. Stress stiffening and large deflection capabilities are available. See LINK10 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK10 Geometry



Tension-Only (Cable) Option

Compression-Only (Gap) Option

LINK10 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "LINK10 Geometry"*. The element is defined by two nodes, the cross-sectional area, an initial strain or gap, and the isotropic material properties. The element x-axis is oriented along the length of the element from node I toward node J.

The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero strain length, L_o. For the cable option, a negative strain indicates a slack condition. For the gap option, a positive strain indicates a gap condition (as shown in *Figure 1, "LINK10 Geometry"*). The gap must be input as a "per unit length" value.

Element loads are described in *Section 2.8: Node and Element Loads*. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature T(J) defaults to T(I).

KEYOPT(2) is used to apply a small stiffness (AE x 10^{-6}/L) across an open gap or to a slack cable to prevent unconstrained portions of the structure from "floating free" if the gap opens or the cable goes slack.

A summary of the element input is given in *LINK10 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK10 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

AREA - Cross-sectional area
ISTRN - Initial strain
If KEYOPT(3) = 0 and ISTRN is less than zero, the cable is initially slack.
If KEYOPT(3) = 1 and ISTRN is greater than zero, the gap is initially open.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Special Features

Nonlinear
Stress stiffening
Large deflection
Birth and death

KEYOPT(2)
Stiffness for slack cable:

0 --
No stiffness associated with slack cable

1 --
Small stiffness assigned to slack cable for longitudinal motion

2 --
Small stiffness assigned to slack cable for both longitudinal and perpendicular motions (applicable only with stress stiffening)

KEYOPT(3)
Tension / compression option:

0 --
Tension-only (cable) option

1 --
Compression-only (gap) option

LINK10 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK10 Element Output Definitions"*.

The axial force, stress, and strain in the element are printed. Only positive values are obtained with the cable option and negative values with the gap option. The element condition (tension or slack, compression or gap) at the end of this substep is indicated by the value of STAT. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK10 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*2*
STAT	Element status	*1*	*1*
TEMP	Temperatures T(I), T(J)	Y	Y
MFORX	Member force in the element coordinate system	Y	Y
SAXL	Axial stress	Y	Y
EPELAXL	Axial elastic strain	Y	Y
EPTHAXL	Axial thermal strain	Y	Y
EPINAXL	Axial initial strain	Y	Y

1. Element status values:

   1 - cable in tension or gap in compression

   2 - cable slack or gap open

2. Available only at centroid as a ***GET** item.

Table 2, "LINK10 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "LINK10 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "LINK10 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 2 LINK10 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SAXL	LS	1	-	-
EPELAXL	LEPEL	1	-	-
EPTHAXL	LEPTH	1	-	-
EPINAXL	LEPTH	3	-	-
MFORX	SMISC	1	-	-
STAT	NMISC	1	-	-
OLDST	NMISC	2	-	-
TEMP	LBFE	-	1	2

LINK10 Assumptions and Restrictions

- The element length must be greater than zero, therefore nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The temperature is assumed to vary linearly along the length of the element.
- The element is nonlinear and requires an iterative solution.
- If ISTRN is 0.0, the element stiffness is included in the first substep.
- With the gap (compression-only) option, a positive axial displacement (in the element coordinate system) of node J relative to node I tends to open the gap.
- The solution procedure is as follows: The element condition at the beginning of the first substep is determined from the initial strain or gap input. If this value is less than zero for the cable option or greater than zero for the gap option, the element stiffness is taken as zero for this substep. If at the end of the substep STAT = 2, an element stiffness of zero is used for the next substep. If STAT = 1, the element stiffness is in-

cluded in the next substep. No significant stiffness is associated with the cable option having a negative relative displacement or with the gap option having a positive relative displacement.

- If the element status changes within a substep, the effect of the changed status is included in the next substep.
- Nonconverged substeps are not in equilibrium.
- The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.
- Stress stiffening should always be used for sagging cable problems to provide numerical stability. Stress stiffening and large deflection effects may be used together for some cable problems (see the *Theory Reference for ANSYS and ANSYS Workbench*).

LINK10 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The birth and death special feature is not allowed.

LINK11

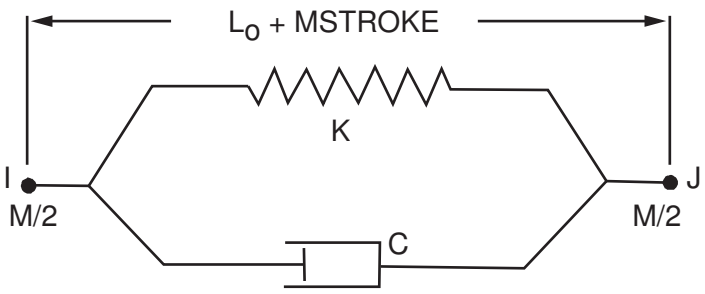
Linear Actuator

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

LINK11 Element Description

LINK11 may be used to model hydraulic cylinders and other applications undergoing large rotations. The element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or twist loads are considered. See LINK11 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK11 Geometry



LINK11 Input Data

The geometry and node locations for the element are shown in *Figure 1, "LINK11 Geometry"*. The element is defined by two nodes, a stiffness, viscous damping, and mass. The element initial length L_o and orientation are determined from the node locations.

Element loads are described in *Section 2.8: Node and Element Loads*. The stroke (length) is defined through the surface load input using the PRES label. The stroke is relative to the zero force position of the element. A force may be defined in the same manner as an alternate to the stroke.

A summary of the element input is given below. A general description of element input is given in *Section 2.1: Element Input*.

LINK11 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

K - Stiffness (force/length)
C - Viscous damping coefficient (force*time/length)
M - Mass (force*time^2/length)

Material Properties
DAMP

Surface Loads

Pressures --

face 1 - Stroke
face 2 - Axial Force

Body Loads
None

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPTs
None

LINK11 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal displacement solution
- Additional element output as shown in *Table 1, "LINK11 Element Output Definitions"*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK11 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
ILEN	Initial element length	Y	Y
CLEN	Current element length (this time step)	Y	Y
FORCE	Axial force (spring force)	Y	Y
DFORCE	Damping force	Y	Y
STROKE	Applied stroke (element load)	Y	Y
MSTROKE	Measured stroke	Y	Y

Table 2, "LINK11 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. The following notation is used in *Table 2, "LINK11 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "LINK11 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 LINK11 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCE	SMISC	1
ILEN	NMISC	1
CLEN	NMISC	2
STROKE	NMISC	3
MSTROKE	NMISC	4
DFORCE	NMISC	5

LINK11 Assumptions and Restrictions

- The element must not have a zero length.
- The element assumes a straight line, axially loaded at the ends.
- A twist (torsion) about the element x-axis (defined from node I to node J) has no effect.
- No bending of the element is considered, as in a pin-jointed structure.
- The mass is equally divided between the nodes.
- Only the lumped mass matrix is available.
- Surface load pressure indicators are not displayed for element or node plots.

LINK11 Product Restrictions

There are no product-specific restrictions for this element.

CONTAC12

2-D Point-to-Point Contact

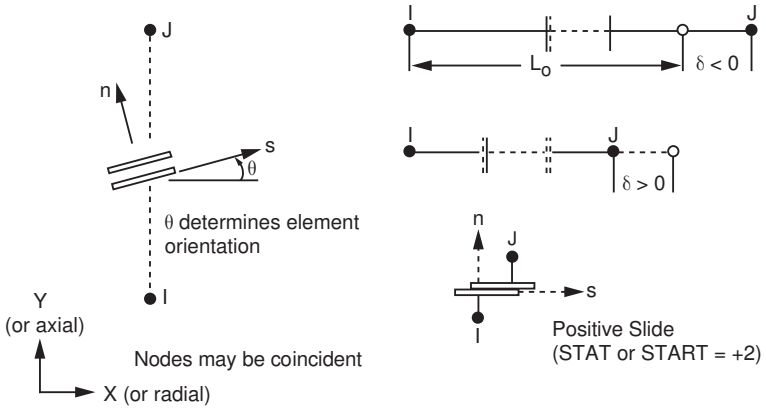
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

CONTAC12 Element Description

CONTAC12 represents two surfaces which may maintain or break physical contact and may slide relative to each other. The element is capable of supporting only compression in the direction normal to the surfaces and shear (Coulomb friction) in the tangential direction. The element has two degrees of freedom at each node: translations in the nodal x and y directions.

The element may be initially preloaded in the normal direction or it may be given a gap specification. A specified stiffness acts in the normal and tangential directions when the gap is closed and not sliding. See CONTAC12 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other contact elements, such as COMBIN40 and CONTAC52, are also available.

Figure 1 CONTAC12 Geometry



CONTAC12 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "CONTAC12 Geometry"*. The element is defined by two nodes, an angle to define the interface, two stiffnesses (KN and KS), an initial displacement interference or gap (INTF), and an initial element status (START). An element coordinate system (s-n) is defined on the interface. The angle θ (THETA) is input (or calculated) in degrees and is measured from the global X axis to the element s-axis. The orientation of the interface may be defined (KEYOPT(2)) by THETA or by the node locations.

The normal stiffness, KN, should be based upon the stiffness of the surfaces in contact. See *Section 8.2: Performing a Node-to-Node Contact Analysis* in the *Contact Technology Guide* for guidelines on choosing a value for KN. In some cases (such as initial interference analyses, nonconvergence, or over penetration), it may be useful to change the KN value between load steps or in a restart in order to obtain an accurate, converged solution. The sticking stiffness, KS, represents the stiffness in the tangential direction when elastic Coulomb friction is selected ($\mu > 0.0$ and KEYOPT(1) = 0). The coefficient of friction μ is input as material property MU and is evaluated at the average of the two node temperatures. Stiffnesses may also be computed from the maximum expected force divided by the maximum allowable surface displacement. KS defaults to KN. Stiffnesses should be on a full 360° basis for an axisymmetric analysis.

The initial displacement interference, Δ, defines the displacement interference (if positive) or the gap size (if negative). The value may be input as a real constant (INTF) or automatically calculated from the input node locations if KEYOPT(4) = 1. Stiffness is associated with a zero or positive interference. The initial element status (START) is used to define the "previous" condition of the interface to be used at the start of the first substep. This input is used to override the condition implied by the interference specification and is useful in anticipating the final interface configuration and in reducing the number of iterations required for convergence.

The force deflection relationships for the interface element can be separated into the normal and tangential (sliding) directions as shown in *Figure 2, "CONTAC12 Force-Deflection Relationship"*. The element condition at the beginning of the first substep is determined from the START parameter. If the interface is open, no stiffness is associated with this element for this substep. If the interface is closed and sticking, KN is used in the gap resistance and KS is used in the sliding resistance. If the interface is closed but sliding, KN is used in the gap resistance and the limit friction force μFN is used for the sliding resistance.

In the normal direction, when the normal force (FN) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted.

KEYOPT(3) can be used to specify a "weak spring" across an open interface, which is useful for preventing rigid body motion that could occur in a static analysis. The weak spring stiffness is computed by multiplying the normal stiffness KN by a reduction factor. The default reduction factor of 1E-6 can be overridden with real constant REDFACT.

In the tangential direction, for FN < 0 and the absolute value of the tangential force (FS) less than (μ|FN|), the interface sticks and responds as a linear spring in the tangential direction. For FN < 0 and FS = μ|FN|, sliding occurs.

If KEYOPT(1) = 1, rigid Coulomb friction is selected, KS is not used, and the elastic sticking capability is removed. This option is useful for displacement controlled problems or for certain dynamic problems where sliding dominates. With this option, no tangential resistance is assumed for the first substep.

The only material property used is the interface coefficient of friction MU. A zero value should be used for frictionless surfaces. Temperatures may be input at the element nodes (for material property evaluation only). The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I). The circular gap option (KEYOPT(2)) is useful where the final contact point (and thus the orientation angle) is not known, such as with concentric cylinders. With this option the angular orientation THETA is initially set to 0.0 and then internally calculated from the relative displacements of the nodes at the end of the substep for use in the next substep. The user specified THETA (if any) is ignored. A negative interference (gap) and a zero coefficient of friction is used with this option.

For analyses involving friction, using **NROPT**,UNSYM is useful (and, in fact, sometimes required) for problems where the normal and tangential (sliding) motions are strongly coupled, such as in a wedge insertion problem.

A summary of the element input is given in *CONTAC12 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTAC12 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY

Real Constants
See *Table 1, "CONTAC12 Real Constants"* for details on these real constants

Material Properties
DAMP, MU

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Special Features

Nonlinear
Adaptive descent

KEYOPT(1)
Type of friction (only with MU > 0.0):

0 --
Elastic coulomb friction (KS used for sticking stiffness)

1 --
Rigid coulomb friction (resisting force only)

KEYOPT(2)
Orientation angle:

0 --
Orientation angle based on Theta real constant

1 --
Circular gap option (THETA orientation determined from direction of motion) (ignore THETA real constant)

KEYOPT(3)
Weak spring across open gap:

0 --
No weak spring across an open gap

1 --
Use a weak spring across an open gap

KEYOPT(4)
Interference or gap:

0 --
Interference (or gap) based on INTF real constant

1 --
Interference (or gap) based on initial node locations (ignore INTF real constant)

KEYOPT(7)
Element level time incrementation control. Note that this option should be activated first at the procedure level if **SOLCONTROL** is ON. **SOLCONTROL**,ON,ON is the most frequent usage with this element. If **SOLCONTROL**,ON,OFF, this keyoption is not activated.

0 --
Predictions are made to achieve the minimum time (or load) increment whenever a change in contact status occurs

1 --
Predictions are made to maintain a reasonable time (or load) increment (recommended)

Table 1 CONTAC12 Real Constants

No.	Name	Description
1	THETA	Interference angle
2	KN	Normal stiffness
3	INTF	Initial displacement interference or gap. A negative INTF (interference) assumes an intially open gap.
4	START	Initial element status If = 0.0 or blank, initial condition of gap status is determined from real constant INTF If = 1.0, gap is initially closed and not sliding (if MU $\neq$ 0.0), or sliding node J is positive (if MU = 0.0) If = 2.0, gap is initially closed and node J is sliding to the right of node I If = -2.0, gap is initially closed and node J is sliding to the left of node I If = 3.0, gap is initially open
5	KS	Sticking stiffness
6	REDFACT	KN reduction factor

CONTAC12 Output Data

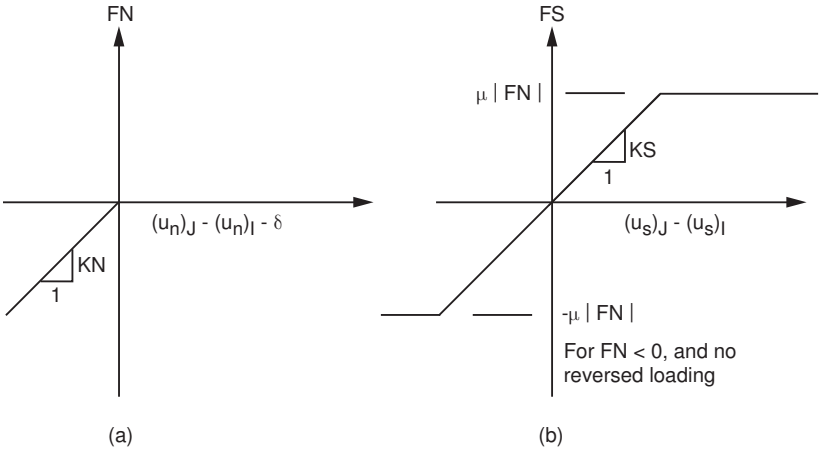
The solution output associated with the element is in two forms:

- nodal displacements included in the overall nodal solution
- additional element output as shown in *Table 2, "CONTAC12 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "CONTAC12 Force-Deflection Relationship"*.

The value of USEP is determined from the normal displacement (u_n) (in the element x-direction) between the interface nodes at the end of this substep. That is: USEP = $(u_n)_J - (u_n)_I - \Delta$. This value is used in determining the normal force, FN. For an axisymmetric analysis, the element forces are expressed on a full 360° basis. The value represented by UT is the total translational displacement. The maximum value printed for the sliding force, FS, is $\mu|FN|$. STAT describes the status of the element at the end of this substep. If STAT = 1, the gap is closed and no sliding occurs. If STAT = 3, the gap is open. A value of STAT = +2 indicates the node J slides positive relative to node I as shown in Figure 4.12-1. STAT = -2 indicates a negative slide. For a frictionless surface (μ = 0.0), the element status is either STAT = $\pm$2 or 3. The value of THETA is either the input orientation angle (if KEYOPT(2) = 0), or the calculated angle (if KEYOPT(2) = 1). A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 CONTAC12 Force-Deflection Relationship



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTAC12 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC	Location where results are reported	Y	*3*
TEMP	Temperatures T(I), T(J)	Y	Y
USEP	Gap size or interference	Y	Y
FN	Normal force	Y	Y
STAT	Element status	*1*	*1*
OLDST	Stat value of the previous time step	*1*	*1*
THETA	Orientation angle	Y	Y
MU	Coefficient of friction	*2*	*2*
UT	Relative displacement in tangential direction (positive for node J moving to right of node I)	*2*	*2*
FS	Tangential force	*2*	*2*

1. Element status values:

   1 - Contact, no sliding

   2 - Sliding contact with node J moving to right of node I

   -2 - Sliding contact with node J moving to left of node I

   3 - Gap open

2. Only if MU > 0.0 and KEYOPT(2) = 0.
3. Available only at centroid as a ***GET** item.

Table 3, "CONTAC12 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "CONTAC12 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTAC12 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 CONTAC12 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FN	SMISC	1
FS	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
USEP	NMISC	3
UT	NMISC	4
MU	NMISC	5
THETA	NMISC	6

CONTAC12 Assumptions and Restrictions

- The 2-D interface element must be defined in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- The element operates bilinearly only in a static or a nonlinear transient dynamic analysis.
- If used in other analysis types, the element maintains its initial status throughout the analysis.
- The element is nonlinear and requires an iterative solution.
- Convergence is also based on forces when friction or the circular gap option is present.
- Nodes I and J may be coincident since the orientation of the interface is defined only by the angle THETA.
- The orientation of the interface does not change (with KEYOPT(2) = 0) during a large deflection analysis. Use CONTA175 if this effect is desired.
- No moment effects due to noncoincident nodes are included. That is, if the nodes are offset from a line perpendicular to the interface, moment equilibrium may not be satisfied.
- The element is defined such that a positive normal displacement (in the element coordinate system) of node J relative to node I tends to open the gap, as shown in *Figure 1, "CONTAC12 Geometry"*. If, for a given set of conditions, node I and J are interchanged, or if the interface is rotated by 180°, the gap element acts as a hook element, i.e., the gap closes as the nodes separate. The element may have rotated nodal coordinates since a displacement transformation into the element coordinate system is included.

- The element stiffness KN cannot be exactly zero.
- Unreasonably high stiffness values also should be avoided.
- The rate of convergence decreases as the stiffness increases. Note that, although it is permissible to change KN, it is not permissible to change any other real constants between load steps. Therefore, if you plan to change KN, you cannot allow the value of KS to be defined by default, because the program would then attempt to redefine KS as KN changed.
- You must explicitly define KS whenever KN changes, to maintain a consistent value throughout all load steps.
- The element may not be deactivated with the **EKILL** command.
- If μ is nonzero, the element is nonconservative as well as nonlinear. Nonconservative elements require that the load be applied very gradually, along the actual load history path, and in the proper sequence (if multiple loadings exist).

CONTAC12 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element is frictionless. Specifically, MU is not allowed as a material property and KS is not allowed as a real constant.
- Temperature body loads are not applicable.
- KEYOPT(1) is not applicable.
- The DAMP material property is not allowed.

PLANE13

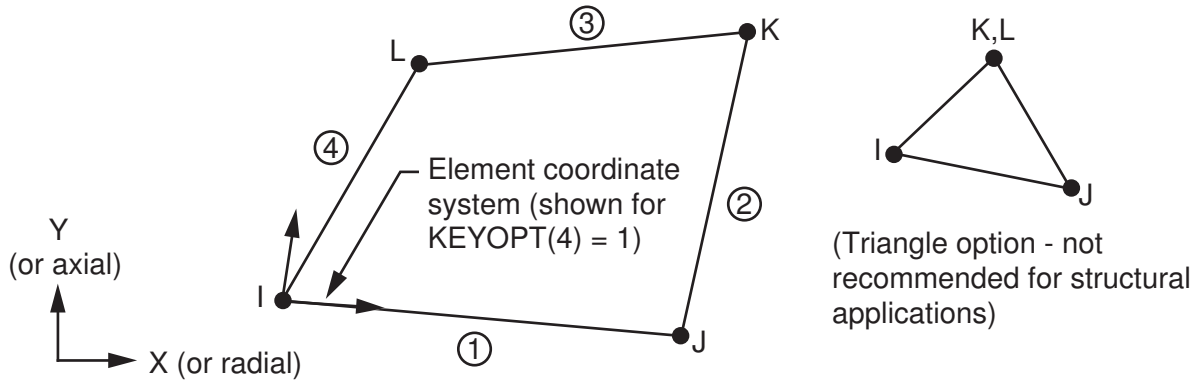
2-D Coupled-Field Solid

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

PLANE13 Element Description

PLANE13 has a 2-D magnetic, thermal, electrical, piezoelectric, and structural field capability with limited coupling between the fields. PLANE13 is defined by four nodes with up to four degrees of freedom per node. The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. PLANE13 has large deflection and stress stiffening capabilities. When used in purely structural analyses, PLANE13 also has large strain capabilities. See PLANE13 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other coupled-field elements are SOLID5, SOLID98, and SOLID62.

Figure 1 PLANE13 Geometry



PLANE13 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE13 Geometry"*. The element input data includes four nodes and magnetic, thermal, electrical, and structural material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4 \pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is specified through the MURX and MURY material property labels.

MGXX and MGYY represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX and MGYY. Permanent magnet polarization and orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Nonlinear magnetic B-H, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. Nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "PLANE13 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated are identified by using the MXWF label on the surface load commands (no value is required). A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

Body loads - temperature, heat generation rate, and magnetic virtual displacement - may be input at the element's nodes or as a single element value [**BF**, **BFE**]. Source current density loads may be applied to an area [**BFA**] or input as an element value [**BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 2 or 4), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal temperatures and heat generation rates default to the uniform value specified with the **BFUNIF** or **TUNIF** command. Heat generation from Joule heating is applied in Solution as thermal loading for static and transient analyses.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

A summary of the element input is given in *PLANE13 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE13 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

AZ if KEYOPT (1) = 0
TEMP if KEYOPT (1) = 2
UX, UY if KEYOPT (1) = 3
UX, UY, TEMP, AZ if KEYOPT (1) = 4
VOLT, AZ if KEYOPT (1) = 6
UX, UY, VOLT if KEYOPT (1) = 7

Real Constants

None

Material Properties

EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ, (or CTEX, CTEY,CTEZ *or* THSX, THSY,THSZ),
DENS, GXY, DAMP,
KXX, KYY, C, ENTH, MUZERO, MURX,
MURY, RSVZ, MGXX, MGYY, PERX, PERY,
plus BH, ANEL, and Piezoelectric data tables (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags--
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Magnetic Virtual Displacements --
VD(I), VD(J), VD(K), VD(L)

Source Current Density --

spare, spare, JSZ(I), PHASE(I), spare, spare,
JSZ(J), PHASE(J), spare, spare, JSZ(K), PHASE(K),
spare, spare, JSZ(L), PHASE(L)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP)
Creep (CREEP)
Swelling (SWELL)
Elasticity (ANEL)
Other material (USER)
Requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
Large deflection
Large strain
Stress stiffening
Birth and death
Adaptive descent

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Element degrees of freedom:

0 --
AZ degree of freedom

2 --
TEMP degree of freedom

3 --
UX, UY degrees of freedom

4 --
UX, UY, TEMP, AZ degrees of freedom

6 --
VOLT, AZ degrees of freedom

7 --
UX, UY, VOLT degrees of freedom

KEYOPT(2)
Extra shapes:

0 --
Include extra shapes

1 --
Do not include extra shapes

KEYOPT(3)
Element behavior:

0 --
Plane strain (with structural degrees of freedom)

1 --
Axisymmetric

2 --
Plane stress (with structural degrees of freedom)

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal stress printout

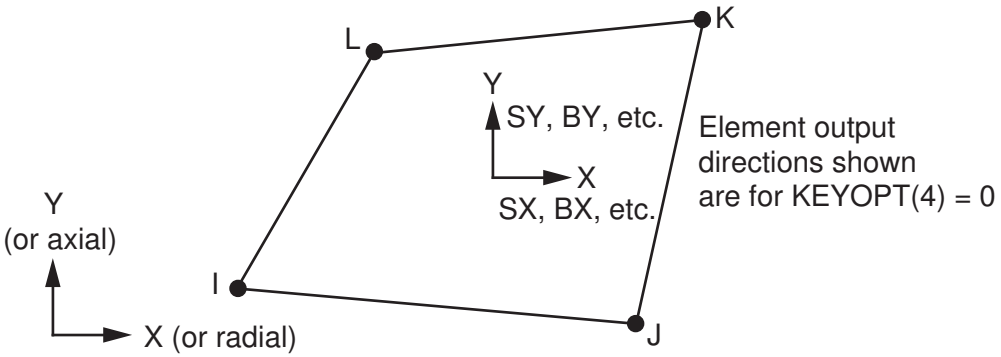
PLANE13 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE13 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "PLANE13 Element Output"*. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE13 Element Output



Because of different sign conventions for Cartesian and polar coordinate systems, magnetic flux density vectors point in opposite directions for planar (KEYOPT(3) = 0) and axisymmetric (KEYOPT(3) =1) analyses.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE13 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*3*
PRES	P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L	Y	Y
TEMP	Input temperatures T(I), T(J), T(K), T(L)	Y	Y
HGEN	Input heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	*1*	*1*
S:1, 2, 3	Principal stresses	*1*	*1*
S:INT	Stress intensity	*1*	*1*
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal elastic strains	*1*	-
EPEL:EQV	Equivalent elastic strain [*4*]	-	*1*
EPTH:X, Y, Z, XY	Average thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strain [*4*]	-	*1*
TG:X, Y, SUM	Thermal gradient components and vector sum	*1*	*1*
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum	*1*	*1*
EF:X, Y	Electric field components (X, Y)	*1*	*1*

Name	Definition	O	R
EF:SUM	Vector magnitude of EF	*1*	*1*
D:X, Y	Electric flux density components (X, Y)	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
UE, UD, UM	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	*1*	*1*
LOC	Output location (X, Y)	*1*	-
MUX, MUY	Magnetic permeability	*1*	*1*
H:X, Y	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y	Magnetic flux density components	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
JSZ	Source current density, available for static analysis only	*1*	*1*
JTZ	Total current density	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
FJB(X, Y)	Lorentz force components	*1*	*1*
FMX(X, Y)	Maxwell force components	*1*	*1*
FVW(X, Y)	Virtual work force components	*1*	*1*
FMAG:X, Y	Combined (FJB and FMX) force components	-	*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
NODES	Face nodes	*2*	-
HFILM	Film coefficient at each node of face	*2*	-
TBULK	Bulk temperature at each node of face	*2*	-
TAVG	Average face temperature	*2*	*2*
HEAT RATE	Heat flow rate across face by convection	*2*	*2*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*2*	-
HFLUX	Heat flux at each node of face	*2*	-
HFAVG	Average film coefficient of the face	*2*	*2*
TBAVG	Average face bulk temperature	-	*2*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*2*
TJB(Z)	Lorentz torque about global Cartesian +Z axis	*1*	*1*
TMX(Z)	Maxwell torque about global Cartesian +Z axis	*1*	*1*
TVW(Z)	Virtual work torque about global Cartesian +Z axis	*1*	*1*

1. Solution values are output only if calculated (based on input data).

Note

For harmonic analysis, joule losses (JHEAT), forces (FJB(X, Y), FMX(X, Y), FVW(X, Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in the "Real" data set. The macros **POWERH**, **FMAGSUM**, and **TORQSUM** can be used to retrieve this data.

2. Available only if a surface load is input.
3. Available only at centroid as a ***GET** item.

4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2 PLANE13 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Pt. Solution	SINT, SEQV, EPEL, S, MUX, MUY, H, HSUM, B, BSUM	*1*	-
Nodal Solution	SINT, SEQV, S, H, HSUM, B, BSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1.
2. Output at each node, if KEYOPT(5) = 2.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively. The X, Y, Z, and XY stress output correspond to the radial, axial, hoop, and in-plane shear stresses, respectively.

For harmonic analysis, joule losses (JHEAT), forces (FJB(X,Y), FMX(X,Y), FVW(X,Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in the "Real" data set. The macros POWERH, FMAGSUM, and TORQSUM can be used to retrieve this data.

Table 3, "PLANE13 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PLANE13 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE13 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

FC*N*
sequence number for solution items for element Face *N*

Table 3 PLANE13 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
JSZ	SMISC	1	-	-	-	-
P1	SMISC	-	4	3	-	-
P2	SMISC	-	-	6	5	-
P3	SMISC	-	-	-	8	7
P4	SMISC	-	9	-	-	10

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
MUX	NMISC	1	-	-	-	-
MUY	NMISC	2	-	-	-	-
FVWX	NMISC	3	-	-	-	-
FVWY	NMISC	4	-	-	-	-
FVWSUM	NMISC	5	-	-	-	-
JTZ	NMISC	7	-	-	-	-
UE	NMISC	8	-	-	-	-
UD	NMISC	9	-	-	-	-
UM	NMISC	10	-	-	-	-
TJB(Z)	NMISC	35	-	-	-	-
TMX(Z)	NMISC	36	-	-	-	-
TVW(Z)	NMISC	37	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	11	17	23	29
HFAVG	NMISC	12	18	24	30
TAVG	NMISC	13	19	25	31
TBAVG	NMISC	14	20	26	32
HEAT RATE	NMISC	15	21	27	33
HFLXAVG	NMISC	16	22	28	34

PLANE13 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE13 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- For structural and piezoelectric problems, the extra displacement and VOLT shapes are automatically deleted for triangular elements so that a constant strain element results.
- Transient magnetic analyses should be performed in a nonlinear transient dynamic analysis.
- A skin-effect analysis (where eddy current formation is permitted in conducting regions with impressed current loading) is performed by using KEYOPT(1) = 6, specifying a resistivity, and coupling all VOLT degrees of freedom for elements in each of such regions. This is valid for both planar and axisymmetric models.
- Current density loading (**BFE**,,JS) is only valid for the AZ option (KEYOPT(1) = 0). For the VOLT, AZ option (KEYOPT(1) = 6) use **F**,,AMPS.
- When this element does not have the VOLT degree of freedom (KEYOPT(1) = 4), for a harmonic or transient analysis, its behavior depends on the applied load. For a **BFE**,,JS load, the element acts as a stranded conductor. Without **BFE**,,JS loads, it acts as a solid conductor modeling eddy current effects.

Note

In this respect, PLANE13 (and PLANE53) are not like the 3-D elements SOLID97 and SOLID117. When SOLID97 and SOLID117 do not have the VOLT degree of freedom, they act as stranded conductors.

- Do not constrain all VOLT DOFs to the same value in a piezoelectric analysis (KEYOPT(1) = 7). Perform a pure structural analysis instead (KEYOPT(1) = 3).
- Permanent magnets are not permitted in a harmonic analysis.
- If a model has at least one element with piezoelectric degrees of freedom (displacements and VOLT) activated, then all elements where a VOLT degree of freedom is needed must be one of the piezoelectric types, and they must all have the piezoelectric degrees of freedom activated. If the piezoelectric effect is not desired in these elements, simply define very small piezoelectric material properties for them.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

PLANE13 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element has only structural, thermal, or piezoelectric capability, and does not have magnetic capability.
- The AZ degree of freedom is not active.
- KEYOPT(1) defaults to 4 (UX, UY, TEMP) instead of 0, and cannot be set to 0. If set to 4 or 6, the AZ degree of freedom is not active.
- The magnetic and electric material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags surface loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are AZ and VOLT.
- The only allowable material properties are the magnetic and electric properties (MUZERO through PERY, plus the BH data table).
- The only applicable surface loads are Maxwell force flags. The heat generation body loads are not applicable. The temperature body load is only used for material property evaluation.
- The element does not allow any special features.
- KEYOPT(1) can only be set to 0 (default) or 6. KEYOPT(3) = 2 is not applicable.

COMBIN14

Spring-Damper

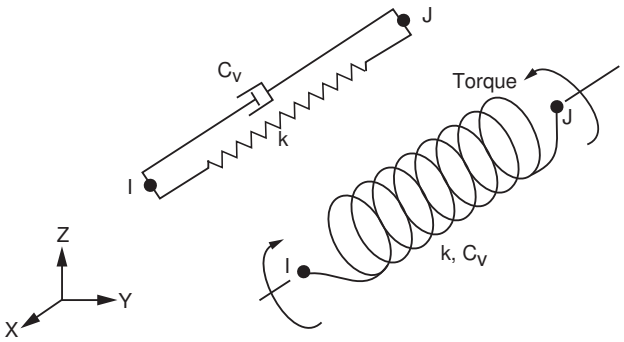
MP ME ST PR PRN DS <> <> <> <> <> PP VT
Product Restrictions

COMBIN14 Element Description

COMBIN14 has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional spring-damper option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.

The spring-damper element has no mass. Masses can be added by using the appropriate mass element (see MASS21). The spring or the damping capability may be removed from the element. See COMBIN14 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A general spring or damper is also available in the stiffness matrix element (MATRIX27). Another spring-damper element (having its direction of action determined by the nodal coordinate directions) is COMBIN40.

Figure 1 COMBIN14 Geometry



2-D elements must lie in a z = constant plane

COMBIN14 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "COMBIN14 Geometry"*. The element is defined by two nodes, a spring constant (k) and damping coefficients $(c_v)_1$ and $(c_v)_2$. The damping capability is not used for static or undamped modal analyses. The longitudinal spring constant should have units of Force/Length, the damping coefficient units are Force*Time/Length. The torsional spring constant and damping coefficient have units of Force*Length/Radian and Force*Length*Time/Radian, respectively. For a 2-D axisymmetric analysis, these values should be on a full 360° basis.

The damping portion of the element contributes only damping coefficients to the structural damping matrix. The damping force (F) or torque (T) is computed as:

$F_x = - c_v du_x/dt$ or $T_\theta = - c_v d\theta/dt$

where c_v is the damping coefficient given by $c_v = (c_v)_1 + (c_v)_2 v$.

v is the velocity calculated in the previous substep. The second damping coefficient $(c_v)_2$ is available to produce a nonlinear damping effect characteristic of some fluid environments. If $(c_v)_2$ is input (as real constant CV2), KEYOPT(1) must be set to 1.

KEYOPT(2) = 1 through 6 is used for defining the element as a one-dimensional element. With these options, the element operates in the nodal coordinate system (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*). The KEYOPT(2) = 7 and 8 options allow the element to be used in a thermal or pressure analysis.

A summary of the element input is given in *COMBIN14 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBIN14 Input Summary

Nodes
I, J

Degrees of Freedom

UX, UY, UZ if KEYOPT (3) = 0
ROTX, ROTY, ROTZ if KEYOPT (3) = 1
UX, UY if KEYOPT (3) = 2
see list below if KEYOPT(2) > 0

Real Constants

K - Spring constant
CV1 - Damping coefficient
CV2 - Damping coefficient (KEYOPT(1) must be set to 1)

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features

Nonlinear (if CV2 is not zero)
Stress stiffening
Large deflection
Birth and death

KEYOPT(1)
Solution type:

0 --
Linear Solution (default)

1 --
Nonlinear solution (required if CV2 is nonzero)

KEYOPT(2)
Degree of freedom selection for 1-D behavior:

0 --
Use KEYOPT(3) options

1 --
1-D longitudinal spring-damper (UX degree of freedom)

2 --
1-D longitudinal spring-damper (UY degree of freedom)

3 --
1-D longitudinal spring-damper (UZ degree of freedom)

4 --
1-D Torsional spring-damper (ROTX degree of freedom)

5 --
1-D Torsional spring-damper (ROTY degree of freedom)

6 --
1-D Torsional spring-damper (ROTZ degree of freedom)

7 --
Pressure degree of freedom element

8 --
Temperature degree of freedom element

Note

KEYOPT(2) overrides KEYOPT(3)

KEYOPT(3)
Degree of freedom selection for 2-D and 3-D behavior:

0 --
3-D longitudinal spring-damper

1 --
3-D torsional spring-damper

2 --
2-D longitudinal spring-damper (2-D elements must lie in an X-Y plane)

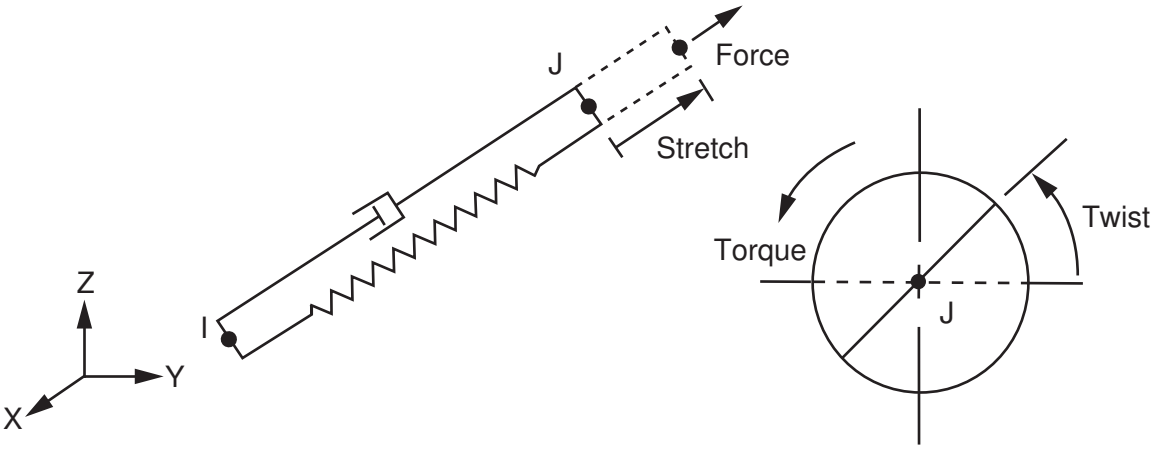
COMBIN14 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "COMBIN14 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "COMBIN14 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 COMBIN14 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 COMBIN14 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
FORC or TORQ	Spring force or moment	Y	Y
STRETCH or TWIST	Stretch of spring or twist of spring (radians)	Y	Y
RATE	Spring constant	Y	Y
VELOCITY	Velocity	-	Y
DAMPING FORCE or TORQUE	Damping force or moment (zero unless **ANTYPE**,TRANS and damping present)	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "COMBIN14 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "COMBIN14 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "COMBIN14 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 COMBIN14 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORC	SMISC	1
STRETCH	NMISC	1
VELOCITY	NMISC	2
DAMPING FORCE	NMISC	3

COMBIN14 Assumptions and Restrictions

- If KEYOPT(2) is zero, the length of the spring-damper element must not be zero, i.e., nodes I and J should not be coincident, since the node locations determine the spring orientation.
- The longitudinal spring element stiffness acts only along its length. The torsion spring element stiffness acts only about its length, as in a torsion bar.
- The element allows only a uniform stress in the spring.
- In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.
- Only the KEYOPT(2) = 0 option supports stress stiffening or large deflection. Also, if KEYOPT(3) = 1 (torsion) is used with large deflection, the coordinates will not be updated.
- The spring or the damping capability may be deleted from the element by setting K or CV equal to zero, respectively.
- If CV2 is not zero, the element is nonlinear and requires an iterative solution (KEYOPT(1) = 1).

The restrictions described below only apply if KEYOPT(2) is greater than zero.

- If KEYOPT(2) is greater than zero, the element has only one degree of freedom. This degree of freedom is specified in the nodal coordinate system and is the same for both nodes (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*). If the nodal coordinate systems are rotated relative to each other, the same degree of freedom may be in different directions (thereby giving possibly unexpected results). The element, however, assumes only a 1-D action. Nodes I and J, then, may be anywhere in space (preferably coincident).
- For noncoincident nodes and KEYOPT(2) = 1, 2, or 3, no moment effects are included. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.
- The element is defined such that a positive displacement of node J relative to node I tends to stretch the spring. If, for a given set of conditions, nodes I and J are interchanged, a positive displacement of node J relative to node I tends to compress the spring.

COMBIN14 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; CV1 and CV2 are not allowed.

- Only stress stiffening and large deflections are allowed.
- KEYOPT(2) = 7 or 8 is not allowed.
- The DAMP material property is not allowed.

ANSYS Professional

Thermal Analysis:

- KEYOPT(2) defaults to 8.
- KEYOPT(3) is not applicable.

DesignSpace

- KEYOPT(2) = 7 is not applicable.
- In Simulation, KEYOPT(2) = 7 *and* KEYOPT(2) = 8 are not applicable .

PIPE16

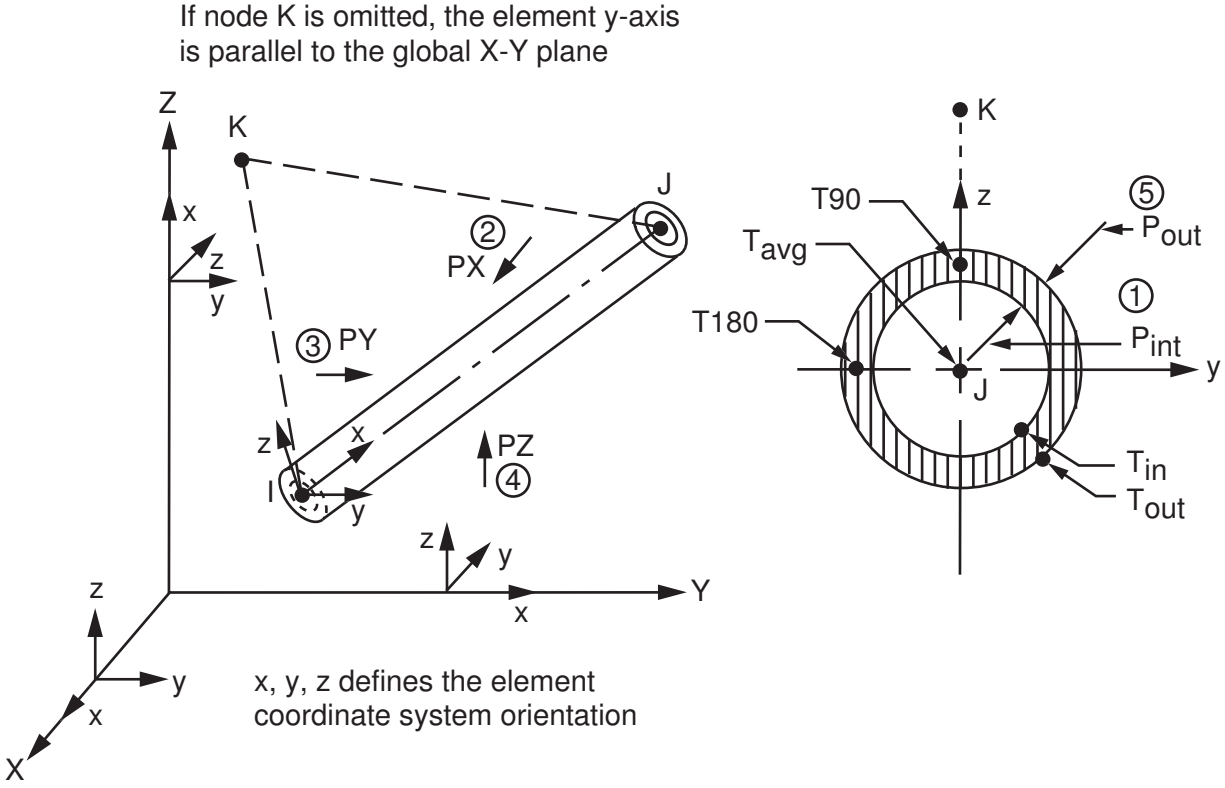
Elastic Straight Pipe

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

PIPE16 Element Description

PIPE16 is a uniaxial element with tension-compression, torsion, and bending capabilities. The element has six degrees of freedom at two nodes: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. This element is based on the 3-D beam element (BEAM4), and includes simplifications due to its symmetry and standard pipe geometry. See PIPE16 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PIPE18 for a curved pipe element. See PIPE17 for a pipe tee element. See PIPE20 for a plastic straight pipe element.

Figure 1 PIPE16 Geometry



PIPE16 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE16 Geometry"*. The element input data include two or three nodes, the pipe outer diameter and wall thickness, stress intensification and flexibility factors, internal fluid density, exterior insulation density and thickness, corrosion thickness allowance, insulation surface area, pipe wall mass, axial pipe stiffness, rotordynamic spin, and the isotropic material properties.

The element X-axis is oriented from node I toward node J. For the two-node option, the element Y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in *Figure 1, "PIPE16 Geometry"*. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element Y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation

about the element X-axis, use the third node option. The third node (K), if used, defines a plane (with I and J) containing the element X and Z axes (as shown). Input and output locations around the pipe circumference identified as being at 0° are located along the element Y-axis, and similarly 90° is along the element Z-axis.

The stress intensification factor (SIF) modifies the bending stress. Stress intensification factors may be input at end I (SIFI) and end J (SIFJ), if KEYOPT(2) = 0, or determined by the program using a tee-joint calculation if KEYOPT(2) = 1, 2, or 3. SIF values less than 1.0 are set equal to 1.0. The flexibility factor (FLEX) is divided into the cross-sectional moment of inertia to produce a modified moment of inertia for the bending stiffness calculation. FLEX defaults to 1.0 but may be input as any positive value.

The element mass is calculated from the pipe wall material, the external insulation, and the internal fluid. The insulation and the fluid contribute only to the element mass matrix. The corrosion thickness allowance contributes only to the stress calculations. A positive wall mass real constant overrides the pipe wall mass calculation. A nonzero insulation area real constant overrides the insulation surface area calculation (from the pipe outer diameter and length). A nonzero stiffness real constant overrides the calculated axial pipe stiffness.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE16 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. The normal component or the projected full pressure may be used (KEYOPT(5)). Tapered pressures are not recognized. Only constant pressures are supported for this element. See PIPE16 in the *Theory Reference for ANSYS and ANSYS Workbench* for more information.

Temperatures may be input as element body loads at the nodes. Temperatures may have wall gradients or diametral gradients (KEYOPT(1)). The average wall temperature at $\theta = 0°$ is computed as 2 * TAVG - T(180) and the average wall temperature at $\theta = -90°$ is computed as 2 * TAVG - T(90). The element temperatures are assumed to be linear along the length. The first temperature at node I (TOUT(I) or TAVG(I)) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other pattern of input temperatures, unspecified temperatures default to TUNIF.

For piping analyses, the PIPE module of PREP7 may be used to generate the input for this element. KEYOPT(4) is used to identify the element type for output labeling and for postprocessing operations.

KEYOPT(7) is used to compute an unsymmetric gyroscopic damping matrix (often used for rotordynamic analyses). The rotational frequency is input with the SPIN real constant (radians/time, positive in the positive element x direction).

A summary of the element input is given in *PIPE16 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PIPE16 Input Summary

Nodes

I, J, K (K, the orientation node, is optional)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

OD, TKWALL, SIFI, SIFJ, FLEX, DENSFL,
DENSIN, TKIN, TKCORR, AREAIN, MWALL, STIFF,
SPIN
See *Table 1, "PIPE16 Real Constants"* for a description of the real constants

Material Properties
EX, ALPX (or CTEX *or* THSX),

PRXY (or NUXY), DENS, GXY, DAMP

Surface Loads

Pressures --
1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --

TOUT(I), TIN(I), TOUT(J), TIN(J) if KEYOPT (1) = 0, or
TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J) if KEYOPT (1) = 1

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(1)
Temperatures represent:

0 --
The through-wall gradient

1 --
The diametral gradient

KEYOPT(2)
Stress intensification factors:

0 --
Stress intensity factors from SIFI and SIFJ

1 --
Stress intensity factors at node I from tee joint calculation

2 --
Stress intensity factors at node J from tee joint calculation

3 --
Stress intensity factors at both nodes from tee joint calculation

KEYOPT(4)
Element identification (for output and postprocessing):

0 --
Straight pipe

1 --
Valve

2 --
Reducer

3 --
Flange

4 --
Expansion joint

5 --
Mitered bend

6 --
Tee branch

KEYOPT(5)
PX, PY, and PZ transverse pressures:

0 --
Use only the normal component of pressure

1 --
Use the full pressure (normal and shear components)

KEYOPT(6)
Member force and moment output:

0 --
Do not print member forces or moments

2 --
Print member forces and moments in the element coordinate system

KEYOPT(7)
Gyroscopic damping matrix:

0 --
No gyroscopic damping matrix

1 --
Compute gyroscopic damping matrix. Real constant SPIN must be greater than zero. DENSFL and DENSIN must be zero.

Note

The real constant MWALL is not used to compute the gyroscopic damping matrix.

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

Table 1 PIPE16 Real Constants

No.	Name	Description
1	OD	Pipe outer diameter
2	TKWALL	Wall thickness
3	SIFI	Stress intensification factor (node I)
4	SIFJ	Stress intensification factor (node J)
5	FLEX	Flexibility factor
6	DENSFL	Internal fluid density
7	DENSIN	Exterior insulation density
8	TKIN	Insulation thickness
9	TKCORR	Corrosion thickness allowance
10	AREAIN	Insulation surface area (replaces program-calculated value)
11	MWALL	Pipe wall mass (replaces program-calculated value)
12	STIFF	Axial pipe stiffness (replaces program-calculated value)
13	SPIN	Rotordynamic spin (required if KEYOPT(7) = 1)

PIPE16 Output Data

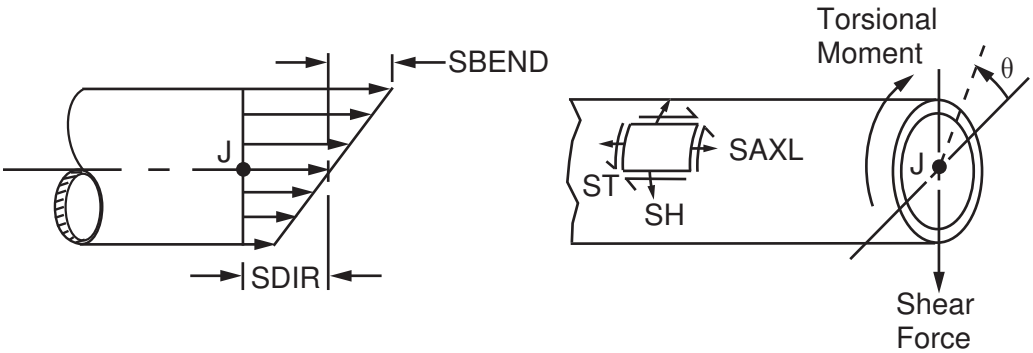
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "PIPE16 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PIPE16 Stress Output"*.

The direct stress (SAXL) includes the internal pressure (closed end) effect. The direct stress does not include the axial component of the transverse thermal stress (STH). The principal stresses and the stress intensity include the shear force stress component, and are based on the stresses at the two extreme points on opposite sides of the neutral axis. These quantities are computed at the outer surface and might not occur at the same location around the pipe circumference. Angles listed in the output are measured as shown (θ) in *Figure 2, "PIPE16 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PIPE16 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 PIPE16 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*6*
CORAL	Corrosion thickness allowance	*1*	*1*
TEMP	TOUT(I), TIN(I), TOUT(J), TIN(J)	*2*	*2*
TEMP	TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)	*3*	*3*
PRES	PINT, PX, PY, PZ, POUT	Y	Y
SFACTI, SFACTJ	Stress intensification factors at nodes I and J	Y	Y
STH	Stress due to maximum thermal gradient through the wall thickness	Y	Y
SPR2	Hoop pressure stress for code calculations	-	Y
SMI, SMJ	Moment stress at nodes I and J for code calculations	-	Y
SDIR	Direct (axial) stress	-	Y
SBEND	Maximum bending stress at outer surface	-	Y
ST	Shear stress at outer surface due to torsion	-	Y
SSF	Shear stress due to shear force	-	Y
S:(1MX, 3MN, INTMX, EQVMX)	Maximum principal stress, minimum principal stress, maximum stress intensity, maximum equivalent stress (all at the outer surface)	Y	Y
S:(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*4*	*4*
S:(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*4*	*4*
EPEL:(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*4*	*4*
EPTH:(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*4*	*4*
MFOR:(X, Y, Z)	Member forces for nodes I and J (in the element coordinate system)	*5*	Y
MMOM:(X, Y, Z)	Member moments for nodes I and J (in the element coordinate system)	*5*	Y

1. If the value is greater than 0.
2. If KEYOPT(1) = 0
3. If KEYOPT(1) = 1
4. The item repeats at 0°, 45°, 90°, 135°, 180°, 225°, 270°, 315° at node I, then at node J, all at the outer surface.
5. If KEYOPT(6) = 2
6. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PIPE16 Item and Sequence Numbers (Node I)"* through *Table 5, "PIPE16 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "PIPE16 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J
sequence number for data at nodes I and J

Table 3 PIPE16 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	5	9	13	17	21	25	29
EPTHRAD	LEPTH	-	2	6	10	14	18	22	26	30
EPTHH	LEPTH	-	3	7	11	15	19	23	27	31
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	13	-	-	-	-	-	-	-	-
ST	SMISC	14	-	-	-	-	-	-	-	-
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	90	-	-	-	-	-	-	-	-
SSF	NMISC	91	-	-	-	-	-	-	-	-
TOUT	LBFE	-	4	-	1	-	2	-	3	-
TIN	LBFE	-	8	-	5	-	6	-	7	-

Table 4 PIPE16 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64
EPTHAXL	LEPTH	-	33	37	41	45	49	53	57	61
EPTHRAD	LEPTH	-	34	38	42	46	50	54	58	62
EPTHH	LEPTH	-	35	39	43	47	51	55	59	63
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	15	-	-	-	-	-	-	-	-
ST	SMISC	16	-	-	-	-	-	-	-	-
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	92	-	-	-	-	-	-	-	-
SSF	NMISC	93	-	-	-	-	-	-	-	-
TOUT	LBFE	-	12	-	9	-	10	-	11	-
TIN	LBFE	-	16	-	13	-	14	-	15	-

Table 5 PIPE16 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
STH	SMISC	17
PINT	SMISC	18
PX	SMISC	19
PY	SMISC	20
PZ	SMISC	21
POUT	SMISC	22
SFACTI	NMISC	81
SFACTJ	NMISC	82

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SPR2	NMISC	83
SMI	NMISC	84
SMJ	NMISC	85
S1MX	NMISC	86
S3MN	NMISC	87
SINTMX	NMISC	88
SEQVMX	NMISC	89

PIPE16 Assumptions and Restrictions

- The pipe must not have a zero length or wall thickness. In addition, the OD must not be less than or equal to zero, the ID must not be less than zero, and the corrosion thickness allowance must be less than the wall thickness.
- The element temperatures are assumed to vary linearly along the length.
- The element may be used for both thin and thick-walled situations; however, some of the stress calculations are based on thin-wall theory.
- The pipe element is assumed to have "closed ends" so that the axial pressure effect is included.
- Shear deflection capability is also included in the element formulation.
- Eigenvalues calculated in a gyroscopic modal analysis can be very sensitive to changes in the initial shift value, leading to potential error in either the real or imaginary (or both) parts of the eigenvalues.

PIPE16 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The SPIN real constant (R13) is not available.
- The DAMP material property is not allowed.
- The only special features allowed are stress stiffening and large deflections.
- KEYOPT(7) (gyroscopic damping) is not allowed.

PIPE17

Elastic Pipe Tee

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

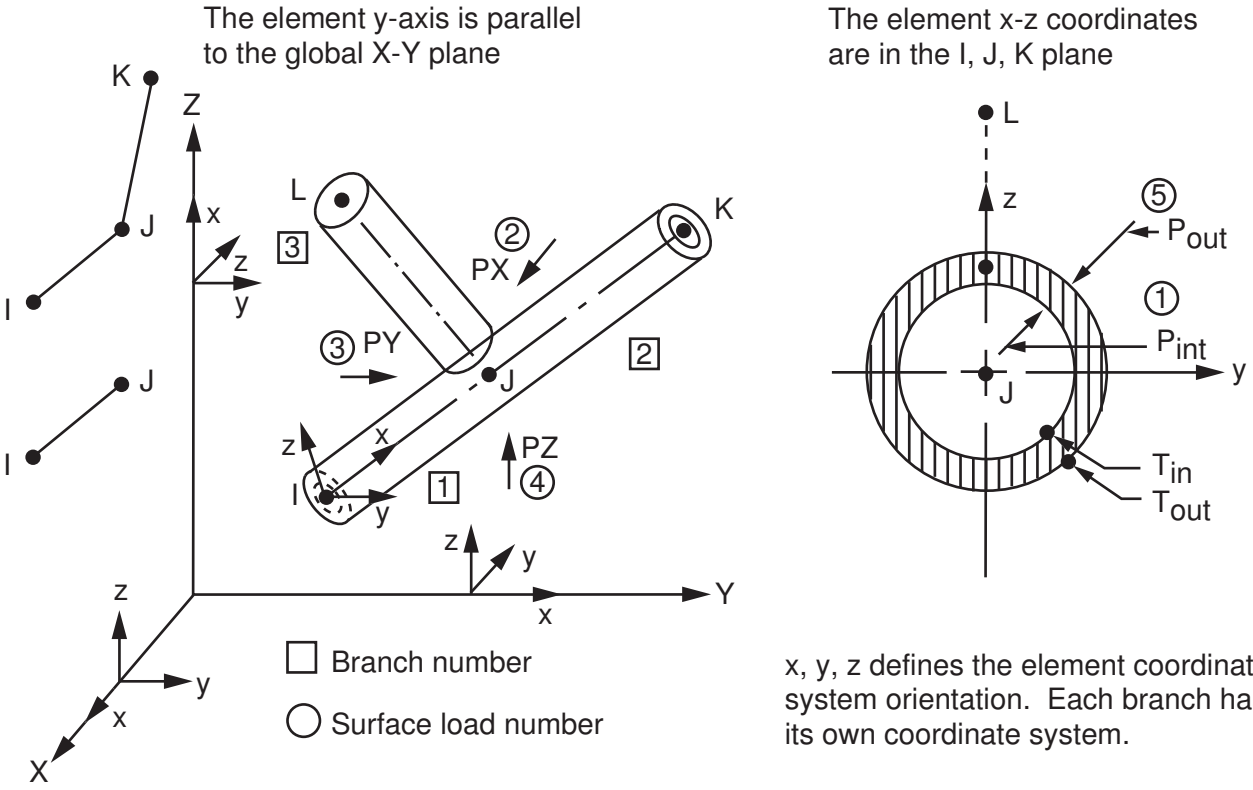
PIPE17 Element Description

PIPE17 is a combination of three uniaxial elastic straight pipe elements (PIPE16) arranged in a "tee" configuration, with tension-compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes.

Options are available to include tee-joint flexibility and stress intensification factors and to print member forces. The element can account for insulation, contained fluid, and a corrosion allowance. See PIPE17 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

The I and J nomenclature used in the description of this element refers to the first and second end of each branch of the element, i.e., I-J for branch 1, J-K for branch 2, and J-L for branch 3.

Figure 1 PIPE17 Geometry



PIPE17 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE17 Geometry"*. The element input data include four nodes, the branch outer diameters, wall thicknesses, material numbers, flexibility factors, stress intensification factors, internal fluid densities, exterior insulation densities and thicknesses, corrosion thickness allowance, and the isotropic material properties. The real constant material number, if supplied, overrides the element material property number applied with the **MAT** command, and defaults to the element material property number. The element degenerates to two branches if three nodes are input, and to one pipe

element if only two nodes are input. The real constants (except DFL, DIN, and TKIN) for the other branches default to those of the first branch if not input.

The bending stiffness of this element is similar to that of BEAM4 except that it is modified by the flexibility factor. Each branch has its own element coordinate system, with its origin at the first node of the branch and the element X-axis along the branch axis. The orientation of the branch Y-axis is automatically calculated to be parallel to the global X-Y plane (see *Figure 1, "PIPE17 Geometry"*). For the case where the branch is parallel to the global Z axis (or within a 0.01 percent slope of it), the branch Y-axis is oriented parallel to the global Y axis. Input and output locations around the pipe circumference identified as being at 0° are located along the branch Y-axis, and similarly 90° is along the branch Z-axis. The flexibility factor (FLEX) is divided into the cross-sectional moment of inertia to produce a modified moment of inertia for the stiffness calculation. FLEX defaults to 1.0 but may be input as any positive value. The internal fluid and external insulation constants are used only to determine the added mass effects for these components.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE17 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. Tapered pressures are not recognized. Only constant pressures are supported for this element. See PIPE17 in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

Temperatures may be input as element body loads at the nodes. Outer and inner wall temperatures may be specified for each branch. Temperatures are assumed to be uniform along each branch. The first temperature for branch 1 (TOUT1) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If both temperatures at branch 1 are input, and all temperatures at branches 2 and 3 are unspecified, they default to the corresponding branch 1 temperatures. For any other input pattern, unspecified temperatures default to TUNIF.

Use the **BETAD** command to supply the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to supply the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the branch, it is used instead of either the global or element value.

The KEYOPT(2) options for stress intensification factors are discussed in *PIPE16 Input Data*.

A summary of the element input is given in *PIPE17 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PIPE17 Input Summary

Nodes

I, J, K, L for three branches (I-J, J-K, J-L), or
I, J, K for two branches (I-J, J-K), or
I, J for one branch (I-J)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

OD1, TK1, MAT1, FLEX1, SIF1I, SIF1J,
OD2, TK2, MAT2, FLEX2, SIF2J, SIF2K,
OD3, TK3, MAT3, FLEX3, SIF3J, SIF3L,
DFL1, DIN1, TKIN1, DFL2, DIN2, TKIN2,
DFL3, DIN3, TKIN3, TKCORR
See *Table 1, "PIPE17 Real Constants"* for a description of the real constants.

Material Properties

EX, ALPX (or CTEX *or* THSX), PRXY (or NUXY), DENS, GXY, DAMP
REFT

Supply DAMP only once for the element (use **MAT** command to assign material property set). REFT may be supplied once for the element, or may be assigned on a per branch basis. See the discussion in *PIPE17 Input Data* for more details.

Surface Loads

Pressures --
1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --
TOUT1, TIN1, TOUT2, TIN2, TOUT3, TIN3 (outer and inner for each branch)

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(2)
Stress intensification factors:

0 --
Stress intensity factors from SIF real constants

1 --
Tee stress intensity factors at first node of each branch from tee joint calculation

2 --
Tee stress intensity factors at second node of each branch from tee joint calculation

3 --
Tee stress intensity factors at both nodes of each branch from tee joint calculation

KEYOPT(6)
Member force and moment output:

0 --
No printout of member forces or moments

2 --
Print member forces and moments in the element coordinate system

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

Table 1 PIPE17 Real Constants

No.	Name	Description
1	OD1	Pipe outer diameter for branch 1
2	TK1	Thickness for branch 1
3	MAT1	Material number for branch 1
4	FLEX1	Flexibility factor for branch 1
5	SIF1I	Stress intensification factor for branch 1, node I
6	SIF1J	Stress intensification factor for branch 1, node J
7	OD2	Pipe outer diameter for branch 2
8	TK2	Thickness for branch 2
9	MAT2	Material number for branch 2
10	FLEX2	Flexibility factor for branch 2
11	SIF2J	Stress intensification factor for branch 2, node J
12	SIF2K	Stress intensification factor for branch 2, node K
13	OD3	Pipe outer diameter for branch 3
14	TK3	Thickness for branch 3
15	MAT3	Material number for branch 3
16	FLEX3	Flexibility factor for branch 3
17	SIF3J	Stress intensification factor for branch 3, node J
18	SIF3L	Stress intensification factor for branch 3, node L
19	DFL1	Internal fluid densities
20	DIN1	Exterior insulation densities
21	TKIN1	Insulation thickness for branch 1
22	DFL2	Internal fluid densities
23	DIN2	Exterior insulation densities
24	TKIN2	Insulation thickness for branch 2
25	DFL3	Internal fluid densities
26	DIN3	Exterior insulation densities
27	TKIN3	Insulation thickness for branch 3
28	TKCORR	Corrosion thickness allowance

PIPE17 Output Data

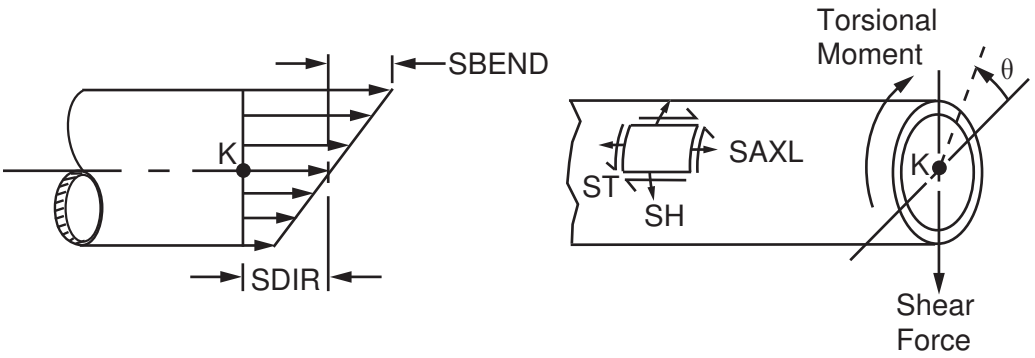
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "PIPE17 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PIPE17 Stress Output"*.

The direct stress includes the internal pressure (closed end) effect. The direct stress does not include the axial component of the transverse thermal stress. Also printed for each end of each branch are the maximum and minimum principal stresses and the stress intensity. These quantities are computed at the outer surface and may not occur at the same location around the pipe circumference. The effect of the corrosion allowance thickness is also included as described in *PIPE16 Input Data*. The principal stresses and the stress intensity include the shear force stress component. The output stresses and the stress intensification factors are calculated as shown in *PIPE16 Input Data*. Angles listed in the output are measured as shown (θ) in *Figure 2, "PIPE17 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PIPE17 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 PIPE17 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*4*
TEMP	TOUT1, TIN1, TOUT2, TIN2, TOUT3, TIN3 (outer and inner for each branch)	Y	Y
PRES	PINT, PX, PY, PZ, POUT	Y	Y
MFOR(X, Y, Z)	Member forces at the ends of each branch (in the branch coordinate system)	*1*	Y
MMOM(X, Y, Z)	Member moments at the ends of each branch (in the branch coordinate system)	*1*	Y
SFACTI, SFACTJ	Stress intensification factors	*2*	*2*
STH	Stress due to maximum thermal gradient through the wall thickness	*2*	*2*
SPR2	Hoop pressure stress for code calculations	-	*2*

Name	Definition	O	R
SMI, SMJ	Moment stress at nodes I and J for code calculations	-	*2*
SDIR	Direct (axial) stress	-	*2*
SBEND	Maximum bending stress at outer surface	-	*2*
ST	Shear stress at outer surface due to torsion	-	*2*
SSF	Shear stress due to shear force	-	*2*
S(1MX, 3MN, INT-MX, EQVMX)	Maximum principal stress, minimum principal stress, maximum stress intensity, maximum equivalent stress (all at the outer surface)	*2*	*2*
S(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*3*	*3*
S(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*3*	*3*
EPEL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*3*	*3*
EPTH(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*3*	*3*

1. Only if KEYOPT(6) = 2
2. The item repeats for each branch
3. The item repeats at 0°, 45°, 90°, 135°, 180°, 225°, 270°, 315° at the ends of each branch (all at the outer surface)
4. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PIPE17 Item and Sequence Numbers (Branch 1, Node I)"* through *Table 12, "PIPE17 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "PIPE17 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K
sequence number for data at nodes I,J, and K

Table 3 PIPE17 Item and Sequence Numbers (Branch 1, Node I)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	5	9	13	17	21	25	29
EPTHRAD	LEPTH	-	2	6	10	14	18	22	26	30
EPTHH	LEPTH	-	3	7	11	15	19	23	27	31
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	268	-	-	-	-	-	-	-	-
SSF	NMISC	269	-	-	-	-	-	-	-	-
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	37	-	-	-	-	-	-	-	-
ST	SMISC	38	-	-	-	-	-	-	-	-

Table 4 PIPE17 Item and Sequence Numbers (Branch 1, Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64
EPTHAXL	LEPTH	-	33	37	41	45	49	53	57	61
EPTHRAD	LEPTH	-	34	38	42	46	50	54	58	62
EPTHH	LEPTH	-	35	39	43	47	51	55	59	63
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	270	-	-	-	-	-	-	-	-
SSF	NMISC	271	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	39	-	-	-	-	-	-	-	-
ST	SMISC	40	-	-	-	-	-	-	-	-

Table 5 PIPE17 Item and Sequence Numbers (Branch 1)

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	Circumferential Location			
			0°	90°	180°	270°
SFACTI	NMISC	241	-	-	-	-
SFACTJ	NMISC	242	-	-	-	-
SPR2	NMISC	243	-	-	-	-
SMI	NMISC	244	-	-	-	-
SMJ	NMISC	245	-	-	-	-
S1MX	NMISC	256	-	-	-	-
S3MN	NMISC	257	-	-	-	-
SINTMX	NMISC	258	-	-	-	-
SEQVMX	NMISC	259	-	-	-	-
STH	SMISC	41	-	-	-	-
TOUT	LBFE	-	4	1	2	3
TIN	LBFE	-	8	5	6	7

Table 6 PIPE17 Item and Sequence Numbers (Branch 2, Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	65	69	73	77	81	85	89	93
SRAD	LS	-	66	70	74	78	82	86	90	94
SH	LS	-	67	71	75	79	83	87	91	95
SXH	LS	-	68	72	76	80	84	88	92	96
EPELAXL	LEPEL	-	65	69	73	77	81	85	89	93
EPELRAD	LEPEL	-	66	70	74	78	82	86	90	94
EPELH	LEPEL	-	67	71	75	79	83	87	91	95
EPELXH	LEPEL	-	68	72	76	80	84	88	92	96
EPTHAXL	LEPTH	-	65	69	73	77	81	85	89	93
EPTHRAD	LEPTH	-	66	70	74	78	82	86	90	94
EPTHH	LEPTH	-	67	71	75	79	83	87	91	95

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
S1	NMISC	-	81	86	91	96	101	106	111	116
S3	NMISC	-	83	88	93	98	103	108	113	118
SINT	NMISC	-	84	89	94	99	104	109	114	119
SEQV	NMISC	-	85	90	95	100	105	110	115	120
SBEND	NMISC	272	-	-	-	-	-	-	-	-
SSF	NMISC	273	-	-	-	-	-	-	-	-
MFORX	SMISC	13	-	-	-	-	-	-	-	-
MFORY	SMISC	14	-	-	-	-	-	-	-	-
MFORZ	SMISC	15	-	-	-	-	-	-	-	-
MMOMX	SMISC	16	-	-	-	-	-	-	-	-
MMOMY	SMISC	17	-	-	-	-	-	-	-	-
MMOMZ	SMISC	18	-	-	-	-	-	-	-	-
SDIR	SMISC	42	-	-	-	-	-	-	-	-
ST	SMISC	43	-	-	-	-	-	-	-	-

Table 7 PIPE17 Item and Sequence Numbers (Branch 2, Node K)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	97	101	105	109	113	117	121	125
SRAD	LS	-	98	102	106	110	114	118	122	126
SH	LS	-	99	103	107	111	115	119	123	127
SXH	LS	-	100	104	108	112	116	120	124	128
EPELAXL	LEPEL	-	97	101	105	109	113	117	121	125
EPELRAD	LEPEL	-	98	102	106	110	114	118	122	126
EPELH	LEPEL	-	99	103	107	111	115	119	123	127
EPELXH	LEPEL	-	100	104	108	112	116	120	124	128
EPTHAXL	LEPTH	-	97	101	105	109	113	117	121	125
EPTHRAD	LEPTH	-	98	102	106	110	114	118	122	126
EPTHH	LEPTH	-	99	103	107	111	115	119	123	127
S1	NMISC	-	121	126	131	136	141	146	151	156
S3	NMISC	-	123	128	133	138	143	148	153	158
SINT	NMISC	-	124	129	134	139	144	149	154	159
SEQV	NMISC	-	125	130	135	140	145	150	155	160
SBEND	NMISC	274	-	-	-	-	-	-	-	-
SSF	NMISC	275	-	-	-	-	-	-	-	-
MFORX	SMISC	19	-	-	-	-	-	-	-	-
MFORY	SMISC	20	-	-	-	-	-	-	-	-
MFORZ	SMISC	21	-	-	-	-	-	-	-	-
MMOMX	SMISC	22	-	-	-	-	-	-	-	-
MMOMY	SMISC	23	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
MMOMZ	SMISC	24	-	-	-	-	-	-	-	-
SDIR	SMISC	44	-	-	-	-	-	-	-	-
ST	SMISC	45	-	-	-	-	-	-	-	-

Table 8 PIPE17 Item and Sequence Numbers (Branch 2)

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	Circumferential Location			
			0°	90°	180°	270°
SFACTI	NMISC	246	-	-	-	-
SFACTJ	NMISC	247	-	-	-	-
SPR2	NMISC	248	-	-	-	-
SMI	NMISC	249	-	-	-	-
SMJ	NMISC	250	-	-	-	-
S1MX	NMISC	260	-	-	-	-
S3MN	NMISC	261	-	-	-	-
SINTMX	NMISC	262	-	-	-	-
SEQVMX	NMISC	263	-	-	-	-
STH	SMISC	46	-	-	-	-
TOUT	LBFE	-	12	9	10	11
TIN	LBFE	-	16	13	14	15

Table 9 PIPE17 Item and Sequence Numbers (Branch 3, Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	129	133	137	141	145	149	153	157
SRAD	LS	-	130	134	138	142	146	150	154	158
SH	LS	-	131	135	139	143	147	151	155	159
SXH	LS	-	132	136	140	144	148	152	156	160
EPELAXL	LEPEL	-	129	133	137	141	145	149	153	157
EPELRAD	LEPEL	-	130	134	138	142	146	150	154	158
EPELH	LEPEL	-	131	135	139	143	147	151	155	159
EPELXH	LEPEL	-	132	136	140	144	148	152	156	160
EPTHAXL	LEPTH	-	129	133	137	141	145	149	153	157
EPTHRAD	LEPTH	-	130	134	138	142	146	150	154	158
EPTHH	LEPTH	-	131	135	139	143	147	151	155	159
S1	NMISC	-	161	166	171	176	181	186	191	196
S3	NMISC	-	163	168	173	178	183	188	193	198
SINT	NMISC	-	164	169	174	179	184	189	194	199
SEQV	NMISC	-	165	170	175	180	185	190	195	200
SBEND	NMISC	276	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SSF	NMISC	277	-	-	-	-	-	-	-	-
MFORX	SMISC	25	-	-	-	-	-	-	-	-
MFORY	SMISC	26	-	-	-	-	-	-	-	-
MFORZ	SMISC	27	-	-	-	-	-	-	-	-
MMOMX	SMISC	28	-	-	-	-	-	-	-	-
MMOMY	SMISC	29	-	-	-	-	-	-	-	-
MMOMZ	SMISC	30	-	-	-	-	-	-	-	-
SDIR	SMISC	47	-	-	-	-	-	-	-	-
ST	SMISC	48	-	-	-	-	-	-	-	-

Table 10 PIPE17 Item and Sequence Numbers (Branch 3, Node L)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	161	165	169	173	177	181	185	189
SRAD	LS	-	162	166	170	174	178	182	186	190
SH	LS	-	163	167	171	175	179	183	187	191
SXH	LS	-	164	168	172	176	180	184	188	192
EPELAXL	LEPEL	-	161	165	169	173	177	181	185	189
EPELRAD	LEPEL	-	162	166	170	174	178	182	186	190
EPELH	LEPEL	-	163	167	171	175	179	183	187	191
EPELXH	LEPEL	-	164	168	172	176	180	184	188	192
EPTHAXL	LEPTH	-	161	165	169	173	177	181	185	189
EPTHRAD	LEPTH	-	162	166	170	174	178	182	186	190
EPTHH	LEPTH	-	163	167	171	175	179	183	187	191
S1	NMISC	-	201	206	211	216	221	226	231	236
S3	NMISC	-	203	208	213	218	223	228	233	238
SINT	NMISC	-	204	209	214	219	224	229	234	239
SEQV	NMISC	-	205	210	215	220	225	230	235	240
SBEND	NMISC	278	-	-	-	-	-	-	-	-
SSF	NMISC	279	-	-	-	-	-	-	-	-
MFORX	SMISC	31	-	-	-	-	-	-	-	-
MFORY	SMISC	32	-	-	-	-	-	-	-	-
MFORZ	SMISC	33	-	-	-	-	-	-	-	-
MMOMX	SMISC	34	-	-	-	-	-	-	-	-
MMOMY	SMISC	35	-	-	-	-	-	-	-	-
MMOMZ	SMISC	36	-	-	-	-	-	-	-	-
SDIR	SMISC	49	-	-	-	-	-	-	-	-
ST	SMISC	50	-	-	-	-	-	-	-	-

Table 11 PIPE17 Item and Sequence Numbers (Branch 3)

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	Circumferential Location			
			0°	90°	180°	270°
SFACTI	NMISC	251	-	-	-	-
SFACTJ	NMISC	252	-	-	-	-
SPR2	NMISC	253	-	-	-	-
SMI	NMISC	254	-	-	-	-
SMJ	NMISC	255	-	-	-	-
S1MX	NMISC	264	-	-	-	-
S3MN	NMISC	265	-	-	-	-
SINTMX	NMISC	266	-	-	-	-
SEQVMX	NMISC	267	-	-	-	-
STH	SMISC	51	-	-	-	-
TOUT	LBFE	-	20	17	18	19
TIN	LBFE	-	24	21	22	23

Table 12 PIPE17 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PINT	SMISC	52
PX	SMISC	53
PY	SMISC	54
PZ	SMISC	55
POUT	SMISC	56

PIPE17 Assumptions and Restrictions

- No branch can have a zero length or wall thickness (although branches may be deleted).
- The OD must not be less than or equal to zero, the ID must not be less than zero, and the corrosion thickness allowance must be less than the wall thickness.
- The element may be used for both thin and thick-walled situations; however, some of the stress calculations are based on thin-wall theory.
- The branches are assumed to have "closed ends" so that the axial pressure effect is included.
- There is no restriction on the angles of intersection of the branches.
- Shear deflection capability is also included in the element formulation.

PIPE17 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.

- The only special features allowed are stress stiffening and large deflections.

PIPE18

Elastic Curved Pipe

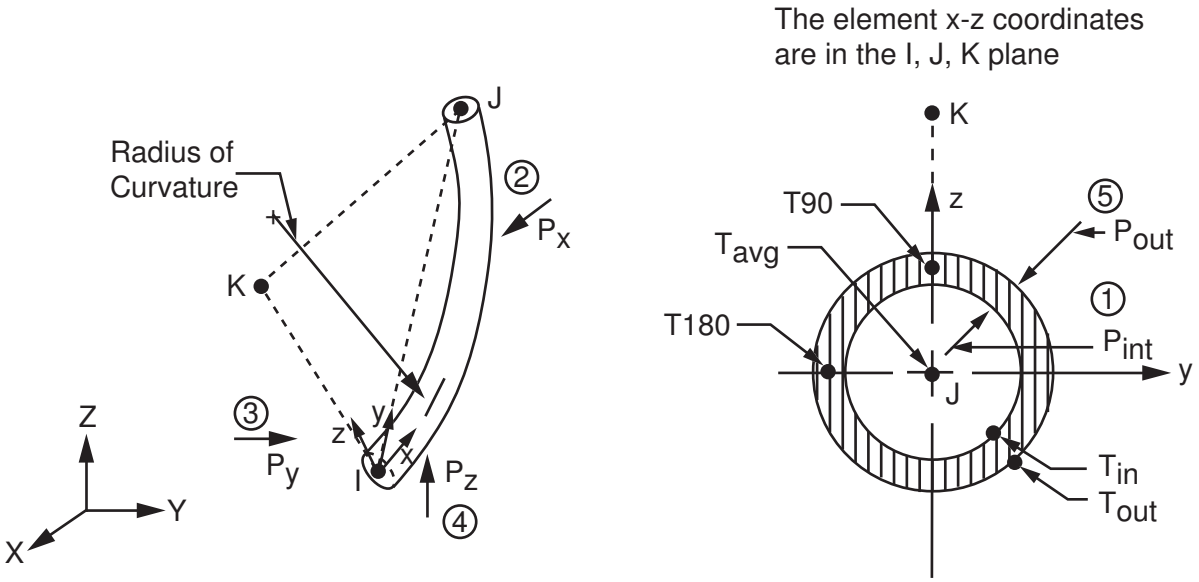
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

PIPE18 Element Description

PIPE18, also known as an elbow element, is a circularly uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes.

Options are available to include various flexibility and stress intensification factors in the formulation. The element can account for insulation, contained fluid, and a corrosion allowance. See PIPE18 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PIPE16 for a straight pipe element. See PIPE17 for a pipe tee element. See PIPE60 for a plastic curved pipe.

Figure 1 PIPE18 Geometry



PIPE18 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE18 Geometry"*. The element input data include three nodes, the pipe outer diameter, wall thickness, radius of curvature, optional stress intensification and flexibility factors, internal fluid density, exterior insulation density and thickness, corrosion thickness allowance, and the isotropic material properties. The internal fluid and external insulation constants are used only to determine the added mass effects for these components.

Although the curved pipe element has only two endpoints (nodes I and J), the third node (K) is required to define the plane in which the element lies. This node must lie in the plane of the curved pipe and on the center-of-curvature side of line I-J. A node point belonging to another element (such as the other node of a connecting straight pipe element) may be used. Input and output locations around the pipe circumference identified as being at 0° are located along the element y-axis, and similarly 90° is along the element z-axis.

Only the lumped mass matrix is available.

The flexibility and stress intensification factors included in the element are calculated as follows:

ANSYS Flexibility Factor = $1.65/(h(1 + PrX_k/tE))$ or 1.0 (whichever is greater) (used if KEYOPT(3) = 0 or 1 and FLXI not input)

Karman Flexibility Factor = $(10 + 12h^2)/(1 + 12h^2)$ (used if KEYOPT(3) = 2 and FLXI not input)

User Defined Flexibility Factors = FLXI (in-plane) and FLXO (out-of-plane) (may be input as any positive value)

FLXO defaults to FLXI for all cases.

Stress Intensification Factor = $0.9/h^{2/3}$ or 1.0 (whichever is greater) (used for SIFI or SIFJ if factor not input or if input less than 1.0 (must be positive))

where:

$h = tR/r^2$
t = thickness
R = radius of curvature
r = average radius
E = modulus of elasticity
$X_k = 6\,(r/t)^{4/3}\,(R/r)^{1/3}$ if KEYOPT(3) = 1 and $R/r \geq 1.7$, otherwise $X_k = 0$
$P = P_i - P_o$ if $P_i - P_o > 0$, otherwise P = 0, P_i = internal pressure, P_o = external pressure

KEYOPT(3) = 1 should not be used if the included angle of the complete elbow is less than $360/(\pi(R/r))°$.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE18 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. Note that when using KEYOPT(8) = 1, the pressure load will be acting on only the wall of the elbow element so that the total pressure load will not be self-equilibrating. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. Tapered pressures are not recognized. Only constant pressures are supported for this element. See the *Theory Reference for ANSYS and ANSYS Workbench* for details.

Temperatures may be input as element body loads at the nodes. Temperatures may have wall gradients or diametral gradients (KEYOPT(1)). The average wall temperature at $\theta = 0°$ is computed as 2 * TAVG - T(180) and the average wall temperature at $\theta = -90°$ is computed as 2 * TAVG - T(90). The element temperatures are assumed to be linear along the length. The first temperature at node I (TOUT(I) or TAVG(I)) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other pattern of input temperatures, unspecified temperatures default to TUNIF.

For piping analyses, the PIPE module of PREP7 may be used to generate the input for this element.

A summary of the element input is given below. A general description of element input is given in *Section 2.1: Element Input*.

PIPE18 Input Summary

Nodes
I, J, K - where node K is in the plane of the elbow, on the center of curvature side of line I-J

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

OD, TKWALL, RADCUR, SIFI, SIFJ, FLXI,
DENSFL, DENSIN, TKIN, TKCORR, (Blank), FLXO
See *Table 1, "PIPE18 Real Constants"* for a description of the real constants

Material Properties
EX, ALPX (or CTEX *or* THSX), PRXY (or NUXY), DENS, GXY, DAMP

Surface Loads

Pressures --
1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --

TOUT(I), TIN(I), TOUT(J), TIN(J) if KEYOPT (1) = 0, or
TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J) if KEYOPT (1) = 1

Special Features

Large deflection
Birth and death

KEYOPT(1)
Temperatures represent:

0 --
The through-wall gradient

1 --
The diametral gradient

KEYOPT(3)
Flex factor (if FLEX is not specified):

0 --
Use ANSYS flexibility factor (without pressure term)

1 --
Use ANSYS flexibility factor (with pressure term)

2 --
Use KARMAN flexibility factor

KEYOPT(6)
Member force and moment output:

0 --
Do not print member forces or moments

2 --
Print member forces and moments in the element coordinate system

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

Table 1 PIPE18 Real Constants

No.	Name	Description
1	OD	Pipe outer diameter
2	TKWALL	Wall thickness
3	RADCUR	Radius of curvature
4	SIFI	Stress intensification factor (node I)
5	SIFJ	Stress intensification factor (node J)
6	FLXI	Flexibility factor (in-plane)
7	DENSFL	Internal fluid density
8	DENSIN	Exterior insulation density
9	TKIN	Insulation thickness
10	TKCORR	Corrosion thickness allowance
11	(Blank)	--
12	FLXO	Flexibility factor (out-of-plane). FLXO defaults to FLXI in all cases.

PIPE18 Output Data

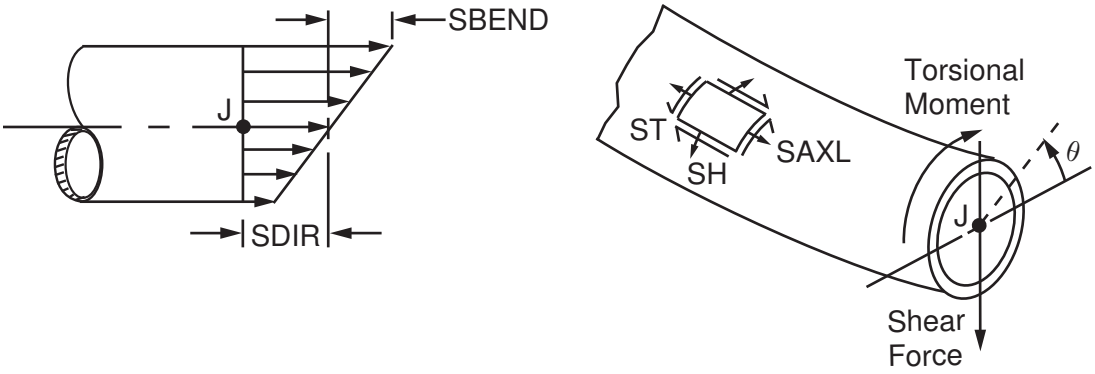
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "PIPE18 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PIPE18 Stress Output"*.

The stresses are computed with the outer diameter of the pipe reduced by twice the corrosion thickness allowance. The direct stress includes the internal pressure (closed end) effect. Also printed for each end are the maximum and minimum principal stresses and the stress intensity. These quantities are computed at the outer surface and may not occur at the same location around the pipe circumference. Some of these stresses are shown in *Figure 2, "PIPE18 Stress Output"*. The direct stress does not include the axial component of the transverse thermal stress. The principal stresses and the stress intensity include the shear force stress component. Angles listed in the output are measured (θ) as shown in *Figure 2, "PIPE18 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PIPE18 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 PIPE18 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*6*
CORAL	Corrosion thickness allowance	*1*	*1*
TEMP	TOUT(I), TIN(I), TOUT(J), TIN(J)	*2*	*2*
TEMP	TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)	*3*	*3*
PRES	PINT, PX, PY, PZ, POUT	Y	Y
FFACT	Element flexibility factor	-	Y
MFOR(X, Y, Z)	Member forces for nodes I and J (in the element coordinate system)	*4*	Y
MMOM(X, Y, Z)	Member moments for nodes I and J (in the element coordinate system)	*4*	Y
SFACTI, SFACTJ	Stress intensification factors at nodes I and J	Y	Y
STH	Stress due to maximum thermal gradient through the wall thickness	Y	Y
SPR2	Hoop pressure stress for code calculations	-	Y
SMI, SMJ	Moment stress at nodes I and J for code calculations	-	Y
SDIR	Direct (axial) stress	-	Y
SBEND	Maximum bending stress at outer surface	-	Y
ST	Shear stress at outer surface due to torsion	-	Y
SSF	Shear stress due to shear force	-	Y

Name	Definition	O	R
S(1MX, 3MN,INTMX, EQVMX)	Maximum principal stress, minimum principal stress, maximum stress intensity, maximum equivalent stress (all at the outer surface)	Y	Y
S(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*5*	*5*
S(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*5*	*5*
EPEL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*5*	*5*
EPTH(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*5*	*5*

1. If the value is greater than 0.
2. If KEYOPT(1) = 0
3. If KEYOPT(1) = 1
4. If KEYOPT(6) = 2
5. The item repeats at 0°, 45°, 90°, 135°, 180°, 225°, 270°, 315° at node I, then at node J (all at the outer surface)
6. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PIPE18 Item and Sequence Numbers (Node I)"* through *Table 5, "PIPE18 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "PIPE18 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 3 PIPE18 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	5	9	13	17	21	25	29
EPTHRAD	LEPTH	-	2	6	10	14	18	22	26	30

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
EPTHH	LEPTH	-	3	7	11	15	19	23	27	31
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	91	-	-	-	-	-	-	-	-
SSF	NMISC	92	-	-	-	-	-	-	-	-
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	13	-	-	-	-	-	-	-	-
ST	SMISC	14	-	-	-	-	-	-	-	-
TOUT	LBFE	-	4	-	1	-	2	-	3	-
TIN	LBFE	-	8	-	5	-	6	-	7	-

Table 4 PIPE18 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64
EPTHAXL	LEPTH	-	33	37	41	45	49	53	57	61
EPTHRAD	LEPTH	-	34	38	42	46	50	54	58	62
EPTHH	LEPTH	-	35	39	43	47	51	55	59	63
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	93	-	-	-	-	-	-	-	-
SSF	NMISC	94	-	-	-	-	-	-	-	-
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	15	-	-	-	-	-	-	-	-
ST	SMISC	16	-	-	-	-	-	-	-	-
TOUT	LBFE	-	12	-	9	-	10	-	11	-
TIN	LBFE	-	16	-	13	-	14	-	15	-

Table 5 PIPE18 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SFACTI	NMISC	81
SFACTJ	NMISC	82
SPR2	NMISC	83
SMI	NMISC	84
SMJ	NMISC	85
S1MX	NMISC	86
S3MN	NMISC	87
SINTMX	NMISC	88
SEQVMX	NMISC	89
FFACT	NMISC	90
STH	SMISC	17
PINT	SMISC	18
PX	SMISC	19
PY	SMISC	20
PZ	SMISC	21
POUT	SMISC	22

PIPE18 Assumptions and Restrictions

- The curved pipe must not have a zero length or wall thickness. In addition, the OD must not be less than or equal to zero and the ID must not be less than zero.
- The corrosion allowance must be less than the wall thickness.
- The element is limited to having an axis with a single curvature and a subtended angle of $0° < \theta \leq 90°$.
- Shear deflection capability is also included in the element formulation.
- The elbow is assumed to have "closed ends" so that the axial pressure effect is included.
- When used in a large deflection analysis, the location of the third node (K) is used only to initially orient the element.

- The element temperatures are assumed to be linear along the length. The average wall temperature at θ = 0° is computed as 2 * TAVG - T(180) and the average wall temperature at θ = -90° is computed as 2 * TAVG - T(90).
- Stress intensification factors input with values less than 1.0 are set to 1.0.
- The element formulation is based upon thin-walled theory. The elbow should have a large radius-to-thickness ratio since the integration points are assumed to be located at the midthickness of the wall.
- Only the lumped mass matrix is available.

PIPE18 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special feature allowed is large deflection.

PIPE20

Plastic Straight Thin-Walled Pipe

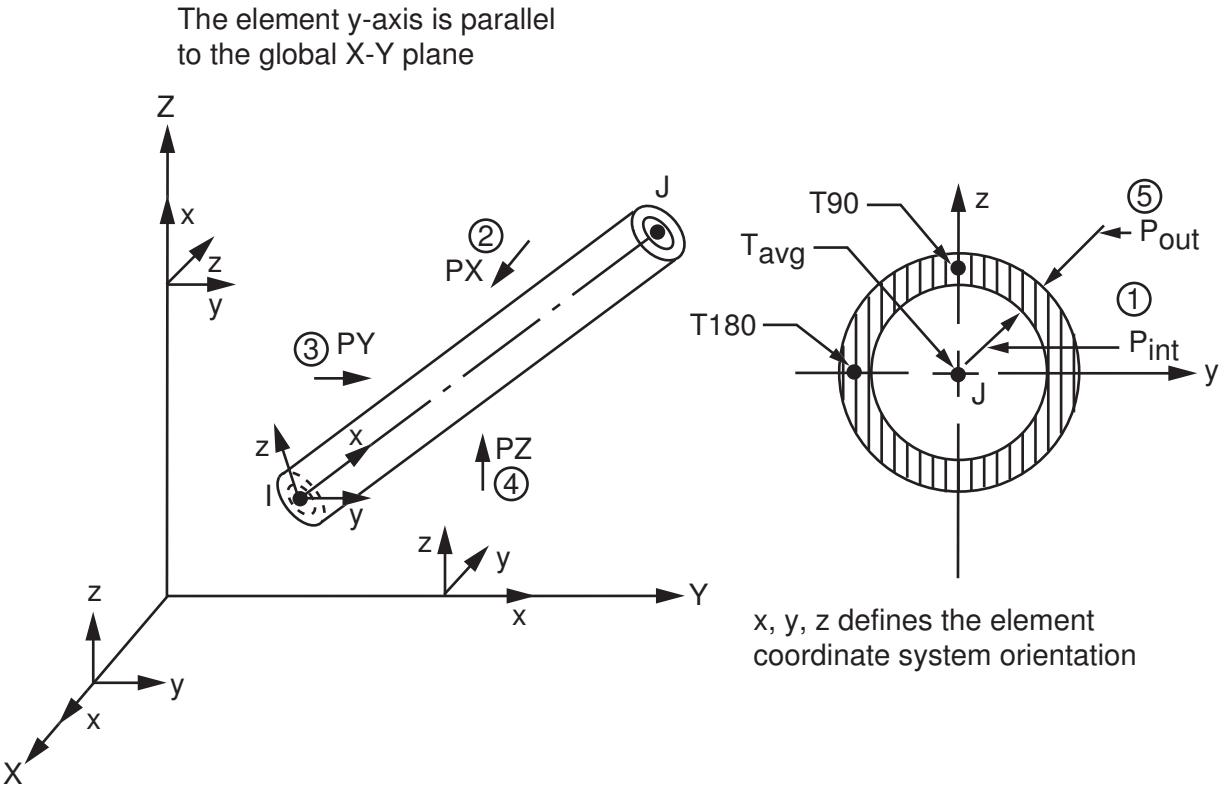
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

PIPE20 Element Description

PIPE20 is a uniaxial element with tension-compression, bending, and torsion capabilities. The element has six degrees of freedom at each node: translations in the nodal, x, y, and z directions, and rotations about the nodal x, y, and z axes.

The element has plastic, creep and swelling capabilities. If these effects are not needed, the elastic pipe element, PIPE16, may be used. An option is available for printing the forces and moments acting on the element in the element coordinate system. See PIPE20 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PIPE60 for a plastic curved pipe element.

Figure 1 PIPE20 Geometry



PIPE20 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE20 Geometry"*. The element input data include two nodes, the pipe outer diameter and wall thickness, optional stress factors, and the isotropic material properties.

The element X-axis is oriented from node I toward node J. The element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in *Figure 1, "PIPE20 Geometry"*. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element Y-axis is oriented

parallel to the global Y-axis (as shown). Input and output locations around the pipe circumference identified as being at 0° are located along the element Y-axis, and similarly 90° is along the element Z-axis.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE20 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. Tapered pressures are not recognized. Only constant pressures are supported for this element. See the *Theory Reference for ANSYS and ANSYS Workbench* for details.

Temperatures and fluences may be input as element body loads at the nodes. The first temperature (TAVG at node I) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other pattern of input temperatures, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

A summary of the element input is given in *PIPE20 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

PIPE20 Input Summary

Nodes
I, J (node I defines end 1)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

OD - Pipe outer diameter
TKWALL - Wall thickness
SIFI - Stress intensification factor (used only if KEYOPT (2) = 4)
SIFJ - Stress intensification factor (used only if KEYOPT (2) = 4)

Material Properties
EX, ALPX (or CTEX *or* THSX), PRXY (or NUXY), DENS, GXY, DAMP

Surface Loads

Pressures --
1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --
TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)

Fluences --
FLAVG(I), FL90(I), FL180(I), FLAVG(J), FL90(J), FL180(J)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(2)
Stress intensity factor:

0 --
No stress intensification factors

4 --
Include stress intensification factors at nodes I and J as input with real constants

KEYOPT(6)
Member force and moment output:

0 --
Do not print member forces or moments

1 --
Print member forces and moments in the element coordinate system

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

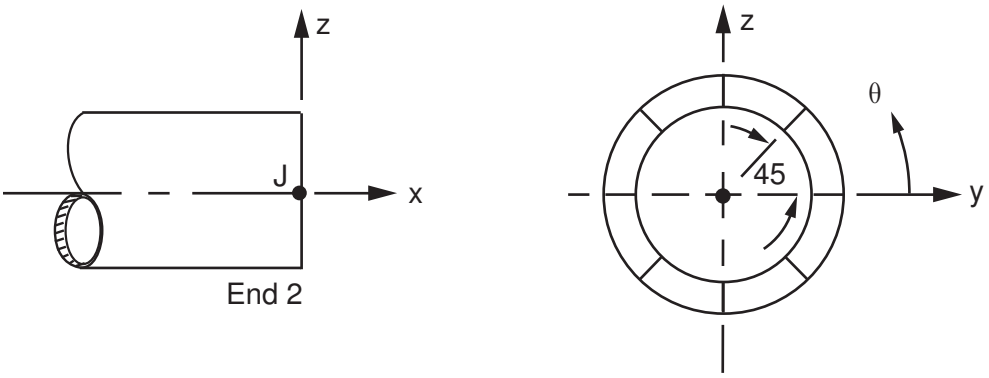
PIPE20 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PIPE20 Element Output Definitions"*

The meaning of THETA is illustrated in *Figure 2, "PIPE20 Stress Output"*. The nonlinear solution is given at eight circumferential locations at both ends of the pipe. The linear solution, similar to that for PIPE16, is also printed as long as the element remains elastic. The initial elastic bending stresses (SBEND) are multiplied by the input stress intensification factors (SIFI and SIFJ) for KEYOPT(2) = 4, provided they are greater than 1.0. No multiplication is done for any other stresses, or for plasticity. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PIPE20 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PIPE20 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*4*
TEMP	Temperatures TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)	Y	Y
FLUEN	Fluences FLAVG(I), FL90(I), FL180(I), FLAVG(J), FL90(J), FL180(J)	Y	Y
PRES	Pressures PINT, PX, PY, PZ, POUT	Y	Y
MFOR(X, Y, Z)	Member forces for nodes I and J (in the element coordinate system)	*1*	*1*
MMOM(X, Y, Z)	Member moments for nodes I and J (in the element coordinate system)	*1*	*1*
SDIR	Direct (axial) stress	-	*2*
SBEND	Maximum bending stress at outer surface	-	*2*
ST	Shear stress at outer surface due to torsion	-	*2*
SSF	Shear stress due to shear force	-	*2*
S1MX, S3MN	Maximum principal stress, minimum principal stress	*2*	*2*
SINTMX, SEQVMX	Maximum stress intensity, maximum equivalent stress all at the outer surface (based on SDIR, SBEND, ST, SSF but also accounting for the values of S1, S3, SINT, SEQV given below)	*2*	*2*
S(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*3*	*3*

Name	Definition	O	R
S(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*3*	*3*
EPEL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*3*	*3*
EPTH(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*3*	*3*
EPSWAXL	Axial swelling strain	*3*	*3*
EPPL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear plastic strains	*3*	*3*
EPCR(AXL, RAD, H, XH)	Axial, radial, hoop, and shear creep strains	*3*	*3*
SEPL	Equivalent stress from stress-strain curve	*3*	*3*
SRAT	Ratio of trial stress to stress on yield surface	*3*	*3*
HPRES	Hydrostatic pressure	-	*3*
EPEQ	Equivalent plastic strain	*3*	*3*

1. If KEYOPT(6) = 1
2. Initial elastic solution only before yield
3. The item repeats for THETA = 0°, 45°, 90°, 135°, 180°, 225°, 270°, 315° at node I, then at node J, all at the mid-thickness of the wall
4. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "PIPE20 Item and Sequence Numbers (Node I)"* through *Table 4, "PIPE20 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PIPE20 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 2 PIPE20 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	6	11	16	21	26	31	36

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
EPTHRAD	LEPTH	-	2	7	12	17	22	27	32	37
EPTHH	LEPTH	-	3	8	13	18	23	28	33	38
EPSWAXL	LEPTH	-	5	10	15	20	25	30	35	40
EPPLAXL	LEPPL	-	1	5	9	13	17	21	25	29
EPPLRAD	LEPPL	-	2	6	10	14	18	22	26	30
EPPLH	LEPPL	-	3	7	11	15	19	23	27	31
EPPLXH	LEPPL	-	4	8	12	16	20	24	28	32
EPCRAXL	LEPCR	-	1	5	9	13	17	21	25	29
EPCRRAD	LEPCR	-	2	6	10	14	18	22	26	30
EPCRH	LEPCR	-	3	7	11	15	19	23	27	31
EPCRXH	LEPCR	-	4	8	12	16	20	24	28	32
SEPL	NLIN	-	1	5	9	13	17	21	25	29
SRAT	NLIN	-	2	6	10	14	18	22	26	30
HPRES	NLIN	-	3	7	11	15	19	23	27	31
EPEQ	NLIN	-	4	8	12	16	20	24	28	32
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	81	-	-	-	-	-	-	-	-
SSF	NMISC	82	-	-	-	-	-	-	-	-
S1MX	NMISC	101	-	-	-	-	-	-	-	-
S3MN	NMISC	102	-	-	-	-	-	-	-	-
SINTMX	NMISC	103	-	-	-	-	-	-	-	-
SEQVMX	NMISC	104	-	-	-	-	-	-	-	-
FOUT	NMISC	-	88	-	85	-	86	-	87	-
FIN	NMISC	-	92	-	89	-	90	-	91	-
TOUT	LBFE	-	4	-	1	-	2	-	3	-
TIN	LBFE	-	8	-	5	-	6	-	7	-
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	13	-	-	-	-	-	-	-	-
ST	SMISC	14	-	-	-	-	-	-	-	-

Table 3 PIPE20 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64
EPTHAXL	LEPTH	-	41	46	51	56	61	66	71	76
EPTHRAD	LEPTH	-	42	47	52	57	62	67	72	77
EPTHH	LEPTH	-	43	48	53	58	63	68	73	78
EPSWAXL	LEPTH	-	45	50	55	60	65	70	75	80
EPPLAXL	LEPPL	-	33	37	41	45	49	53	57	61
EPPLRAD	LEPPL	-	34	38	42	46	50	54	58	62
EPPLH	LEPPL	-	35	39	43	47	51	55	59	63
EPPLXH	LEPPL	-	36	40	44	48	52	56	60	64
EPCRAXL	LEPCR	-	33	37	41	45	49	53	57	61
EPCRRAD	LEPCR	-	34	38	42	46	50	54	58	62
EPCRH	LEPCR	-	35	39	43	47	51	55	59	63
EPCRXH	LEPCR	-	36	40	44	48	52	56	60	64
SEPL	NLIN	-	33	37	41	45	49	53	57	61
SRAT	NLIN	-	34	38	42	46	50	54	58	62
HPRES	NLIN	-	35	39	43	47	51	55	59	63
EPEQ	NLIN	-	36	40	44	48	52	56	60	64
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	83	-	-	-	-	-	-	-	-
SSF	NMISC	84	-	-	-	-	-	-	-	-
S1MX	NMISC	105	-	-	-	-	-	-	-	-
S3MN	NMISC	106	-	-	-	-	-	-	-	-
SINTMX	NMISC	107	-	-	-	-	-	-	-	-
SEQVMX	NMISC	108	-	-	-	-	-	-	-	-
FOUT	NMISC	-	96	-	93	-	94	-	95	-
FIN	NMISC	-	100	-	97	-	98	-	99	-
TOUT	LBFE	-	12	-	9	-	10	-	11	-
TIN	LBFE	-	16	-	13	-	14	-	15	-
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	15	-	-	-	-	-	-	-	-
ST	SMISC	16	-	-	-	-	-	-	-	-

Table 4 PIPE20 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PINT	SMISC	17
PX	SMISC	18
PY	SMISC	19
PZ	SMISC	20
POUT	SMISC	21

PIPE20 Assumptions and Restrictions

- The pipe element is assumed to have "closed ends" so that the axial pressure effect is included.
- The equations used in the development of this element are the standard equations for small deflection of beams, including shear deflections.
- The element formulation is based upon thin-walled theory. The elbow should have a large radius-to-thickness ratio since the integration points are assumed to be located at the midthickness of the wall. If the ratio is less than 5.0 (OD/TKWALL = 10.0), an error message will be generated. If the ratio is less than 10.0 (OD/TKWALL = 20.0), a warning message will be generated.
- The average wall temperature at $\theta = 0°$ is computed as 2 * TAVG - T(180) and the average wall temperature at $\theta = -90°$ is computed as 2 * TAVG - T(90).
- The element temperatures are assumed to vary linearly along the length.
- Stress intensification factors input with values less than 1.0 are set to 1.0.

PIPE20 Product Restrictions

There are no product-specific restrictions for this element.

MASS21

Structural Mass

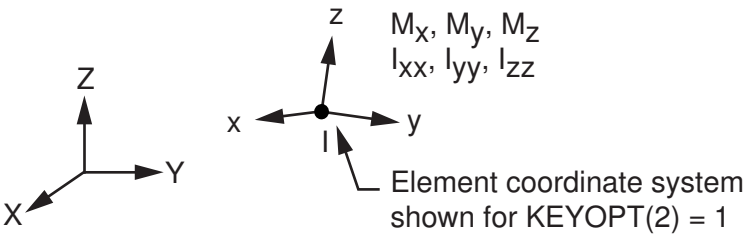
MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

MASS21 Element Description

MASS21 is a point element having up to six degrees of freedom: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction. See MASS21 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Another element with a full mass matrix capability (off-diagonal terms) is MATRIX27.

Figure 1 MASS21 Geometry



MASS21 Input Data

The mass element is defined by a single node, concentrated mass components (Force*Time^2/Length) in the element coordinate directions, and rotary inertias (Force*Length*Time^2) about the element coordinate axes. The element coordinate system may be initially parallel to the global Cartesian coordinate system or to the nodal coordinate system (KEYOPT(2)). The element coordinate system rotates with the nodal coordinate rotations during a large deflection analysis. Options are available to exclude the rotary inertia effects and to reduce the element to a 2-D capability (KEYOPT(3)). If the element requires only one mass input, it is assumed to act in all appropriate coordinate directions. The coordinate system for this element is shown in *Figure 1, "MASS21 Geometry"*.

KEYOPT(1) = 1 defines the mass in volume*density form, which allows plotting of the mass using **/ESHAPE**, as well as the use of a temperature-dependent density.

A summary of the element input is given in *MASS21 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

MASS21 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT (3) = 0
UX, UY, UZ if KEYOPT (3) = 2
UX, UY, ROTZ if KEYOPT (3) = 3
UX, UY if KEYOPT (3) = 4
(degrees of freedom are in the nodal coordinate system)

Real Constants

MASSX, MASSY, MASSZ, IXX, IYY, IZZ, if KEYOPT (3) = 0
MASS, if KEYOPT (3) = 2
MASS, IZZ, if KEYOPT (3) = 3
MASS, if KEYOPT (3) = 4
(MASSX, MASSY, and MASSZ are concentrated mass components in the element coordinate directions. IXX, IYY, and IZZ are rotary inertias about the element coordinate axes. See also KEYOPT(2)).

Material Properties
DENS (if KEYOPT(1) = 1)

Surface Loads
None

Body Loads
None

Special Features

Large deflection
Birth and death

KEYOPT(1)
Real constant interpretation (mass/volume or rotary inertia/density):

0 --
Interpret real constants as masses and rotary inertias

1 --
Interpret real constants as volumes and rotary inertias/density (Density must be input as a material property)

KEYOPT(2)
Initial element coordinate system:

0 --
Element coordinate system is initially parallel to the global Cartesian coordinate system

1 --
Element coordinate system is initially parallel to the nodal coordinate system

KEYOPT(3)
Rotary inertia options:

0 --
3-D mass with rotary inertia

2 --
3-D mass without rotary inertia

3 --
2-D mass with rotary inertia

4 --
2-D mass without rotary inertia

MASS21 Output Data

Nodal displacements are included in the overall displacement solution. There is no element solution output associated with the element unless element reaction forces and/or energies are requested.

MASS21 Assumptions and Restrictions

- 2-D elements are assumed to be in a global Cartesian Z = constant plane.
- The mass element has no effect on the static analysis solution unless acceleration or rotation is present, or inertial relief is selected [**IRLF**].
- The standard mass summary printout is based on the average of MASSX, MASSY, and MASSZ if (KEYOPT(3) = 0).
- In an inertial relief analysis, the full matrix is used. All terms are used during the analysis.

MASS21 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is large deflection.

BEAM23

2-D Plastic Beam

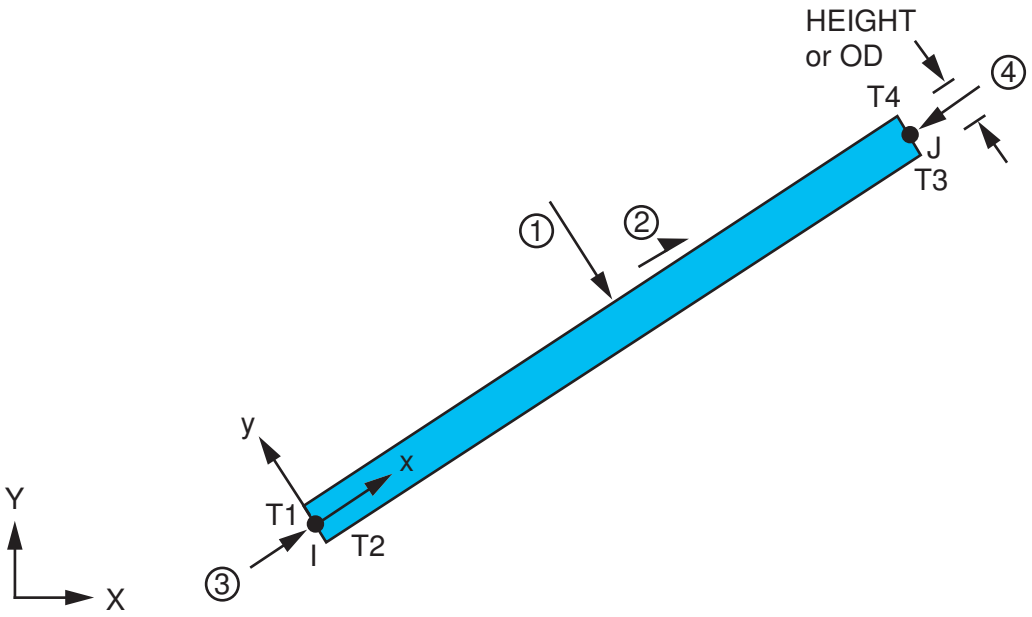
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

BEAM23 Element Description

BEAM23 is a uniaxial element with tension-compression and bending capabilities. The element has three degrees of freedom at each node: translations in the nodal x and y direction and rotation about the nodal z-axis. See BEAM23 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

The element has plastic, creep, and swelling capabilities. If these effects are not needed, BEAM3, the 2-D elastic beam, may be used. See BEAM54 for a 2-D tapered elastic beam.

Figure 1 BEAM23 Geometry



BEAM23 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "BEAM23 Geometry"*. Any one of four cross-sections may be selected with the appropriate value of KEYOPT(6). The element is defined by two nodes, the cross-sectional area, moment of inertia, the height for rectangular beams, the outer diameter (OD), and the wall thickness (TKWALL), for thin walled pipes, the outer diameter for solid circular bars, and the isotropic material properties.

The general cross-section option (KEYOPT(6) = 4) allows inputting a section height and a five-location area distribution. If the section is symmetric, only the first three of the five areas need be input since the fourth area defaults to the second and the fifth area defaults to the first. The areas input should be a weighted distribution at the -50% integration point A(-50), the -30% integration point A(-30), the 0% integration point A(0), the 30% integration point A(30), and the 50% integration point A(50). Each area A(i) is as shown in *Figure 2, "BEAM23 Weighting Functions for General Section (KEYOPT(6) = 4)"*. The height is defined as the distance between the ± 50% integration points, and is not necessarily the distance between the outermost fibers of the section. Determination of the input areas is accomplished as follows. Estimate one of the input areas by the formula A(i) = L(i) x HEIGHT, where L(i) is the width of the section at integration point i (see *Figure 2, "BEAM23 Weighting Functions for General Section*

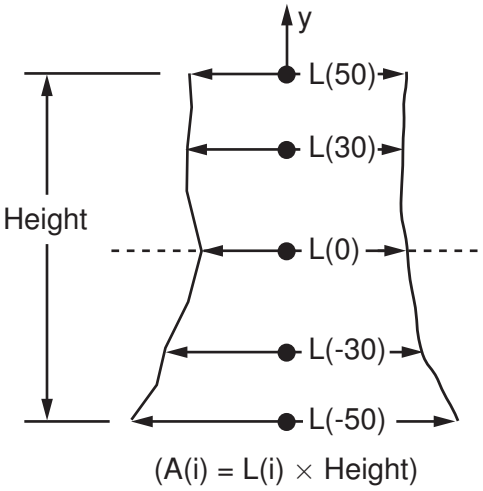
(KEYOPT(6) = 4)". Substitute this area along with the section moment of inertia, I_{zz}, and total area, A, into the above equations and solve them simultaneously for the remaining two input areas. A(0) is usually the easiest to estimate; for instance, as a first guess A(0) for an I-beam would be the web thickness times the height. A trial and error procedure (by modifying the estimated input area) may be needed if the calculated input areas are inconsistent, such as a negative area. The input areas, A(i), are related to the true areas, A_t(i), corresponding to each integration point, by:

A_t (-50) = 0.0625 A(-50), A_t (50) = 0.0625 A(50),

A_t(-30) = 0.28935 A(-30), A_t (30) = 0.28935 A(30),

A_t (0) = 0.29630 A(0)

Figure 2 BEAM23 Weighting Functions for General Section (KEYOPT(6) = 4)



Shear deflection may be controlled with the KEYOPT(2) value. The shear deflection constant (SHEARZ) is input only for the general cross-section. See *Section 2.14: Shear Deflection* for details. The shear modulus (GXY) is used only with shear deflection.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM23 Geometry"*. Positive normal pressures act into the element. Lateral pressures are input as a force per unit length. End "pressures" are input as a force. KEYOPT(10) allows tapered lateral pressures to be offset from the nodes. Temperatures and fluences may be input as element body loads at the four "corner" locations shown in *Figure 1, "BEAM23 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

A summary of the element input is given in *BEAM23 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

BEAM23 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, ROTZ

Real Constants

See *Table 1, "BEAM23 Real Constants"* for descriptions of the real constants.

Material Properties

EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J) -Y normal direction
face 2 (I-J) +X tangential direction
face 3 (I) +X axial direction
face 4 (J) X axial direction (use negative value for loading in opposite direction)

Body Loads

Temperatures --

T1, T2, T3, T4

Fluences --

FL1, FL2, FL3, FL4

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(2)

Shear deflection:

0 --

No shear deflection

1 --

Include shear deflection (also input SHEARZ if KEYOPT(6) = 4)

KEYOPT(4)

Member force and moment output:

0 --

No printout of member forces and moments

1 --

Print out member forces and moments in the element coordinate system

KEYOPT(6)

Cross-section:

0 --
Rectangular section

1 --
Thin walled pipe

2 --
Round solid bar

4 --
General section

KEYOPT(10)
Load location, used in conjunction with the offset values input on the **SFBEAM** command):

0 --
Offset is in terms of length units

1 --
Offset is in terms of a length ratio (0.0 to 1.0)

Table 1 BEAM23 Real Constants

No.	Name	Description
Rectangular Section (KEYOPT(6) = 0)		
1	AREA	Cross-sectional area
2	IZZ	Area moment of inertia
3	HEIGHT	Section height
Thin Walled Pipe (KEYOPT(6) = 1)		
1	OD	Outer diameter
2	WTHK	Wall thickness
Round Solid Bar (KEYOPT(6) = 2)		
1	OD	Outer diameter
General Section (KEYOPT(6) = 4)		
1	HEIGHT	Section height
2	A(-50)	Area at given location (see *Figure 2, "BEAM23 Weighting Functions for General Section (KEYOPT(6) = 4)"*)
3	A(-30)	Area at given location
4	A(0)	Area at given location
5	A(30)	Area at given location
6	A(50)	Area at given location
7	SHEARZ	Shear deflection constant

BEAM23 Output Data

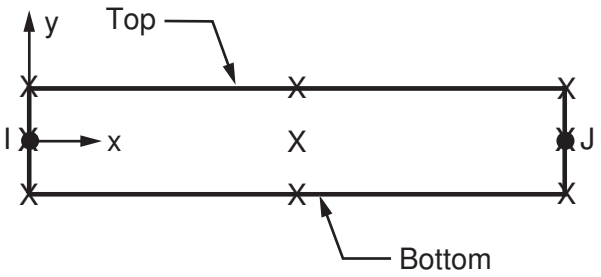
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "BEAM23 Element Output Explanations"*.

Several items are illustrated in *Figure 2, "BEAM23 Weighting Functions for General Section (KEYOPT(6) = 4)"*.

The printout contains the stresses and strains at nine locations in the beam. The locations are at three points through the height of the element (bottom, middle, and top) at each of three axial stations (end I, midlength, and end J) (see *Figure 3, "BEAM23 Printout Locations"*). The post data items [**ETABLE**] contain the stresses and strains at the five weighted-area locations (regardless of the KEYOPT(6) setting) at each of the three axial stations. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 3 BEAM23 Printout Locations



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 BEAM23 Element Output Explanations

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*3*
TEMP	Temperatures T1, T2, T3, T4	Y	Y
FLUEN	Fluences FL1, FL2, FL3, FL4	Y	Y
PRES	Pressures P1 at nodes I, J; OFFST1 at I, J; P2 at I, J; OFFST2 at I, J; P3 at I; P4 at J	Y	Y
S(MAX, MIN)	Maximum axial stress, minimum axial stress	-	Y
SAXL	Axial stress	*1*	*1*
EPELAXL	Axial elastic strain	*1*	*1*
EPTHAXL	Axial thermal strain	*1*	*1*
EPSWAXL	Axial swelling strain	*1*	*1*
EPCRAXL	Axial creep strain	*1*	*1*
EPPLAXL	Axial plastic strain	*1*	*1*
SEPL	Equivalent stress from stress-strain curve	*1*	*1*
SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
EPEQ	Equivalent plastic strain	*1*	*1*

Name	Definition	O	R
HPRES	Hydrostatic pressure	-	*1*
MFOR(X, Y)	Member forces for each node in the element coordinate system	*2*	Y
MMOMZ	Member moments for each node in the element coordinate system	*2*	Y

1. The item repeats at the top, middle, and bottom for end I, midlength, and end J
2. If KEYOPT(4) = 1
3. Available only at centroid as a ***GET** item.

The following tables list output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "BEAM23 Item and Sequence Numbers (Node I)"* through *Table 5, "BEAM23 Item and Sequence Numbers (Node J)"*:

Name
output quantity as defined in the *Table 2, "BEAM23 Element Output Explanations"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

-50, -30, 0, 30, 50
sequence number for data at weighted-area locations

Table 3 BEAM23 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	% Integration Point				
			-50	-30	0	30	50
SAXL	LS	-	1	2	3	4	5
EPELAXL	LEPEL	-	1	2	3	4	5
EPTHAXL	LEPTH	-	1	3	5	7	9
EPSWAXL	LEPTH	-	2	4	6	8	10
EPPLAXL	LEPPL	-	1	2	3	4	5
EPCRAXL	LEPCR	-	1	2	3	4	5
SEPL	NLIN	-	1	5	9	13	17
SRAT	NLIN	-	2	6	10	14	18
HPRES	NLIN	-	3	7	11	15	19
EPEQ	NLIN	-	4	8	12	16	20
MFORX	SMISC	1	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-
P1	SMISC	13	-	-	-	-	-
P2	SMISC	17	-	-	-	-	-
P3	SMISC	21	-	-	-	-	-
SMAX	NMISC	1	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	% Integration Point				
			-50	-30	0	30	50
SMIN	NMISC	2	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	Corner Location		
			1	2
FLUEN	NMISC	-	7	8
TEMP	LBFE	-	1	2

Table 4 BEAM23 Item and Sequence Numbers (Midlength)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	% Integration Point				
			-50	-30	0	30	50
SAXL	LS	-	6	7	8	9	10
EPELAXL	LEPEL	-	6	7	8	9	10
EPTHAXL	LEPTH	-	11	13	15	17	19
EPSWAXL	LEPTH	-	12	14	16	18	20
EPPLAXL	LEPPL	-	6	7	8	9	10
EPCRAXL	LEPCR	-	6	7	8	9	10
SEPL	NLIN	-	21	25	29	33	37
SRAT	NLIN	-	22	26	30	34	38
HPRES	NLIN	-	23	27	31	35	39
EPEQ	NLIN	-	24	28	32	36	40
SMAX	NMISC	3	-	-	-	-	-
SMIN	NMISC	4	-	-	-	-	-

Table 5 BEAM23 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	% Integration Point				
			-50	-30	0	30	50
SAXL	LS	-	11	12	13	14	15
EPELAXL	LEPEL	-	11	12	13	14	15
EPTHAXL	LEPTH	-	21	23	25	27	29
EPSWAXL	LEPTH	-	22	24	26	28	30
EPPLAXL	LEPPL	-	11	12	13	14	15
EPCRAXL	LEPCR	-	11	12	13	14	15
SEPL	NLIN	-	41	45	49	53	57
SRAT	NLIN	-	42	46	50	54	58
HPRES	NLIN	-	43	47	51	55	59
EPEQ	NLIN	-	44	48	52	56	60
MFORX	SMISC	7	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	% Integration Point				
			-50	-30	0	30	50
MMOMZ	SMISC	12	-	-	-	-	-
P1	SMISC	14	-	-	-	-	-
P2	SMISC	18	-	-	-	-	-
P4	SMISC	22	-	-	-	-	-
SMAX	NMISC	5	-	-	-	-	-
SMIN	NMISC	6	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	Corner Location		
			3	4
FLUEN	NMISC	-	9	10
TEMP	LBFE	-	3	4

BEAM23 Assumptions and Restrictions

- The applied thermal gradient is assumed linear across the height of the element and along its length.
- The beam element must lie in an X-Y plane and must not have a zero length or area.
- The height is used in calculating the bending and thermal stresses and for locating the integration points.
- For the rectangular section (KEYOPT(6) = 0), the input area, moment of inertia, and height should be consistent with each other.
- The effect of implied offsets on the mass matrix (possible with KEYOPT(6) = 4) is ignored if the lumped mass matrix formulation is specified [**LUMPM**,ON].

BEAM23 Product Restrictions

There are no product-specific restrictions for this element.

BEAM24

3-D Thin-walled Beam

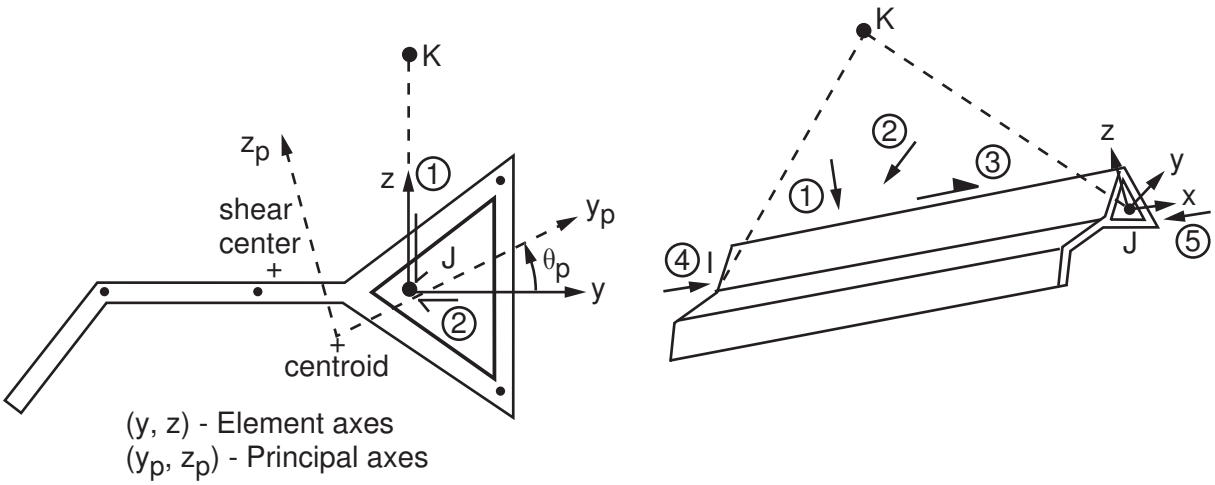
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

BEAM24 Element Description

BEAM24 is a uniaxial element of arbitrary cross-section (open or closed) with tension-compression, bending and St.Venant torsional capabilities. Any open cross-section or single-celled closed section can be used. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the x, y, and z axes.

The element has plastic, creep, and swelling capabilities in the axial direction as well as a user-defined cross-section. If these capabilities are not needed, the elastic beams BEAM4 or BEAM44 may be used. Other beam elements also having plastic, creep, and swelling capabilities are PIPE20 and BEAM23. The element also has stress stiffening, large deflection and shear deflection capabilities. The cross-section is defined by a series of rectangular segments. The orientation of the beam about its longitudinal axis is specified by a third node. See BEAM24 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 BEAM24 Geometry



BEAM24 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "BEAM24 Geometry"*. The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element z axis. The element x axis runs parallel to the centroidal line of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. If this element is used in large deflection analysis, the location of node K is used only to *initially* orient the element. (For information about orientation nodes and beam meshing, see Meshing Your Solid Model of the *Modeling and Meshing Guide*)

The cross-section is input as a continuous series of straight segments in the element y-z plane. The centroid and shear center locations of the beam, with respect to the origin, define the implied nodal offsets (unless KEYOPT(3) is used).

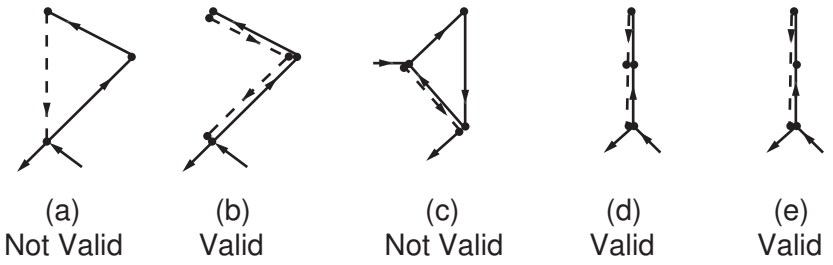
The element real constants are used to describe the cross-section of the beam. The input consists of the coordinates (y, z) of 20 segment points in the element y-z plane and the thickness of the corresponding segment (THK) in a

(y, z, THK) format. Not all 20 points need to be used in defining the cross-section. The segments are input such that they make a continuous outline of the cross-section - the end point of one segment is the beginning point of the next segment. Segments may be given a zero thickness in order to backtrack over previously defined segments to continue the outline. The thickness in (y, z, THK) is the thickness of the segment that is defined by this point and the previous point. The thickness of the first point is therefore not used and should be zero.

BEAM24 Input Restrictions

1. Zero thickness (backtrack) segments must follow the original geometry, i.e., they cannot enclose an area (see *Figure 2, "Valid and invalid uses of BEAM24"*, sections a and b, where the dashed lines represent zero thickness segments).
2. A zero thickness segment cannot be used anywhere in the outline of a closed loop (*Figure 2, "Valid and invalid uses of BEAM24"*, section c).
3. A straight zero thickness line need not have the same number of segments as the original straight line (*Figure 2, "Valid and invalid uses of BEAM24"*, sections d, e).
4. A single straight line cross-section is not permitted.
5. Multiple-celled closed sections (such as a double box beam) are not allowed.
6. Consecutive segment points must be a distance of at least 1.0E-8 units apart in either the y or z direction. Otherwise, the points are considered coincident.

Figure 2 Valid and invalid uses of BEAM24



The user is urged to verify the cross-section input by the calculated cross-section parameters in the element output and the geometry shape display ([**/ESHAPE**,1] and [**EPLOT**]).

Real constants DXI and DXJ define the rigid nodal offsets, measured positive from the node in the element x direction, at end I and end J respectively. The remaining real constants, SHEARZ and SHEARY, are the shear deflection constants and are computed in the principal coordinate system. They are only used if shear deflection is to be included. A zero value of SHEAR_ may be used to neglect shear deflection in a particular direction. See *Section 2.14: Shear Deflection* for details.

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause a bending of the element. If the axis through the shear center is not colinear with the element x-axis, applied shear forces will cause torsional rotation of the element. The nodes should therefore be located at the desired point of application of the forces.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM24 Geometry"*. Positive normal pressures act into the element. Lateral pressures are input as a force per unit length. End "pressures" are input as a force. Temperatures and fluences may be input as element body loads at three locations at each end of the beam. At end I, the element temperatures are input (see *Figure 1, "BEAM24 Geometry"*) at the element x-axis (T(0,0)), at one unit from the x-axis in the element y direction (T(1,0)), and at one unit from the x-axis in the element z direction

(T(0,1)). A similar temperature occurs at end J. The fluences are input in the same manner at both ends of the beam. The first coordinate temperature T(0,0) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

KEYOPT(2) allows a reduced mass matrix formulation (lumping procedure deleting off-diagonal terms having rotational degrees of freedom terms). This option is normally used only for very long, thin members. KEYOPT(3) allows the nodes to be located at the centroid or shear center, regardless of the location of the y-z origin (the default node location).

A summary of the element input is given in *BEAM24 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

BEAM24 Input Summary

Nodes
I, J, K (node K defines orientation)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
See *Table 1, "BEAM24 Real Constants"* for details on these real constants.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J) (-Z normal direction), face 2 (I-J) (-Y normal direction),
face 3 (I-J) (+X tangential direction), face 4 (I) (+X axial direction),
face 5 (J) (-X axial direction) (use negative value for opposite loading)

Body Loads

Temperatures --
T(0,0), T(1,0), T(0,1) at node I, same at node J

Fluences --
FL(0,0), FL(1,0), FL(0,1) at node I, same at node J

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Additional cross-section output:

0 --
No printout of additional cross-section check data

1 --
Print additional cross-section check data

KEYOPT(2)
Mass matrix type:

0 --
Consistent

1 --
Reduced

KEYOPT(3)
Location of nodes:

0 --
Origin of element Y-Z axes at nodes

1 --
Centroid of element at nodes

2 --
Shear center of element at nodes

KEYOPT(6)
Member force and moment output:

0 --
No member force printout

1 --
Print member forces and moments in the principal coordinate system

KEYOPT(10)
Load location, used in conjunction with the offset values input on the **SFBEAM** command):

0 --
Offset is in terms of length units

1 --
Offset is in terms of a length ratio (0.0 to 1.0)

Note

SHEARZ and SHEARY correspond to the principal coordinate system. SHEARZ goes with IZP and SHEARY goes with IYP. If SHEARZ = 0.0, there is no shear deflection in the principal Y direction.

Table 1 BEAM24 Real Constants

No.	Name	Description
1	Y1	Y coordinate at defined segment point 1
2	Z1	Z coordinate at defined segment point 1
3	THK1	Thickness at defined segment point 1
4	Y2	Y coordinate at defined segment point 2
5	Z2	Z coordinate at defined segment point 2

No.	Name	Description
6	THK2	Thickness at defined segment point 2
7, ... 60	Y3, Z3, THK3, ... Y20, Z20, THK20	Define input of Y*n*, Z*n*, and THK*n* for each segment point 3 through 20, as needed; these comprise the first **RMORE** command through the ninth **RMORE** command
61	DXI	Rigid nodal offset (I-node); this starts the tenth **RMORE** command
62	DXJ	Rigid nodal offset (J-node)
63	SHEARZ	Shear deflection constant for Z
64	SHEARY	Shear deflection constant for Y

BEAM24 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "BEAM24 Element Output Definitions"*.

In addition, printout of the segment point locations and other cross-section data can be obtained with KEYOPT(1) = 1. The solution header (printed only once per element per run) consists of the calculated cross-sectional parameters: centroid and shear center location, area, torsional constant, warping moment of inertia, and the principal moments of inertia along with the angle (θ_p) between the element y-axis and one of the principal axes. The computed output consists of the axial stresses and strains at each segment point. A coincident segment point is not printed but is output for postprocessing. If KEYOPT(6) = 1, the 12-member forces and moments (6 at each end) are also printed in the principal coordinate system. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 BEAM24 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes I, J; P2 at I,J; P3 at I,J; P4 at I; P5 at J	Y	Y
TEMP	Temperatures T(0,0), T(1,0), T(0,1) at node I, same at node J	Y	Y
FLUEN	Fluences FL(0,0), FL(1,0), FL(0,1) at node I, same at node J	Y	Y
S(MAX, MIN)	Maximum axial stress, minimum axial stress	Y	Y
CENTROID	Centroid location (Y, Z)	Y	-
SHEAR CENTER	Shear center location (Y, Z)	Y	-

Name	Definition	O	R
AREA	Cross-sectional area	Y	-
J	Torsional constant	Y	-
IW	Warping moment of inertia	Y	-
IYP	Moment of inertia about principal Y axis	Y	-
IZP	Moment of inertia about principal Z axis	Y	-
THETAP	Rotation angle (radians) from element Y-axis to principal Y-axis	Y	-
END	End I or end J	*1*	-
PT	Segment point number (1-20)	*1*	-
TEMP	Temperature	*1*	*1*
SAXL	Axial stress	*1*	*1*
EPELAXL	Axial elastic strain	*1*	*1*
EPTHAXL	Axial thermal strain	*1*	*1*
EPSWAXL	Axial swelling strain	*1*	*1*
EPCRAXL	Axial creep strain	*1*	*1*
EPPLAXL	Axial plastic strain	*1*	*1*
SEPL	Equivalent stress from stress-strain curve	*1*	*1*
SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
EPEQ	Equivalent plastic strain	*1*	*1*
HPRES	Hydrostatic pressure	-	*1*
MFOR(X, Y, Z)	Member forces for each node in the principal coordinate system	*2*	Y
MMOM(X, Y, Z)	Member moments for each node in the principal coordinate system	*2*	Y

1. The segment point solution value for the specified END and PT
2. If KEYOPT(6) = 1
3. Available only at centroid as a ***GET** item.

Table 3, "BEAM24 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "BEAM24 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "BEAM24 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J
: sequence number for data at nodes I and J

Table 3 BEAM24 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
SAXL	LS	i	20+i
EPELAXL	LEPEL	i	20+i
EPTHAXL	LEPTH	2*i-1	40+(2*i-1)
EPSWAXL	LEPTH	2*i	40+(2*i)
EPPLAXL	LEPPL	i	20+i
EPCRAXL	LEPCR	i	20+i
SEPL	NLIN	4*i-3	80+(4*i-3)
SRAT	NLIN	4*i-2	80+(4*i-2)
HPRES	NLIN	4*i-1	80+(4*i-1)
EPEQ	NLIN	4*i	80+(4*i)
MFORX	SMISC	1	7
MFORY	SMISC	2	8
MFORZ	SMISC	3	9
MMOMX	SMISC	4	10
MMOMY	SMISC	5	11
MMOMZ	SMISC	6	12
P1	SMISC	13	14
P2	SMISC	17	18
P3	SMISC	21	22
P4	SMISC	25	-
P5	SMISC	-	26
SMAX	NMISC	1	3
SMIN	NMISC	2	4
FL(0,0)	NMISC	5	8
FL(1,0)	NMISC	6	9
FL(0,1)	NMISC	7	10
T(0,0)	LBFE	1	4
T(1,0)	LBFE	2	5
T(0,1)	LBFE	3	6

Note

The i in *Table 3, "BEAM24 Item and Sequence Numbers"* refers to a segment point of the beam where $1 \leq i \geq 20$.

BEAM24 Assumptions and Restrictions

- The beam must not have a zero length.
- The beam can have any open or single-cell closed cross-sectional shape for which the area and moments of inertia are nonzero.
- Warping torsion is assumed negligible and the warping moment of inertia is not used in the stiffness computation.

- Nonlinear material effects are only included in the axial direction (shear and torsional nonlinear material effects are neglected).
- The beam is assumed to be thin-walled (small strain) with a non-deforming cross-section. Warping of the cross-section is unconstrained and is the same for all cross-sections; therefore, the torsional rotation of the cross-section is assumed to vary linearly along the length. The effect of implied offsets on the mass matrix is ignored if the lumped mass matrix formulation is specified [**LUMPM**,ON].

BEAM24 Product Restrictions

There are no product-specific restrictions for this element.

PLANE25

Axisymmetric-Harmonic 4-Node Structural Solid

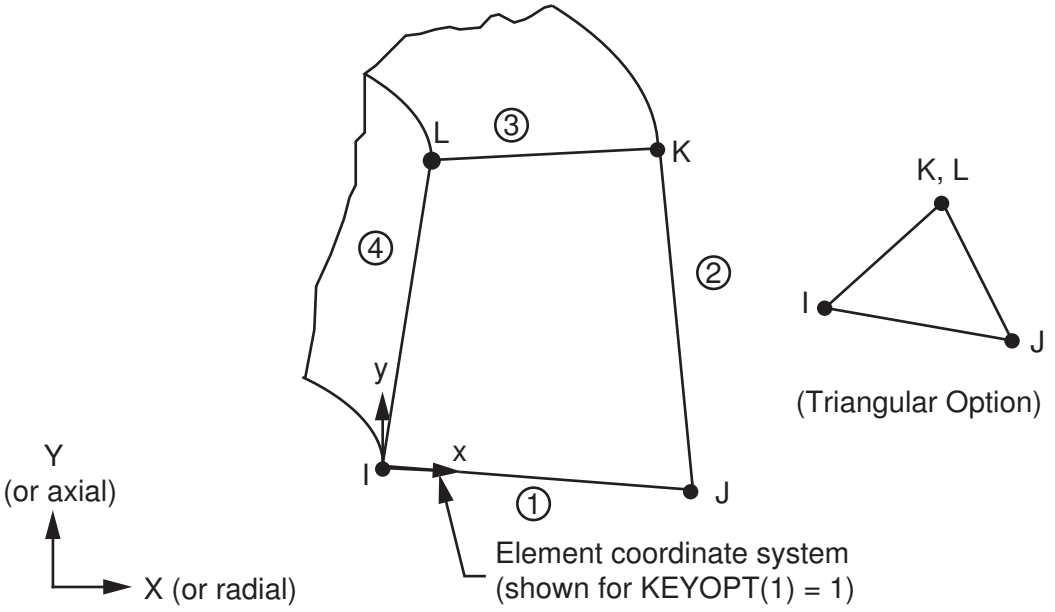
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

PLANE25 Element Description

PLANE25 is used for 2-D modeling of axisymmetric structures with nonaxisymmetric loading. Examples of such loading are bending, shear, or torsion. The element is defined by four nodes having three degrees of freedom per node: translations in the nodal x, y, and z direction. For unrotated nodal coordinates, these directions correspond to the radial, axial, and tangential directions, respectively.

The element is a generalization of the axisymmetric version of PLANE42, the 2-D structural solid element, in that the loading need not be axisymmetric. See *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads* for a description of various loading cases. See PLANE25 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PLANE83 for a multi-node version of this element.

Figure 1 PLANE25 Geometry



PLANE25 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE25 Geometry"*. The element input data includes four nodes, the number of harmonic waves (MODE on the **MODE** command), the symmetry condition (ISYM on the **MODE** command) and the orthotropic material properties. If MODE = 0, the element behaves similar to the axisymmetric case of PLANE42. The MODE and ISYM parameters are discussed in detail in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

The material may be orthotropic, with directions corresponding to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Harmonically varying nodal forces, if any, should be input on a full 360° basis.

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE25 Geometry"*. Positive pressures act into the element.

Harmonically varying temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(2) is used to include or suppress the extra displacement shapes. KEYOPT(3) is used for temperature loading with MODE greater than zero and temperature dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If MODE equals zero, material properties are always evaluated at the average element temperature.

KEYOPT(4), (5), and (6) provide various element printout options (see *Section 2.2.2: Element Solution*).

A summary of the element input is given in *PLANE25 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

PLANE25 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Mode Number --
Input mode number on **MODE** command

Loading Condition --
Input this value for *ISYM* on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

Special Features

Stress stiffening
Birth and death

KEYOPT(1)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side.

KEYOPT(2)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(3)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(4)
Extra stress output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points

2 --
Nodal Stress Solution

KEYOPT(5)
Combined stress output:

0 --
No combined stress solution

1 --
Combined stress solution at centroid and nodes

KEYOPT(6)
Include extra surface output (surface solution valid only for isotropic materials):

0 --
Basic element solution

1 --
Surface solution for face I-J also

2 --
Surface solution for both faces I-J and K-L also

PLANE25 Output Data

The solution output associated with the element is in two forms:

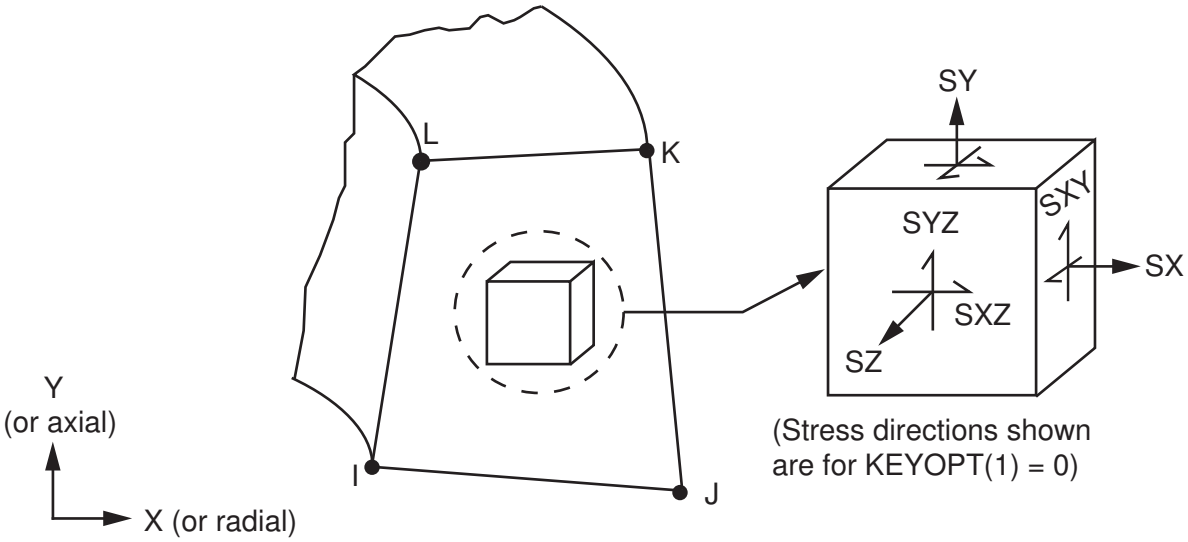
- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE25 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "PLANE25 Stress Output"*.

In the displacement printout, the UZ component is out-of-phase with the UX and UY components. For example, in the MODE = 1, ISYM = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. The same occurs for the reaction forces (FX, FY, etc.). The element stress directions are parallel to the element coordinate system. We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Non-axisymmetric Loads*

The sign convention on the surface shears is such that for a rectangular element that is lined up parallel to the axes with node J in the positive Y direction from node I, the shear stresses on surfaces I-J and K-L are analogous to the centroidal SYZ in both definition and sign. Stress components which are inherently zero for a load case are printed for clarity. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE25 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE25 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	-
MODE	Number of waves in loading	Y	-
VOLU:	Volume	Y	Y
PRES	Pressure P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y

Name	Definition	O	R
PK ANG	Angle where component stresses have peak values: 0 and 90/MODE degrees. Blank if MODE = 0.	Y	Y
XC, YC	Location where results are reported	Y	*3*
S:X, Y, Z	Direct stresses (radial, axial, hoop) at *PK ANG* locations	Y	Y
S:XY, YZ, XZ	Shear stresses (radial-axial, axial-hoop, radial-hoop) at *PK ANG* locations	Y	Y
S:1, 2, 3	Principal stresses at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*
S:INT	Stress intensity at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*
S:EQV	Equivalent stress at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*
EPEL:X, Y, Z, XY	Elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain [*4*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strain [*4*]	-	*1*
FACE	Face label	*2*	*2*
TEMP	Surface average temperature	*2*	*2*
EPEL(PAR, PER, Z, SH)	Surface strains (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	*2*	*2*
S(PAR, PER, Z, SH)	Surface stresses (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	*2*	*2*

1. These items are output only if KEYOPT(5) = 1.
2. These items are printed only if KEYOPT(6) is greater than zero.
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2, "PLANE25 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "PLANE25 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "PLANE25 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,K,L
: sequence number for data at nodes I,J,K,L

Table 2 PLANE25 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
THETA = 0					
S1	NMISC	1	16	31	46
S2	NMISC	2	17	32	47
S3	NMISC	3	18	33	48
SINT	NMISC	4	19	34	49
SEQV	NMISC	5	20	35	50
THETA = 90/MODE					
S1	NMISC	6	21	36	51
S2	NMISC	7	22	37	52
S3	NMISC	8	23	38	53
SINT	NMISC	9	24	39	54
SEQV	NMISC	10	25	40	55
EXTR Values					
S1	NMISC	11	26	41	56
S2	NMISC	12	27	42	57
S3	NMISC	13	28	43	58
SINT	NMISC	14	29	44	59
SEQV	NMISC	15	30	45	60

Note

The NMISC items (1 thru 60) in the above table represent the combined stress solution, KEYOPT(5) = 1. If MODE = 0, their values are zero at THETA = 90/MODE and at EXTR.

See *Section 2.2.2.5: Surface Solution* of this manual for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE25 Assumptions and Restrictions

- The area of the element must be positive.
- The element must be defined in the global X-Y plane as shown in *Figure 1, "PLANE25 Geometry"* and the global X-axis must be the radial direction. Negative X coordinates should not be used.
- The element assumes a linear elastic material. Post-analysis superposition of results is valid only with other linear elastic solutions. The element should not be used with the large deflection option.
- A triangular element may be formed by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). The extra shapes are automatically deleted for triangular elements so that a constant strain element results.
- Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met.

- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.
- Modeling hints: If shear effects are important in a shell-like structure, you should use at least two elements through the thickness.

PLANE25 Product Restrictions

There are no product-specific restrictions for this element.

MATRIX27

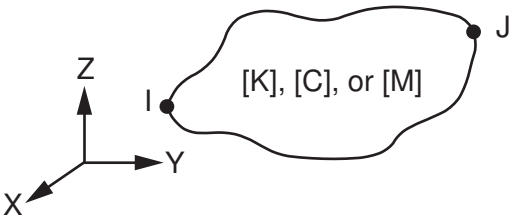
Stiffness, Damping, or Mass Matrix

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MATRIX27 Element Description

MATRIX27 represents an arbitrary element whose geometry is undefined but whose elastic kinematic response can be specified by stiffness, damping, or mass coefficients in matrix form. The matrix is assumed to relate two nodes, each with six degrees of freedom per node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. See MATRIX27 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other similar, but less general, elements are the spring-damper element (COMBIN14), and the mass element (MASS21).

Figure 1 MATRIX27 Schematic



MATRIX27 Input Data

The node locations and the coordinate system for this element are shown in *Figure 1, "MATRIX27 Schematic"*. The element is defined by two nodes and the matrix coefficients. The stiffness, damping, or mass matrix constants are input as real constants. The units of the stiffness constants are Force/Length or Force*Length/Radian and the damping constants, Force*Time/Length and Force*Length*Time/Radian. The mass constants should have units of Force*Time^2/Length or Force*Time^2*Length/Radian.

All matrices generated by this element are 12 by 12. The degrees of freedom are ordered as UX, UY, UZ, ROTX, ROTY, ROTZ for node I followed by the same for node J. If one node is not used, simply let all rows and columns relating to that node default to zero.

A structural matrix that combines the effects of many elements is normally positive or zero definite, as are the element matrices that contribute to it. There may be unusual circumstances where an element matrix is negative definite, and this is okay if there are other matrices connected to the same nodes that are positive definite, resulting in a final system of equations is still positive or zero definite. A simple example of such a circumstance is a beam element loaded with half of the buckling load. The stress stiffness matrix is negative definite, but the combined regular and stress stiffness matrix is positive definite.

When using MATRIX27 with symmetric element matrices (KEYOPT(2) = 0), positive or zero definite matrices should be input using KEYOPT(1) = 0, the default. If it is desired to input a negative definite element matrix, the user should set KEYOPT(1) = 1 so that the negative definite checking will be bypassed. However, when using an unsymmetric or skew-symmetric element matrix, (KEYOPT(2) = 2 or 3), there are no limits on the form of the matrix, as the unsymmetric solvers are designed to solve any system of equations, as long as it is not singular.

The matrix constants should be input according to the matrix diagrams shown in *MATRIX27 Output Data*. For example, if a simple spring of stiffness K in the nodal x direction is desired, the input constants would be C_1 = C_{58} = K and C_7 = -K for KEYOPT(2) = 0 and KEYOPT(3) = 4.

A summary of the element input is given in *MATRIX27 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

MATRIX27 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

C1, C2, ... C78 - Define the upper triangular portion of the symmetric matrix (if KEYOPT(2) = 0)
C1, C2, ... C144 - Define all terms of the unsymmetric matrix (if KEYOPT(2) = 2)
C1, C2, ... C66 - Define upper triangular portion (less diagonal terms) if skew symmetric (KEYOPT(2) = 3)

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features
Birth and death

KEYOPT(1)
Matrix form (only works with KEYOPT(2) = 0):

0 --
Input positive or zero definite matrices only

1 --
Input positive, zero, or negative definite matrices

KEYOPT(2)
Matrix formulation:

0 --
Symmetric matrices (78 constants required)

2 --
Unsymmetric matrices(144 constants required)

3 --
Skew symmetric matrices (66 constants required)

KEYOPT(3)
Real constant input data:

2 --
Defines a 12 x 12 mass matrix

4 --
Defines a 12 x 12 stiffness matrix

5 --
Defines a 12 x 12 damping matrix

KEYOPT(4)

Element matrix output:

0 --

Do not print element matrix

1 --

Print element matrix at beginning of solution phase

MATRIX27 Output Data

The solution output associated with the element consists of node displacements included in the overall nodal solution. There is no element solution output associated with the element unless element reaction forces and/or energies are requested. KEYOPT(4) = 1 causes the element matrix to be printed (for the first substep of the first load step only). *Section 2.2: Solution Output* gives a general description of solution output.

For KEYOPT(2) = 0, the symmetric matrix has the form:

$$\begin{bmatrix} C_1 & C_2 & C_3 & \cdots & C_{12} \\ & C_{13} & C_{14} & \cdots & C_{23} \\ & & C_{24} & \cdots & \cdots \\ & & & \ddots & \vdots \\ & & \text{Symmetric} & & C_{78} \end{bmatrix}$$

For KEYOPT(2) = 2, the unsymmetric matrix has the form:

$$\begin{bmatrix} C_1 & C_2 & C_3 & . & . & . & . & . & . & . & . & C_{12} \\ C_{13} & C_{14} & C_{15} & . & . & . & . & . & . & . & . & C_{24} \\ C_{25} & C_{26} & C_{27} & . & . & . & . & . & . & . & . & C_{36} \\ . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ \hline . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ . & . & . & . & . & . & . & . & . & . & . & . \\ C_{133} & C_{134} & C_{135} & . & . & . & . & . & . & . & . & C_{144} \end{bmatrix}$$

For KEYOPT(2) = 3, the skew symmetric matrix has the form:

$$\begin{bmatrix} 0 & C_1 & C_2 & C_3 & . & . & . & . & . & . & . & C_{11} \\ & 0 & C_{12} & C_{13} & . & . & . & . & . & . & . & C_{21} \\ & & 0 & C_{22} & . & . & . & . & . & . & . & C_{30} \\ & & & 0 & . & . & . & . & . & . & . & . \\ & & & & 0 & . & . & . & . & . & . & . \\ & & & & & 0 & . & . & . & . & . & . \\ \hline & & & & & & 0 & . & . & . & . & . \\ & & & & & & & 0 & . & . & . & . \\ & & & & & & & & 0 & . & . & . \\ & & & \text{Skew Symmetric} & & & & & & 0 & C_{64} & C_{65} \\ & & & & & & & & & & 0 & C_{66} \\ & & & & & & & & & & & 0 \end{bmatrix}$$

MATRIX27 Assumptions and Restrictions

- Nodes may be coincident or noncoincident.
- Since element matrices should normally not be negative definite, a note is printed for those cases where this can be easily detected.

- With a lumped mass matrix [**LUMPM**,ON] all off-diagonal terms must be zero.
- The matrix terms are associated with the nodal degrees of freedom and are assumed to act in the nodal coordinate directions (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*).

MATRIX27 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- Damping and unsymmetric matrices are not allowed.
- Real constants C79 through C144, for unsymmetric matrices, are not applicable.
- The birth and death special feature is not allowed.
- KEYOPT(2) can only be set to 0 (default). KEYOPT(3) = 5 is not allowed.
- The DAMP material property is not allowed.

SHELL28

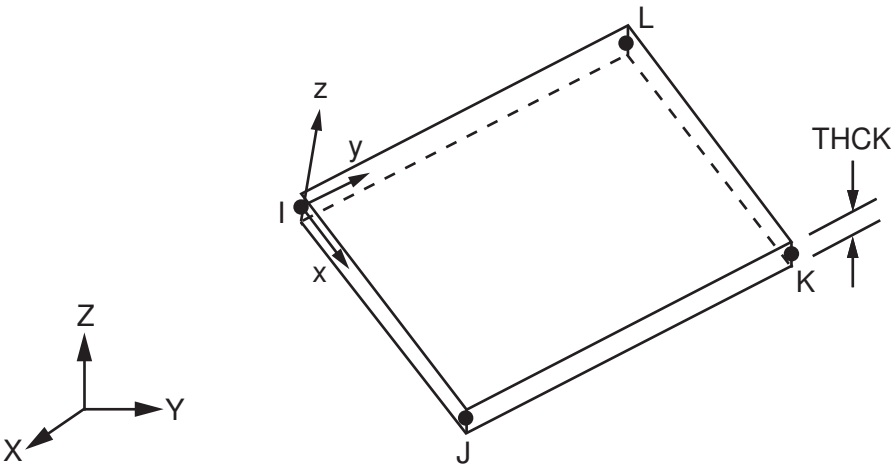
Shear/Twist Panel

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL28 Element Description

SHELL28 is used to carry shear load in a frame structure. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions, or rotations about the nodal x, y, and z axes. See SHELL28 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL28 Geometry



SHELL28 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL28 Geometry"*. The element is defined by four nodes, a thickness, and material properties. The only material properties actually used are GXY and DENS. GXY may be entered directly or calculated from EX and either NUXY or PRXY. Also, EX must be input, whether or not GXY is entered. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Real constant SULT is the ultimate shear stress used for the margin of safety calculation. ADMSUA is the added mass per unit area. KEYOPT(1) is used to select whether the element should be used as a shear panel or as a twist panel. Only the lumped mass matrix is available.

Element loads are described in *Section 2.8: Node and Element Loads*. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF. Temperatures are used only for material property evaluation.

A summary of the element input is given in *SHELL28 Input Summary*. *Section 2.1: Element Input* gives a general description of element input.

SHELL28 Input Summary

Nodes
I, J, K, L

Degrees of Freedom

UX, UY, UZ if KEYOPT(1) = 0
ROTX, ROTY, ROTZ if KEYOPT(1) = 1

Real Constants

THCK - Panel thickness
SULT - Ultimate shear stress
ADMSUA - Added mass/unit area

Material Properties
EX, PRXY (or NUXY), GXY, DENS, DAMP

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features
Stress stiffening

KEYOPT(1)
Element behavior:

0 --
Shear panel

1 --
Twist panel

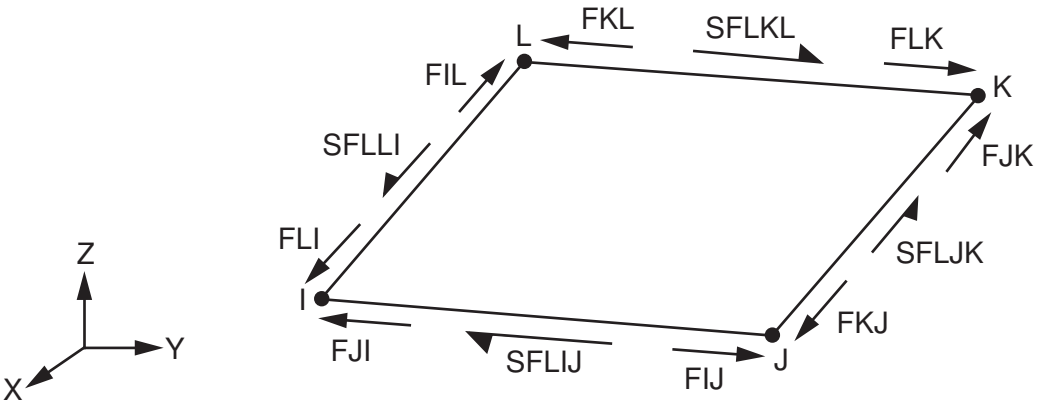
SHELL28 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL28 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL28 Stress Output"*. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL28 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL28 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
SXY	Average of four corner shear stresses	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
SXY(I,J,K,L)	Shear stresses at corner nodes	Y	Y
SXY(MAX)	Maximum of four corner shear stresses	Y	Y
SMARGN	Margin of safety on shear	Y	Y
FDIK, FDJL	Forces along diagonals I-K and J-L	*1*	*1*
FLI, FJI	Forces at node I from node L and node J	*1*	*1*
FIJ, FKJ	Forces at node J from node I and node K	*1*	*1*
FJK, FLK	Forces at node K from node J and node L	*1*	*1*
FKL, FIL	Forces at node L from node K and node I	*1*	*1*
SFLIJ	Shear flow on edge I - J	*1*	*1*
SFLJK	Shear flow on edge J - K	*1*	*1*
SFLKL	Shear flow on edge K - L	*1*	*1*
SFLLI	Shear flow on edge L - I	*1*	*1*
FZI	Z - Force at node I	*1*	*1*
FZJ	Z - Force at node J	*1*	*1*
FZK	Z - Force at node K	*1*	*1*
FZL	Z - Force at node L	*1*	*1*
MDIK, MDJL	Moments about diagonals I-K and J-L	*2*	*2*

1. The values are output only if KEYOPT(1) = 0
2. The values are output in place of FDIK and FDJL only if KEYOPT(1) = 1
3. Available only at centroid as a ***GET** item.

Table 2, "SHELL28 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL28 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL28 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 SHELL28 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FDIK (MDIK)	SMISC	1
FDJL (MDJL)	SMISC	2
FLI	SMISC	3
FJI	SMISC	4
FIJ	SMISC	5
FKJ	SMISC	6
FJK	SMISC	7
FLK	SMISC	8
FKL	SMISC	9
FIL	SMISC	10
FZI	SMISC	11
FZJ	SMISC	12
FZK	SMISC	13
FZL	SMISC	14
SXY	SMISC	15
SXYI	SMISC	16
SXYJ	SMISC	17
SXYK	SMISC	18
SXYL	SMISC	19
SXYMAX	SMISC	20
SMARGN	SMISC	21
SFLIJ	SMISC	22
SFLJK	SMISC	23
SFLKL	SMISC	24
SFLLI	SMISC	25

SHELL28 Assumptions and Restrictions

- Zero area elements are not allowed. This condition occurs most often when the elements are numbered improperly.
- This element is most often used with a latticework of beam or spar elements. If this element is used alone it is almost always unstable, because it carries only shear (and not tension or compression) loading.
- This element is not recommended for general use. Its use should be restricted to applications which have historically used such an element. For all other applications, you should use other shell elements such as SHELL41, SHELL43, SHELL63, SHELL181, and SHELL281, which automatically combine tension, compression, bending, shear, and twisting effects.

- This element is based on the premise of having only shear, but no normal stress along the edges. Since this is possible only for rectangles, you can expect the accuracy of the element to degrade if nonrectangular shapes are used.

SHELL28 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.

FLUID29

2-D Axisymmetric Harmonic Acoustic Fluid

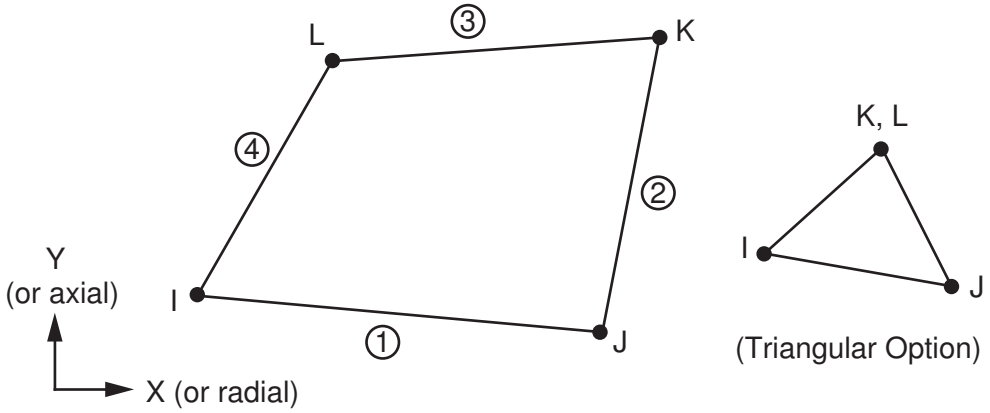
MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID29 Element Description

FLUID29 is used for modeling the fluid medium and the interface in fluid/structure interaction problems. Typical applications include sound wave propagation and submerged structure dynamics. The governing equation for acoustics, namely the 2-D wave equation, has been discretized taking into account the coupling of acoustic pressure and structural motion at the interface. The element has four corner nodes with three degrees of freedom per node: translations in the nodal x and y directions and pressure. The translations, however, are applicable only at nodes that are on the interface. Acceleration effects, such as in sloshing problems, may be included.

The element has the capability to include damping of sound absorbing material at the interface. The element can be used with other 2-D structural elements to perform unsymmetric or damped modal, full harmonic response and full transient method analyses (see the description of the **TRNOPT** command). When there is no structural motion, the element is also applicable to static, modal and reduced harmonic response analyses. See FLUID29 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID29 Geometry



FLUID29 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID29 Geometry"*. The element is defined by four nodes, the number of harmonic waves (MODE on the **MODE** command), the symmetry condition (ISYM on the **MODE** command), a reference pressure, and the isotropic material properties. The MODE and ISYM parameters are discussed in detail in *Section 2.13: Axisymmetric Elements with Non-axisymmetric Loads*. The reference pressure (PREF) is used to calculate the element sound pressure level (defaults to 20×10^{-6} N/m^2). The speed of sound ($\sqrt{k / \rho_o}$) in the fluid is input by SONC where k is the bulk modulus of the fluid (Force/Area) and ρ_o is the mean fluid density (Mass/Volume) (input as DENS). The dissipative effect due to fluid viscosity is neglected, but absorption of sound at the interface is accounted for by generating a damping matrix using the surface area and boundary admittance at the interface. Experimentally measured values of the boundary admittance for the sound absorbing material may be input as material property MU. We recommend MU values from 0.0 to 1.0; however, values greater than 1.0 are allowed. MU = 0.0 represents no sound absorption and MU = 1.0 represents full sound absorption. DENS, SONC and MU are evaluated at the average of the nodal temperatures.

Nodal flow rates, if any, may be specified using the **F** command where both the real and imaginary components may be applied. Nodal flow rates should be input per unit of depth for a plane analysis and on a 360° basis for an axisymmetric analysis.

Element loads are described in *Section 2.8: Node and Element Loads*. Fluid-structure interfaces (FSI) may be flagged by surface loads at the element faces as shown by the circled numbers on *Figure 1, "FLUID29 Geometry"*. Specifying the FSI label (without a value) [**SF**, **SFA**, **SFE**] will couple the structural motion and fluid pressure at the interface. Deleting the FSI specification [**SFDELE**, **SFADELE**, **SFEDELE**] removes the flag. The flag specification should be on the fluid elements at the interface. See Acoustics in the *Coupled-Field Analysis Guide* for more information on the use of the fluid-structure interaction flag. The surface load label IMPD with a value of unity should be used to include damping that may be present at a structural boundary with a sound absorption lining. A zero value of IMPD removes the damping calculation. The displacement degrees of freedom (UX and UY) at the element nodes not on the interface should be set to zero to avoid zero-pivot warning messages.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(2) is used to specify the absence of a structure at the interface and, therefore, the absence of coupling between the fluid and structure. Since the absence of coupling produces symmetric element matrices, a symmetric eigensolver [**MODOPT**] may be used within the modal analysis. However, for the coupled (unsymmetric) problem, a corresponding unsymmetric eigensolver [**MODOPT**] must be used.

Vertical acceleration (ACELY on the **ACEL** command) is needed for the gravity regardless of the value of MODE, even for a modal analysis.

A summary of the element input is given in *FLUID29 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

FLUID29 Input Summary

Nodes
    I, J, K, L

Degrees of Freedom

    UX, UY, PRES if KEYOPT (2) = 0
    PRES if KEYOPT (2) = 1

Real Constants
    PREF - Reference pressure

Material Properties
    DENS, SONC, MU

Surface Loads

    Fluid-structure Interface Flag --
        face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

    Impedance --
        face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

    Temperatures --
        T(I), T (J), T(K), T(L)

Mode Number
Input mode number on **MODE** command

Loading Condition
Input for ISYM on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

Special Features
None

KEYOPT(2)
Structure at element interface:

0 --
Structure present at interface (unsymmetric element matrix)

1 --
No structure at interface (symmetric element matrix)

KEYOPT(3)
Element behavior:

0 --
Planar

1 --
Axisymmetric

2 --
Axiharmonic

KEYOPT(7)
Free surface effect:

0 --
Do not include sloshing effect

1 --
Include sloshing effect on face of elements located on Y = 0.0 plane (elements must not have positive Y coordinates)

FLUID29 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pressures included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID29 Element Output Definitions"*.

Section 2.2: Solution Output gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID29 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
PRESSURE	Average pressure	Y	Y
PG(X, Y, SUM)	Pressure gradient components and vector sum	Y	Y
VL(X, Y, SUM)	Fluid velocity components and vector sum	*1*	*1*
SOUND PR.LEVEL	Sound pressure level (in decibels)	*1*	*1*

1. Output only if **ANTYPE**,HARMIC
2. Available only at centroid as a ***GET** item.

Table 2, "FLUID29 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "FLUID29 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID29 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID29 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PGX	SMISC	1
PGY	SMISC	2
VLX	SMISC	3
VLY	SMISC	4
PRESSURE	NMISC	1
PGSUM	NMISC	2
VLSUM	NMISC	3
SOUND PR. LEVEL	NMISC	4

FLUID29 Assumptions and Restrictions

- The area of the element must be positive.

- The element must lie in a global X-Y plane as shown in *Figure 1, "FLUID29 Geometry"*.
- All elements must have 4 nodes. A triangular element may be formed by defining duplicate K and L nodes (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- The acoustic pressure in the fluid medium is determined by the wave equation with the following assumptions:
  - The fluid is compressible (density changes due to pressure variations).
  - Inviscid fluid (no dissipative effect due to viscosity).
  - There is no mean flow of the fluid.
  - The mean density and pressure are uniform throughout the fluid. Note that the acoustic pressure is the excess pressure from the mean pressure.
  - Analyses are limited to relatively small acoustic pressures so that the changes in density are small compared with the mean density.

The lumped mass matrix formulation [**LUMPM**,ON] is not allowed for this element.

FLUID29 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Multiphysics

- KEYOPT(3) = 2 is valid only in ANSYS Multiphysics
- KEYOPT(7) is valid only in ANSYS Multiphysics

FLUID30

3-D Acoustic Fluid

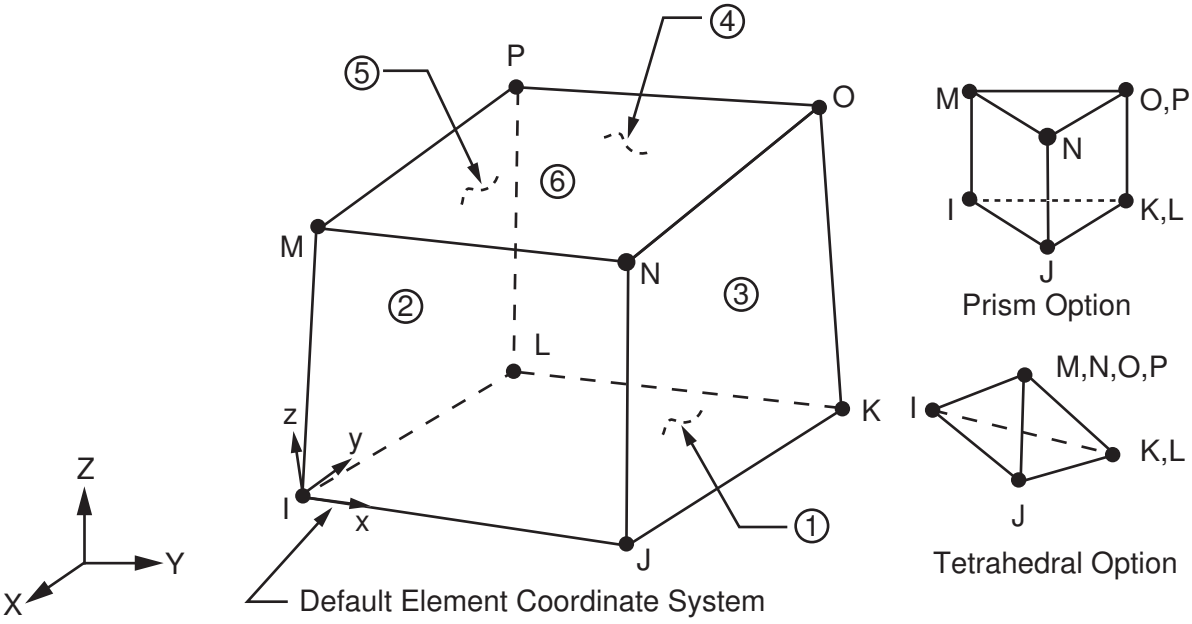
MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID30 Element Description

FLUID30 is used for modeling the fluid medium and the interface in fluid/structure interaction problems. Typical applications include sound wave propagation and submerged structure dynamics. The governing equation for acoustics, namely the 3-D wave equation, has been discretized taking into account the coupling of acoustic pressure and structural motion at the interface. The element has eight corner nodes with four degrees of freedom per node: translations in the nodal x, y and z directions and pressure. The translations, however, are applicable only at nodes that are on the interface. Acceleration effects, such as in sloshing problems, may be included.

The element has the capability to include damping of sound absorbing material at the interface. The element can be used with other 3-D structural elements to perform unsymmetric or damped modal, full harmonic response and full transient method analyses (see the description of the **TRNOPT** command). When there is no structural motion, the element is also applicable to static, modal and reduced harmonic response analyses. See FLUID30 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID30 Geometry



FLUID30 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID30 Geometry"*. The element is defined by eight nodes, a reference pressure, and the isotropic material properties. The reference pressure (PREF) is used to calculate the element sound pressure level (defaults to 20×10^{-6} N/m^2). The speed of sound ($\sqrt{k/\rho_o}$) in the fluid is input by SONC where k is the bulk modulus of the fluid (Force/Area) and ρ_o is the mean fluid density (Mass/Volume) (input as DENS). The dissipative effect due to fluid viscosity is neglected, but absorption of sound at the interface is accounted for by generating a damping matrix using the surface area and boundary admittance at the interface. Experimentally measured values of the boundary admittance for the sound absorbing material may be input as material property MU. We recommend MU values from 0.0 to 1.0;

however, values greater than 1.0 are allowed. MU = 0.0 represents no sound absorption and MU = 1.0 represents full sound absorption. DENS, SONC and MU are evaluated at the average of the nodal temperatures.

Nodal flow rates may be specified using the **F** command where both the real and imaginary components may be applied.

Element loads are described in *Section 2.8: Node and Element Loads*. Fluid-structure interfaces (FSI) may be flagged by surface loads at the element faces as shown by the circled numbers on *Figure 1, "FLUID30 Geometry"*. Specifying the FSI label (without a value) [**SF**, **SFA**, **SFE**] will couple the structural motion and fluid pressure at the interface. Deleting the FSI specification [**SFDELE**, **SFADELE**, **SFEDELE**] removes the flag. The flag specification should be on the fluid elements at the interface. See Acoustics in the *Coupled-Field Analysis Guide* for more information on the use of the fluid-structure interaction flag. The surface load label IMPD with a value of unity should be used to include damping that may be present at a structural boundary with a sound absorption lining. A zero value of IMPD removes the damping calculation. The displacement degrees of freedom (UX, UY and UZ) at the element nodes not on the interface should be set to zero to avoid zero-pivot warning messages.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(2) is used to specify the absence of a structure at the interface and, therefore, the absence of coupling between the fluid and structure. Since the absence of coupling produces symmetric element matrices, a symmetric eigensolver [**MODOPT**] may be used within the modal analysis. However, for the coupled (unsymmetric) problem, a corresponding unsymmetric eigensolver [**MODOPT**] must be used.

Vertical acceleration (ACELZ on the **ACEL** command) is needed for the gravity, even for a modal analysis.

A summary of the element input is given in *FLUID30 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

FLUID30 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, PRES if KEYOPT (2) = 0
PRES if KEYOPT (2) = 1

Real Constants
PREF - Reference pressure

Material Properties
DENS, SONC, MU

Surface Loads
Fluid-structure interface flag:

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Impedance:

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T (I), T (J), T (K), T (L), T (M), T (N), T(O), T(P)

Special Features
None

KEYOPT(2)
Structure at element interface:

0 --
Structure present at interface (unsymmetric element matrix)

1 --
No structure at the interface (symmetric element matrix)

KEYOPT(7)
Free surface effect:

0 --
Do not include sloshing effect

1 --
Include sloshing effect on face of elements located on Z = 0.0 plane (elements must not have positive Z coordinates)

FLUID30 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pressures included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID30 Element Output Definitions"*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID30 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
PRESSURE	Average pressure	Y	Y

Name	Definition	O	R
PG(X,Y,Z,SUM)	Pressure gradient components and vector sum	Y	Y
VL(X,Y,Z,SUM)	Fluid velocity components and vector sum	*1*	*1*
SOUND PR. LEVEL	Sound pressure level (in decibels)	*1*	*1*

1. Output only if **ANTYPE**,HARMIC
2. Available only at centroid as a ***GET** item.

Table 2, "FLUID30 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "FLUID30 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID30 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID30 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PGX	SMISC	1
PGY	SMISC	2
PGZ	SMISC	3
VLX	SMISC	4
VLY	SMISC	5
VLZ	SMISC	6
PRESSURE	NMISC	1
PGSUM	NMISC	2
VLSUM	NMISC	3
SOUND PR. LEVEL	NMISC	4

FLUID30 Assumptions and Restrictions

- The element must not have a zero volume.
- Element nodes may be numbered either as shown in *Figure 1, "FLUID30 Geometry"* or may have planes IJKL and MNOP interchanged.
- The element may not be twisted such that it has two separate volumes. This occurs usually when the element nodes are not in the correct sequence.
- All elements must have 8 nodes. A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P nodes (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). A tetrahedron shape is also available.
- We do not recommend using degenerate shapes with KEYOPT(7) = 1.

- The acoustic pressure in the fluid medium is determined by the wave equation with the following assumptions:
  - The fluid is compressible (density changes due to pressure variations).
  - Inviscid fluid (no dissipative effect due to viscosity).
  - There is no mean flow of the fluid.
  - The mean density and pressure are uniform throughout the fluid. Note that the acoustic pressure is the excess pressure from the mean pressure.
  - Analyses are limited to relatively small acoustic pressures so that the changes in density are small compared with the mean density.
- The lumped mass matrix formulation [**LUMPM**,ON] is not allowed for this element.

FLUID30 Product Restrictions

ANSYS Multiphysics

- KEYOPT(7) is valid only in ANSYS Multiphysics

LINK31

Radiation Link

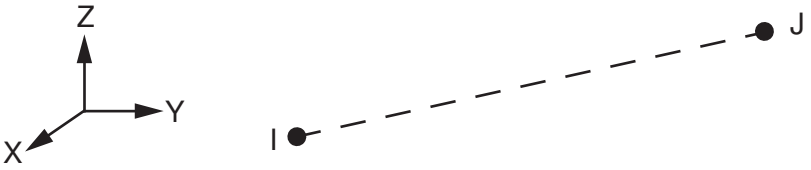
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

LINK31 Element Description

LINK31 is a uniaxial element which models the radiation heat flow rate between two points in space. The link has a single degree of freedom, temperature, at each node. The radiation element is applicable to a 2-D (plane or axisymmetric) or 3-D, steady-state or transient thermal analysis.

An empirical relationship allowing the form factor and area to multiply the temperatures independently is also available. The emissivity may be temperature dependent. If the model containing the radiation element is also to be analyzed structurally, the radiation element should be replaced by an equivalent (or null) structural element. See LINK31 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK31 Geometry



LINK31 Input Data

The geometry, node locations, and the coordinate system for this radiation element are shown in *Figure 1, "LINK31 Geometry"*. The element is defined by two nodes, a radiating surface area, a geometric form factor, the emissivity, and the Stefan-Boltzmann constant (SBC). For axisymmetric problems, the radiation area should be input on a full 360° basis.

The emissivity may be constant or temperature (absolute) dependent. If it is constant, the value is input as a real constant. If it is temperature dependent, the values are input for the material property EMIS and the real constant value is used only to identify the material property number. In this case the MAT value associated with element is not used. EMIS defaults to 1.0.

The standard radiation function is defined as follows:

$$q = \sigma \varepsilon FA(T(I)^4 - T(J)^4)$$

where:

σ = Stefan-Boltzmann Constant (SBC)
(defaults to 0.119×10^{-10} (BTU/Hr*in^{2}* °R^{4})
ε = emissivity
F = geometric form factor
A = area (Length)^2
q = heat flow rate (Heat/Time)

The nonlinear temperature equation is solved by a Newton-Raphson iterative solution based on the form:

$$[(T(I)^2 + T(J)^2)(T(I) + T(J))]_p(T(I) - T(J))$$

where the $[\ ]_p$ term is evaluated at the temperature of the previous substep. The initial temperature should be near the anticipated solution and should not be zero (i.e., both TUNIF and TOFFST should not be zero).

An empirical radiation function of the following form may also be selected with KEYOPT(3):

$$q = \sigma \varepsilon (FT(I)^4 - AT(J)^4)$$

where F and A are arbitrary input constants.

A summary of the element input is given in *LINK31 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK31 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants

AREA - Radiating surface area
FORM FACTOR - Geometric form factor
EMISSIVITY - Emissivity (If EMISSIVITY = -*n*, use material *n* for emissivity vs. temperature definition)
SBC - Stefan-Boltzmann constant

Material Properties
EMIS (required only if EMISSIVITY = -N)

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Birth and death

KEYOPT(3)
Radiation equation:

0 --
Use standard radiation equation

1 --
Use empirical radiation equation

Note

The Stefan-Boltzmann constant (SBC) defaults to 0.1190E-10 with units of Btu, hr, in, °R (or °F if **TOFFST** is used)

LINK31 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution

- Additional element output as shown in *Table 1, "LINK31 Element Output Definitions"*

The heat flow rate is positive from node I to node J. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK31 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
AREA	AREA	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
EMIS(I, J)	Emissivity - I, J	Y	Y
TEMP(I, J)	Temperatures - I, J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "LINK31 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "LINK31 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "LINK31 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 LINK31 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1
TEMPI	SMISC	2
TEMPJ	SMISC	3
EMISI	NMISC	1
EMISJ	NMISC	2

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
AREA	NMISC	3
FORM FACTOR	NMISC	4

LINK31 Assumptions and Restrictions

- If the default Stefan-Boltzmann constant is used, the units associated with this element are Btu, inches, hours and °R (or °F + TOFFST). Other data input for this analysis must be consistent with this set of units or an appropriate conversion factor should be included in the radiation element's real constants.
- Nodes may or may not be coincident.
- An iterative solution is required with this element.

LINK31 Product Restrictions

There are no product-specific restrictions for this element.

LINK32

2-D Conduction Bar

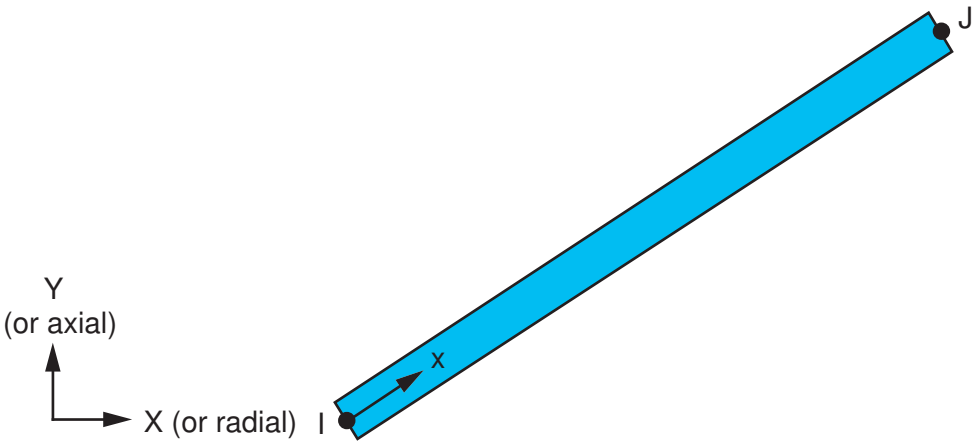
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

LINK32 Element Description

LINK32 is a uniaxial element with the ability to conduct heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The conducting bar is applicable to a 2-D (plane or axisymmetric), steady-state or transient thermal analysis.

If the model containing the conducting bar element is also to be analyzed structurally, the bar element should be replaced by an equivalent structural element. See LINK32 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK32 Geometry



LINK32 Input Data

The geometry, node locations, and the coordinate system for this conducting bar element are shown in *Figure 1, "LINK32 Geometry"*. The element is defined by two nodes, a cross-sectional area, and the material properties. For an axisymmetric analysis the area must be defined on a full 360° basis. Specific heat and density are ignored for steady-state solutions. The thermal conductivity is in the element longitudinal direction. The element x-axis extends from node I to node J.

Element loads are described in *Section 2.8: Node and Element Loads*. Heat generation rates may be input as element body loads at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

A summary of the element input is given in *LINK32 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK32 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants
AREA - Cross-sectional area

Material Properties
KXX, DENS, C, ENTH

Surface Loads
None

Body Loads

Heat Generation --
HG(I), HG(J)

Special Features
Birth and death

KEYOPTS
None

LINK32 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK32 Element Output Definitions"*

The heat flow rate is in units of Heat/Time and is positive from node I to node J. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK32 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*1*
LENGTH	Length	Y	Y
AREA	Input area	Y	Y
TEMP(I, J)	Temperatures - I, J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y
THERMAL FLUX	Thermal flux (heat flow rate/cross-sectional area)	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "LINK32 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "LINK32 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "LINK32 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 LINK32 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1
TEMPI	SMISC	2
TEMPJ	SMISC	3
THERMAL FLUX	SMISC	4
LENGTH	NMISC	1
AREA	NMISC	2

LINK32 Assumptions and Restrictions

- Heat is assumed to flow only in the longitudinal element direction.
- The element must be in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- The element must not have a zero length, so nodes I and J must not be coincident.
- A free end of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

LINK32 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

LINK33

3-D Conduction Bar

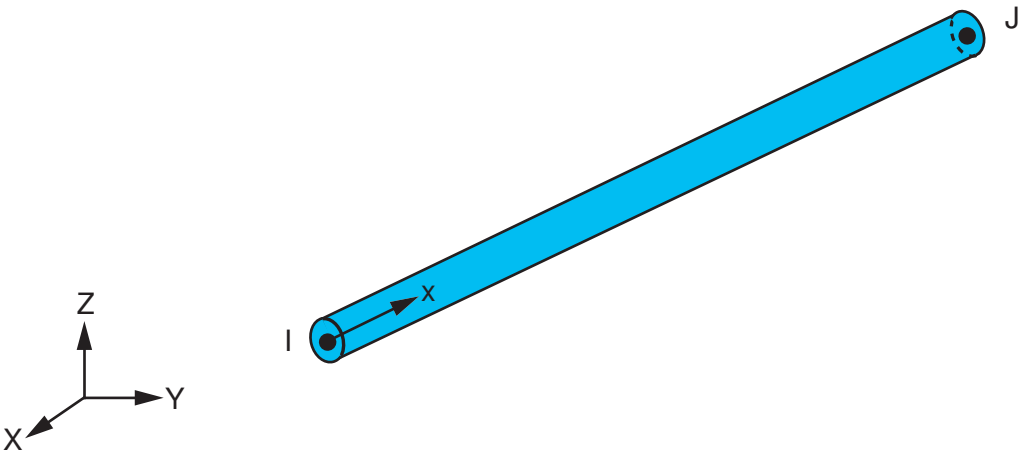
MP ME <> PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

LINK33 Element Description

LINK33 is a uniaxial element with the ability to conduct heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The conducting bar is applicable to a steady-state or transient thermal analysis.

If the model containing the conducting bar element is also to be analyzed structurally, the bar element should be replaced by an equivalent structural element. See LINK33 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK33 Geometry



LINK33 Input Data

The geometry, node locations, and the coordinate system for this conducting bar are shown in *Figure 1, "LINK33 Geometry"*. The element is defined by two nodes, a cross-sectional area, and the material properties. Specific heat and density are ignored for steady-state solutions. The thermal conductivity is in the element longitudinal direction.

Element loads are described in *Section 2.8: Node and Element Loads*. Heat generation rates may be input as element body loads at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

A summary of the element input is given in *LINK33 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK33 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants
AREA - Cross-sectional area

Material Properties
KXX, DENS, C, ENTH

Surface Loads
None

Body Loads

Heat Generation --
HG(I), HG(J)

Special Features
Birth and death

KEYOPTS
None

LINK33 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK33 Element Output Definitions"*

The heat flow rate is in units of Heat/Time and is positive from node I to node J. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK33 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
LENGTH	Length	Y	Y
AREA	Input area	Y	Y
TEMP(I, J)	Temperatures - I, J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y
THERMAL FLUX	Thermal flux (heat flow rate/cross-sectional area)	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "LINK33 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "LINK33 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "LINK33 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 LINK33 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1
TEMPI	SMISC	2
TEMPJ	SMISC	3
THERMAL FLUX	SMISC	4
LENGTH	NMISC	1
AREA	NMISC	2

LINK33 Assumptions and Restrictions

- Heat is assumed to flow only in the longitudinal element direction.
- The element must not have a zero length, so nodes I and J must not be coincident.
- A free end of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

LINK33 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

LINK34

Convection Link

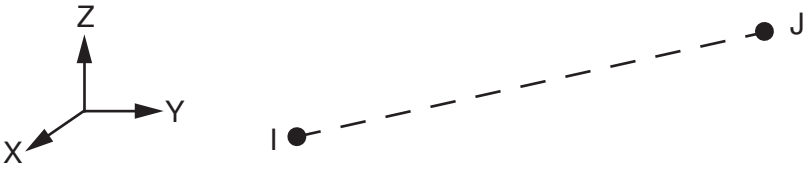
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

LINK34 Element Description

LINK34 is a uniaxial element with the ability to convect heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The convection element is applicable to a 2-D (plane or axisymmetric) or 3-D, steady-state or transient thermal analysis.

If the model containing the convection element is also to be analyzed structurally, the convection element should be replaced by an equivalent (or null) structural element. The element may have a nonlinear film coefficient which may also be a function of temperature or time. See LINK34 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK34 Geometry



LINK34 Input Data

The geometry and node locations for this convection element are shown in *Figure 1, "LINK34 Geometry"*. The element is defined by two nodes, a convection surface area, two empirical terms, and a film coefficient. In an axisymmetric analysis the convection area must be expressed on a full 360° basis. The empirical terms n (input as EN) and CC determine the form of the convection equation in conjunction with KEYOPT(3).

The convection function is defined as follows:

$$q = h_f*A*E*(T(I) - T(J))$$

where:

q = heat flow rate (Heat/Time)
h_f = film coefficient (Heat/Length^2*Time*Deg)
A = area (Length^2)
T = temperature (this substep) (Deg)
E = empirical convection term = $F*IT_p(I) - T_p(J)I^n + CC/h_f$
T_p = temperature (previous substep) (Deg)
n = empirical coefficient (EN)
CC = input constant

Note

E = F if n and CC = 0.0. F = 1.0 unless KEYOPT(3) = 2. If KEYOPT(3) = 3, E equals the larger of $IT_p(I) - T_p(J)I^n$ or CC/h_f.

A special option obtained with KEYOPT(3) = 2 allows an alternate input for h_f and an input scale factor (F). This option uses the *VAL1* field of the **SFE** command with *KVAL* = 0 for the h_f value and *KVAL* = 2 for the F value. If the

h_f value is zero (or blank), the HF material property is used for h_f. If the F value is zero (or blank) or negative, a value of 1.0 is assumed for F. Note, the F value input in this field will ramp within a load step if KBC = 0. An **SFE** command must be included (even if the values are left blank) for all LINK34 elements having KEYOPT(3) = 2.

Heat generation rates may be input as element body loads at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

A summary of the element input is given in *LINK34 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK34 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants

AREA - Convection surface area
EN - Empirical coefficient
CC - Input constant

Material Properties
HF

Surface Loads

Convections --
1 - Alternate input of HF and F if KEYOPT(3) = 2 (see text above)

Body Loads

Heat Generation --
HG(I), HG(J)

Special Features

Nonlinear if real constant EN is not equal to zero or if KEYOPT(3) = 3
Birth and death

KEYOPT(2)
Evaluation of film coefficient:

0 --
Use average of T(I) and T(J) to evaluate HF

1 --
Use greater of T(I) or T(J) to evaluate HF

2 --
Use lesser of T(I) or T(J) to evaluate HF

3 --
Use differential |T(I) - T(J)| to evaluate HF

KEYOPT(3)
Film coefficient and scale factor:

0 --
Standard element input and empirical term

2 --
Use alternate input for HF and F (input with **SFE** command)

3 --
Use discontinuous empirical term

LINK34 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK34 Element Output Definitions"*

The heat flow rate is in units of Heat/Time and is positive from node I to node J. In an axisymmetric analysis, the heat flow is on a full 360° basis. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK34 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC	Location where results are reported	Y	*1*
H	Film coefficient (includes empirical term)	Y	Y
AREA	Input area	Y	Y
TEMP	Temperature at node I and node J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "LINK34 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "LINK34 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "LINK34 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 2 LINK34 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
HEAT RATE	SMISC	1	-	-
TEMP	SMISC	-	2	3
H	NMISC	1	-	-
AREA	NMISC	2	-	-

LINK34 Assumptions and Restrictions

- If $T_p(I) = T_p(J)$ and n are nonzero, the first term of E is defined to be zero.
- Since all unspecified nodal temperatures are initially set to the uniform temperature, a nonzero value of n may result in no heat flowing through the element in the first substep of a thermal solution.
- Nodes may or may not be coincident.
- The element is nonlinear if n is nonzero or KEYOPT(3) = 3. However, the solver always assumes the element is nonlinear and, therefore, always performs an iterative solution. (Only 2 iterations are performed if the element is linear.)

LINK34 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

PLANE35

2-D 6-Node Triangular Thermal Solid

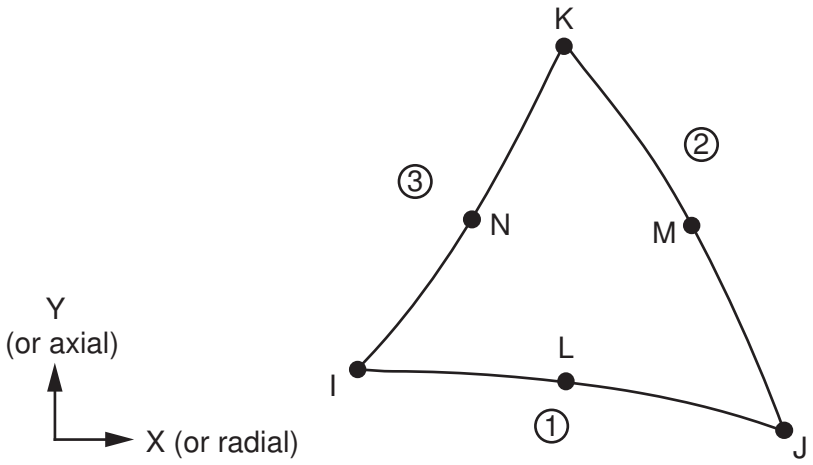
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

PLANE35 Element Description

PLANE35 is a 6-node triangular element compatible with the 8-node PLANE77 element. The triangular shape makes it well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node.

The 6-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis. If the model containing this element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE183). The element may be used as a plane element or as an axisymmetric ring element. See PLANE35 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE35 Geometry



PLANE35 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE35 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "PLANE35 Geometry"*. Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes. An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A summary of the element input is given in *PLANE35 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE35 Input Summary

Nodes
I, J, K, L, M, N

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (I-K)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

PLANE35 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE35 Element Output Definitions"*

For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE35 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	*1*
HFILM	Film coefficient	*1*	*1*
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFLUX	Heat flux at each node of face	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*

1. If a surface load has been input
2. Available only at centroid as a ***GET** item.

Table 2, "PLANE35 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "PLANE35 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "PLANE35 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

FC*N*
: sequence number for solution items for element Face *N*

Table 2 PLANE35 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	FC1	FC2	FC3
AREA	NMISC	1	7	13
HFAVG	NMISC	2	8	14
TAVG	NMISC	3	9	15
TBAVG	NMISC	4	10	16
HEAT RATE	NMISC	5	11	17
HFLXAVG	NMISC	6	12	18

PLANE35 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in an X-Y plane as shown in *Figure 1, "PLANE35 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will require a fine mesh at the surface.

PLANE35 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

SOURC36

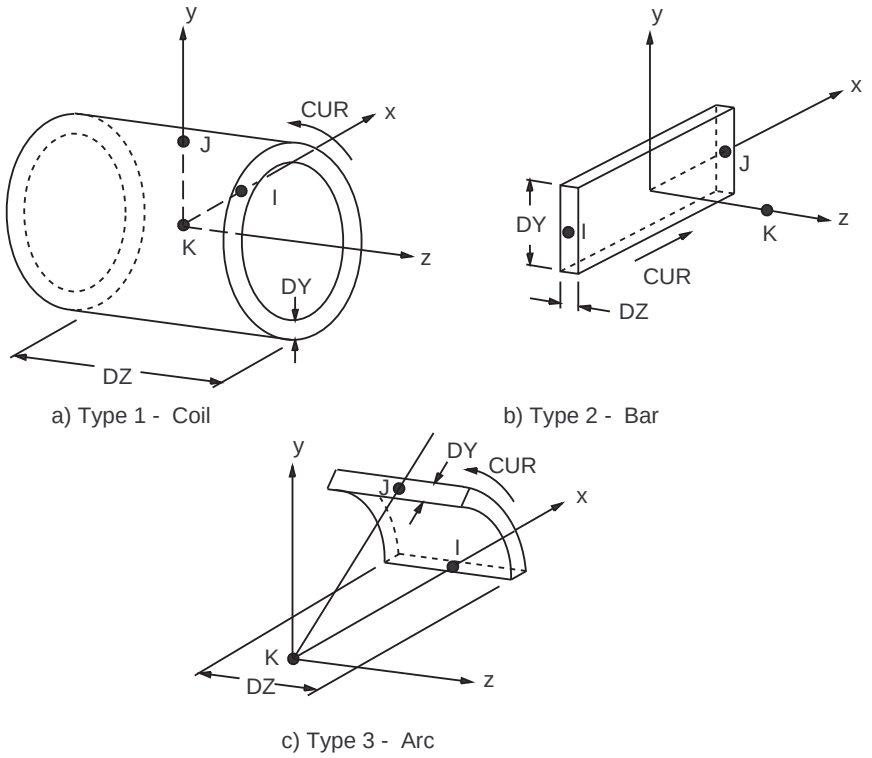
Current Source

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOURC36 Element Description

SOURC36 is a primitive (consisting of predefined geometries) used to supply current source data to magnetic field problems. The element represents a distribution of current in a model employing a scalar potential formulation (degree of freedom MAG) or an edge-based formulation. The currents are used to calculate a source magnetic field intensity (H_s) using a numerical integration technique involving the Biot-Savart law. The H_s term is used in the formulation as a magnetic load on the model. See SOURC36 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOURC36 Geometry



SOURC36 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOURC36 Geometry"*. The element input data includes three nodes and the following real constants (see *SOURC36 Input Summary*):

TYPE
Source type - use 1 for Coil, 2 for Bar, 3 for Arc.

CUR
Total current flowing through source (i.e., number of turns times current per turn).

DY
Characteristic y dimension for source type.

DZ
Characteristic z dimension for source type.

EPS
Convergence criterion for source field (H_s) calculations for arc and coils. Defaults to 0.001. EPS represents the relative maximum difference in the field H_s calculated at any node during the iterative calculation of the source field. EPS does not apply for bar sources.

Characteristic dimensions described above are in the element coordinate system. In the case of circular sources (coils, arcs) the radius is determined from the first and third nodes (I, K). For bar sources, the length is determined from the first two nodes (I, J).

As a modeling aid, a magnetic command macro, **RACE**, is available within the ANSYS command set. This macro enables the user to build a racetrack conductor from SOURC36 primitives. The macro is discussed in further detail in the *Commands Reference* and in the *Low-Frequency Electromagnetic Analysis Guide*.

A summary of the element input is given in *SOURC36 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOURC36 Input Summary

Nodes
I, J, K (nodes I, J and K define the characteristic length, current flow direction, and orient the source)

Degrees of Freedom
None

Real Constants

TYPE, CUR, DY, DZ, (Blank), (Blank),
(Blank), (Blank), EPS
See *Table 1, "SOURC36 Real Constants"* for a description of the real constants.

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

Table 1 SOURC36 Real Constants

No.	Name	Description
1	TYPE	Source type
2	CUR	Total current through source
3	DY	Characteristic Y dimension
4	DZ	Characteristic Z dimension

No.	Name	Description
5 ... 8	(Blank)	
9	EPS	Convergence criteria for H_s calculations

As a modeling aid, a magnetic command macro, **RACE**, is available within the ANSYS command set. This macro enables the user to build a racetrack conductor from SOURCE36 primitives. The macro is discussed in further detail in the *Commands Reference* and in the *Low-Frequency Electromagnetic Analysis Guide*.

SOURC36 Output Data

The source element has no output of its own since it is used only to supply current source data to magnetic field problems.

SOURC36 Assumptions and Restrictions

- The source element must have characteristic DY or DZ values that are greater than zero.
- The third node must not be colinear with the first two nodes.
- The nodes for this element need not be attached to any other elements.
- For the coil and the arc (types 1 and 3), the K-I line determines the radius (and the x axis) and the J node orients the x-y plane.
- For the arc (type 3) the subtended angle must be less than 180°. When you specify an arc using three points, ANSYS will always use the angle that is less than 180°.
- All source element nodes should be located a least 1E-6 units apart.
- Source element cannot have a zero inside radius (Radius $\neq$ DY/2 for types 1 and 3).
- The EPS convergence criterion is a measure of the relative difference in the calculated H_s field used during an iterative numerical integration procedure for coil and arc source primitives. The default value (.001) provides for good accuracy in regions outside of the source primitive location. For highly accurate calculations within the source primitive domain, the criteria may have to be tightened (i.e., a factor of 20 increase would be represented by EPS = .00005).
- Tightening the convergence criteria will significantly increase the solution run time.
- Users concerned with accurate calculations within the coil and arc source primitive domain should experiment with the criteria until satisfied with the degree of accuracy obtained.
- All currents for a magnetostatic model employing the scalar potential formulation must be specified. Whereas symmetry conditions on the finite element model may be employed, no symmetry may be employed on the current source elements.
- When used with SOLID117, the SOURC36 elements must be inside the SOLID117 finite element domain.

SOURC36 Product Restrictions

There are no product-specific restrictions for this element.

COMBIN37

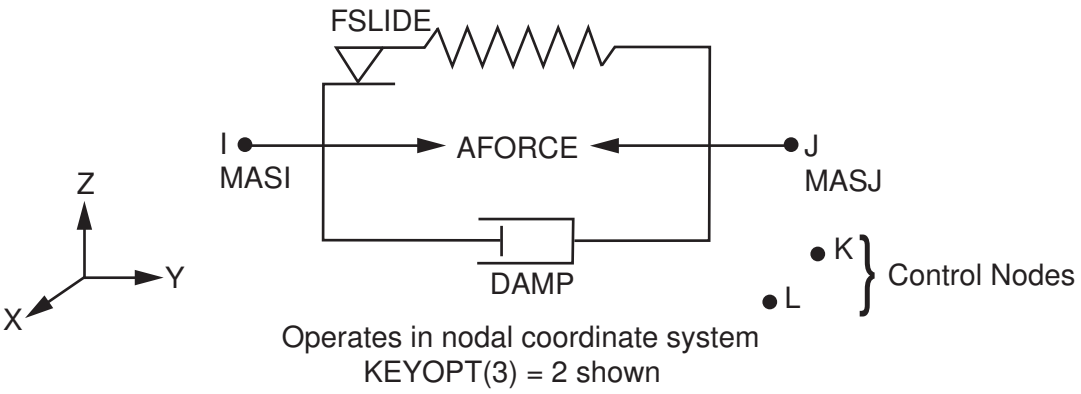
Control

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

COMBIN37 Element Description

COMBIN37 is a unidirectional element with the capability of turning on and off during an analysis. The element has one degree of freedom at each node, either a translation in a nodal coordinate direction, rotation about a nodal coordinate axis, pressure, or temperature. A control element with more capabilities (six degrees of freedom and large deflection) is described in COMBIN7. Similar unidirectional elements (without remote control capability) are COMBIN14, COMBIN39, and COMBIN40. The element has many applications, such as controlling heat flow as a function of temperature (thermostat), controlling damping as a function of velocity (mechanical snubber), controlling flow resistance as a function of pressure (relief valve), controlling friction as a function of displacement (friction clutch), etc. See COMBIN37 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 COMBIN37 Geometry



COMBIN37 Input Data

The functioning of this element is shown in *Figure 1, "COMBIN37 Geometry"*. The element is defined by two pairs of nodes, these being active nodes (I, J) and optional control nodes (K, L). Generally in the cases using UX, UY, or UZ as the active degrees of freedom, it is recommended to have the active nodes be coincident as this eliminates the possibility of moment disequilibrium. However, for visualization purposes, it may be useful to give node J a slightly greater coordinate value than node I. The element is defined such that a positive displacement of node J relative to node I will stretch the spring. Thus, if nodes I and J are interchanged, the same nodal motions will compress the spring.

Certain parameters associated with the control nodes are used to determine whether the control element is part of the structure (on) or not (off) and, thus, can be used to disconnect regions of the model during time dependent or iterative analyses. Other input values are stiffness (STIF), damping coefficient (DAMP), concentrated nodal masses (MASI, MASJ), on/off control values (ONVAL, OFFVAL), element load (AFORCE: positive pulls node I in the positive nodal coordinate direction, and pulls node J in the negative nodal coordinate direction), initial on/off element status (START: -1 if explicitly off, 0 if determined from starting value of control parameter, 1 if explicitly on), several nonlinear constants (C1, C2, C3, C4), and a limiting sliding force (FSLIDE).

The FSLIDE value represents the absolute value of the spring force that must be exceeded before sliding occurs. If FSLIDE is 0.0, the sliding capability of the element is removed, that is, a rigid connection is assumed. For structural analyses, units are force/length or moment/rotation for stiffness, force*time/length or moment*time/ro-

tation for damping, force*time^2/length or moment*time^2/rotation for mass, and force or moment for element load. For thermal analyses with temperature degrees of freedom, stiffness represents conductance and has units of heat/time*degrees, mass represents thermal capacitance with units of heat/degrees, and element load represents heat flow with units of heat/time. Also, in analyses with pressure degrees of freedom, stiffness represents flow conductance with units of length^2/time. Stiffness, damping, mass, and element load should be defined on a full 360° basis for axisymmetric analyses.

The active nodes (I, J) have only one degree of freedom each, selected with the KEYOPT(3) option. The control nodes (K, L) can have the same, or a different, degree of freedom as specified with KEYOPT(2). The KEYOPT(1) option assigns to the parameters of the control nodes either the value of the degree of freedom, the first or second derivative of the value, the integral of the value, or time, for example:

$$CPAR = UX_K - UX_L$$
$$CPAR = d(T_K - T_L)/dt$$
$$CPAR = d^2(ROTZ_K - ROTZ_L)/dt^2$$
$$CPAR = \int_o^t (UY_K - UY_L)dt$$
$$CPAR = t$$

Control nodes need not be connected to any other element. If node L is not defined, the control parameter is based only upon node K. If time is the control parameter (KEYOPT(1)), control nodes K and L need not be defined.

When the element is active and used in structural analyses, the element acts like any other spring/damper/mass element (such as COMBIN14, MASS21, and COMBIN40). In addition, the element can exhibit nonlinear behavior according to the function: RVMOD = RVAL + C1|CPAR|^{C2} + C3|CPAR|^{C4}, where RVMOD is the modified value of an input real constant value RVAL (identified by KEYOPT(6)), C1 through C4 are other real constants, and CPAR is the control parameter (see KEYOPT(1)). RVMOD may also be defined by user subroutine USERRC and is accessed by KEYOPT(9) = 1. Note, FSLIDE modified to a negative value is set to zero. In a field analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

As illustrated in *Figure 2, "COMBIN37 Behavior as a Function of Control Parameter"*, the KEYOPT(4) and KEYOPT(5) options, when used in combination with ONVAL and OFFVAL, set the control behavior of the element. The element is either on or off depending on the position of the control parameter with respect to the values of ONVAL and OFFVAL. Also, note that when KEYOPT(4) = 0 and the control parameter (CPAR) is within the ONVAL/OFFVAL interval, the element's status depends on the direction of the CPAR (i.e., on going from on to off, and vice-versa). If ONVAL = OFFVAL = 0.0 (or blank), the on/off capability is ignored and the element is always active.

A summary of the element input is given in *COMBIN37 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBIN37 Input Summary

Nodes

I, J, K, L (or I, J, K or I, J)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ, PRESS, or TEMP (depending on KEYOPT(2) and KEYOPT (3) below)

Real Constants

STIF, DAMP, MASJ, ONVAL, OFFVAL, AFORCE,
MASI, START, C1, C2, C3, C4,
FSLIDE

See *Table 1, "COMBIN37 Real Constants"* for a description of the real constants

Note

The DAMP real constant represents the damping coefficient for the damper component of the element, and should not be confused with the DAMP material property listed below.

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Adaptive descent

KEYOPT(1)
Control parameter:

0, 1 --
Control on value (UK-UL) (or UK if L not defined)

2 --
Control on first derivative of value with respect to time

3 --
Control on second derivative of value with respect to time

4 --
Control on integral of value with respect to time (zero initial condition assumed)

5 --
Control on time value (KEYOPT(2) and nodes K and L ignored)

KEYOPT(2)
Degree of freedom for control nodes (K and L):

N --
Use degree of freedom N as listed for KEYOPT(3) (defaults to KEYOPT(3))

KEYOPT(3)
Degree of freedom for active nodes (I and J):

0, 1 --
UX (Displacement along nodal X axes)

2 --
UY (along nodal Y)

3 --
UZ (along nodal Z)

4 --
ROTX (rotation about nodal X axes)

5 --
ROTY (about nodal Y)

6 --
ROTZ (about nodal Z)

7 --
PRESS

8 --
TEMP

KEYOPT(4)
ON-OFF range behavior (see *Figure 2, "COMBIN37 Behavior as a Function of Control Parameter"*):

0 --
Overlapping ranges

1 --
Unique ranges

KEYOPT(5)
ON-OFF position behavior (see *Figure 2, "COMBIN37 Behavior as a Function of Control Parameter"*):

0 --
OFF-either-ON (or OFF-ON-OFF if unique)

1 --
ON-either-OFF (or ON-OFF-ON if unique)

KEYOPT(6)
Real constants used for RVMOD function (used if C1 or C3 is not equal to zero; see *COMBIN37 Input Data*):

0, 1 --
Use STIF for nonlinear function. (Both STIF and FSLIDE cannot be zero).

2 --
Use DAMP

3 --
Use MASJ

4 --
Use ONVAL

5 --
Use OFFVAL

6 --
Use AFORCE

7 --
Use MASI

8 --
Use FSLIDE

KEYOPT(9)
Method to define nonlinear behavior:

0 --
Use RVMOD expression for real constant modifications

1 --
Real constants modified by user subroutine USERRC

Note

See the *Guide to ANSYS User Programmable Features* information about user written subroutines

Table 1 COMBIN37 Real Constants

No.	Name	Description
1	STIF	Spring stiffness
2	DAMP	Damping coefficient
3	MASJ	Nodal mass at node J
4	ONVAL	"ON" control value
5	OFFVAL	"OFF" control value
6	AFORCE	Element load
7	MASI	Nodal mass at node I
8	START	Initial on/off element status
9	C1	First scalar in RVMOD equation
10	C2	First exponent in RVMOD equation
11	C3	Second scalar in RVMOD equation
12	C4	Second exponent in RVMOD equation
13	FSLIDE	Limiting sliding force

COMBIN37 Output Data

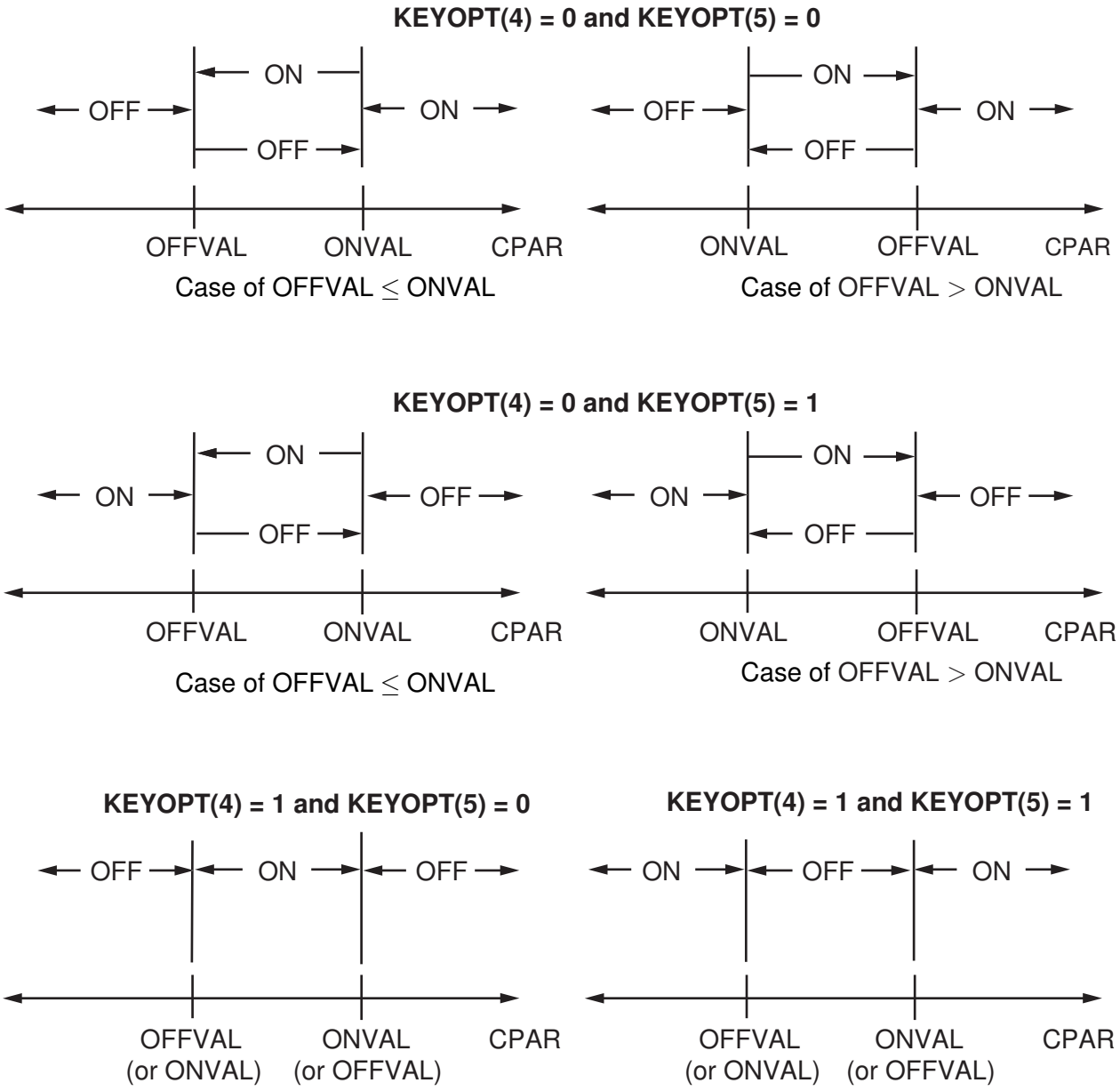
The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "COMBIN37 Element Output Definitions"*.

The active nodal displacements and forces correspond to the degree of freedom selected with the KEYOPT(3) option. For axisymmetric analysis, the element forces are expressed on a full 360° basis.

The element value STRETCH is the relative deflection at the end of the substep less the amount of sliding (e.g., UJ-UI-SLIDE). STATUS and OLDST indicate if the element is on or off at the end of the current and previous substeps, respectively. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 COMBIN37 Behavior as a Function of Control Parameter



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 COMBIN37 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
ACTIVE NODES	Nodes - I, J	Y	Y

Name	Definition	O	R
CONTROL NODES	Nodes - K, L	Y	Y
XC, YC, ZC	Location where results are reported	Y	*5*
CONTROL PARAM	CPAR value (see KEYOPT(1)) of the control nodes	Y	Y
STAT	Element status	*1*	*1*
OLDST	STAT value of the previous time step	*1*	*1*
UI	Displacement of node I	*2*	*2*
UJ	Displacement of node J	*2*	*2*
UK	Displacement of node K	*2*	*2*
UL	Displacement of node L	*2*	*2*
STRETCH	Relative displacement	*2*	*2*
SFORCE	Spring force in element	*2*	*2*
AFORCE	Applied force in the element	*2*	*2*
SLSTAT	Sliding status	*3*	*3*
OLDSLS	Sliding status value of the previous time step	*3*	*3*
SLIDE	Amount of sliding	*4*	*4*

1. If the value of the element status is:

   0 - OFF

   1 - ON

2. For the thermal and fluid options, analogous items are output. Thermal option output items TEMPI, TEMPJ, TEMPK, TEMPL, DELTEMP, SHEAT, and AHEAT and fluid option output items PRESI, PRESJ, PRESK, PRESL, DELPRES, SFLOW, and AFLOW are respectively analogous to output items UI, UJ, UK, UL, STRETCH, SFORCE, and AFORCE.
3. Output only if FSLIDE is greater than zero. If the value of the sliding status is:

   0 - No sliding

   1 - Sliding right (node J moving to right of node I)

   -1- Sliding left (node J moving to left of node I)
4. If FSLIDE is greater than zero
5. Available only at centroid as a ***GET** item.

Table 3, "COMBIN37 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "COMBIN37 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "COMBIN37 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 COMBIN37 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SFORCE	SMISC	1
AFORCE	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
SLSTAT	NMISC	3
OLDSLS	NMISC	4
STRETCH	NMISC	5
UI	NMISC	6
UJ	NMISC	7
UK	NMISC	8
UL	NMISC	9
CPAR	NMISC	10
SLIDE	NMISC	11

Analogous thermal and fluid option output items use the same item and sequence numbers. See footnote *2* of *Table 2, "COMBIN37 Element Output Definitions"*.

COMBIN37 Assumptions and Restrictions

- The element may have only one degree of freedom per node which is specified in the nodal coordinate system (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*).
- The element assumes only a one-dimensional action.
- Nodes I and J may be anywhere in space (preferably coincident).
- No moment effects are included due to noncoincident nodes. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.
- The nonlinear capabilities of the element operate only in static and nonlinear transient dynamic analyses.
- If used in other analysis types, the element maintains its initial status (on or off), throughout the analysis.
- The real constants for this element are not allowed to be changed from their initial values.
- The element can not be deactivated with the **EKILL** command.
- Only the lumped mass matrix is available.

COMBIN37 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(2) = 8 is not allowed.
- KEYOPT(3) = 8 is not allowed.

FLUID38

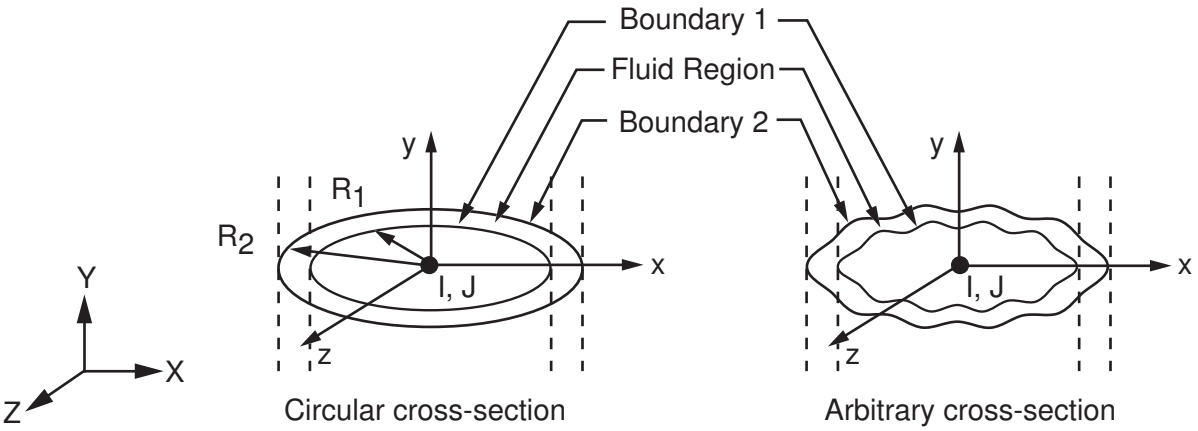
Dynamic Fluid Coupling

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID38 Element Description

FLUID38 is used to represent a dynamic coupling between two points of a structure. The coupling is based on the dynamic response of two points connected by a constrained mass of fluid. The points represent the centerlines of concentric cylinders. The fluid is contained in the annular space between the cylinders. The cylinders may be circular or have an arbitrary cross-section. The element has two degrees of freedom per node: for example, translations in the nodal x and z directions. The axes of the cylinders are then assumed to be in the nodal y directions. The element may be used in any structural dynamic analysis. For certain cases the axisymmetric harmonic fluid element, FLUID81 (with MODE = 1), can also be used. See FLUID38 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID38 Geometry



FLUID38 Input Data

The node locations and the coordinate system for this element are shown in *Figure 1, "FLUID38 Geometry"*. The element is defined by two nodes and several real constants. The real constants are defined in *Table 1, "FLUID38 Real Constants"*.

KEYOPT(3) is used to select the form of the fluid coupling element. The form of the element determines the real constants required, the material properties (if any), and the matrices calculated. The density is input as material property DENS and is evaluated at the average of the two node temperatures. The damping matrix is calculated only if F is nonzero. KEYOPT(6) is used to select the direction of operation for the element. If KEYOPT(6) = 1, the X and Y labels used in this description should be interchanged. Similarly, if KEYOPT(6) = 3, interchange the Z and Y labels.

A summary of the element input is given in *FLUID38 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

FLUID38 Input Summary

Nodes
I, J

Degrees of Freedom

UX, UZ if KEYOPT(6) = 0 or 2, or
UY, UZ if KEYOPT(6) = 1, or
UX, UY if KEYOPT(6) = 3

Real Constants

If KEYOPT(3) = 0:

R2, R1, L, F, DX, DZ,
WX, WZ

If KEYOPT(3) = 2:

M2, M1, MHX, MHZ, DX, DZ,
WX, WZ, CX, CZ
See *Table 1, "FLUID38 Real Constants"* for a description of the real constants

Material Properties

DENS if KEYOPT (3) = 0
None if KEYOPT (3) = 2

Surface Loads

None

Body Loads

Temperature --
T(I), T(J)

Special Features

None

KEYOPT(3)

Cross-section of cylinders:

0 --
Concentric circular cylinders

2 --
Concentric arbitrary cylinders

KEYOPT(6)

Flow axis parallel to:

0, 2 --
Nodal Y axis (UX, UZ degrees of freedom)

1 --
Nodal X axis (UX, UZ degrees of freedom)

3 --
Flow axis parallel to nodal Z axis (UX, UY degrees of freedom)

Table 1 FLUID38 Real Constants

No.	Name	Description
Concentric Circular Cylinders: KEYOPT(3) = 0		
1	R2	Radius of outer cylinder (length); node J refers to outer boundary
2	R1	Radius of inner cylinder (length); node I refers to outer boundary
3	L	Length of cylinders
4	F	Darcy friction factor for turbulent flow
5	DX	Estimate of maximum relative amplitude DX
6	DZ	Estimate of maximum relative amplitude DZ
7	WX	Estimate of resonant X frequency (Rad/Time)
8	WZ	Estimate of resonant Z frequency (Rad/Time)
Concentric Arbitrary Cylinders: KEYOPT(3) = 2		
1	M2	Mass of fluid that could be contained within the outer boundary (Boundary 2) in absence of inner boundary.
2	M1	Mass of fluid displaced by the inner boundary (Boundary 1)
3	MHX	Hydrodynamic mass in X direction
4	MHZ	Hydrodynamic mass in Z direction
5	DX	Estimate of maximum relative amplitude DX
6	DZ	Estimate of maximum relative amplitude DZ
7	WX	Estimate of resonant X frequency (Rad/Time)
8	WZ	Estimate of resonant Z frequency (Rad/Time)
9	CX	Flow and geometry constant for X motion (mass/lenght)
10	CZ	Flow and geometry constant for Z motion (mass/length)

FLUID38 Output Data

There is no element solution output associated with the element.

FLUID38 Assumptions and Restrictions

- The element operates in the nodal coordinate system (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*).
- No fluid coupling exists in the flow axis direction.
- The element has no nodal coordinate system transformation to account for nonparallel nodal coordinate systems.
- Nodes I and J may be located anywhere in space (preferably coincident).
- The lumped mass option [**LUMPM**] is not available with this element.

FLUID38 Product Restrictions

There are no product-specific restrictions for this element.

COMBIN39

Nonlinear Spring

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

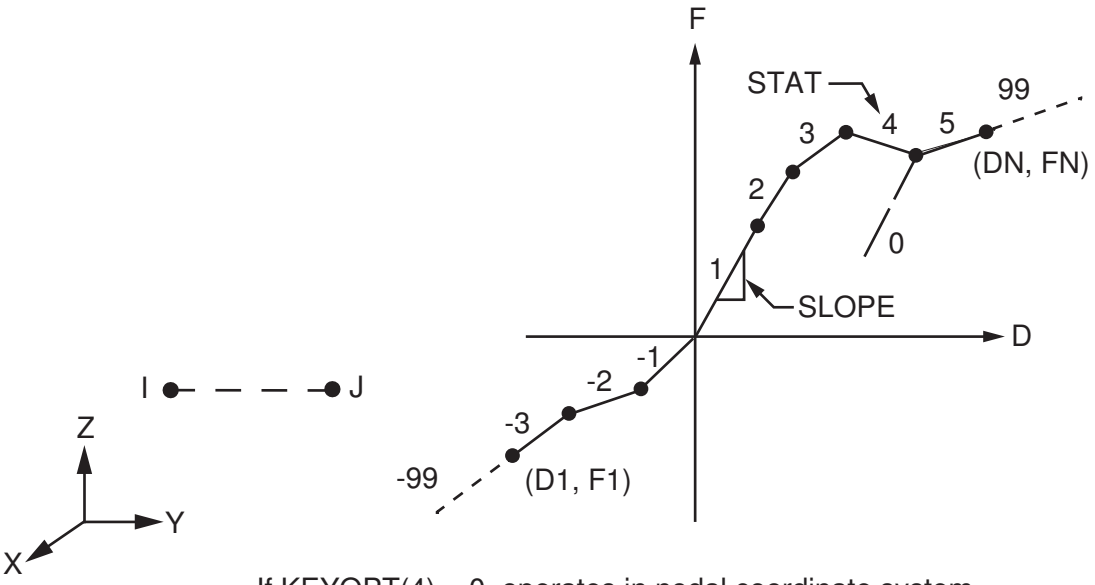
COMBIN39 Element Description

COMBIN39 is a unidirectional element with nonlinear generalized force-deflection capability that can be used in any analysis. The element has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.

The element has large displacement capability for which there can be two or three degrees of freedom at each node.

See COMBIN39 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. The element has no mass or thermal capacitance. These may be added by using the appropriate elements (see MASS21 and MASS71). A bilinear force-deflection element with damping and gaps is also available (COMBIN40).

Figure 1 COMBIN39 Geometry



If KEYOPT(4) = 0, operates in nodal coordinate system

COMBIN39 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "COMBIN39 Geometry"*. The element is defined by two (preferably coincident) node points and a generalized force-deflection curve. The points on this curve (D1, F1, etc.) represent force (or moment) versus relative translation (or rotation) for structural analyses, and heat (or flow) rate versus temperature (or pressure) difference for a thermal analyses. The loads should be defined on a full 360° basis for an axisymmetric analysis.

The force-deflection curve should be input such that deflections are increasing from the third (compression) to the first (tension) quadrants. Adjacent deflections should not be nearer than 1E-7 times total input deflection range. The last input deflection must be positive. Segments tending towards vertical should be avoided. If the

force-deflection curve is exceeded, the last defined slope is maintained, and the status remains equal to the last segment number. If the compressive region of the force-deflection curve is explicitly defined (and not reflected), then at least one point should also be at the origin (0,0) and one point in the first (tension) quadrant. If KEYOPT(2) = 1 (no compressive resistance), the force-deflection curve should not extend into the third quadrant. Note that this tension-only behavior can cause convergence difficulties similar to those that can be experienced by contact elements. See the *Contact Technology Guide*, as well as various contact element descriptions, for guidelines on overcoming convergence difficulties. Note that the number of points defining the loading curve (20 points) can be effectively doubled by using the reflective option.

Slopes of segments may be either positive or negative, except that the slopes at the origin must be positive and, if KEYOPT(1) = 1, slopes at the ends may not be negative. Also, if KEYOPT(1) = 1, force-deflection points may not be defined in the second or fourth quadrants and the slope of any segment may not be greater than the slope of the segment at the origin in that quadrant.

The KEYOPT(1) option allows either unloading along the same loading curve or unloading along the line parallel to the slope at the origin of the curve. This second option allows modeling of hysteretic effects. As illustrated in *Figure 2, "COMBIN39 Force-Deflection Curves"*, the KEYOPT(2) option provides several loading curve capabilities.

The KEYOPT(3) option selects one degree of freedom. This may be a translation, a rotation, a pressure or a temperature.

Alternately, the element may have more than one type of degree of freedom (KEYOPT(4) > 0). The two nodes defining the element should not be coincident, since the load direction is colinear with the line joining the nodes. The longitudinal option (KEYOPT(4) = 1 or 3) creates a uniaxial tension-compression element with two or three translational degrees of freedom at each node. No bending or torsion is considered. The torsional option (KEYOPT(4) = 2) creates a purely rotational element with three rotational degrees of freedom at each node. No bending or axial loads are considered. The stress stiffening capability is applicable when forces are applied, but not when torsional loads are applied.

The element has large displacement capability with two or three degrees of freedom for each node when you use KEYOPT(4) = 1 or 3 in combination with **NLGEOM**,ON.

Convergence difficulties caused by moving through rapid changes of the slope (tangent) of the force-deflection diagram are sometimes helped by use of line search (**LNSRCH**,ON).

A summary of the element input is given in *COMBIN39 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBIN39 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, or TEMP.
Make 1-D choices with KEYOPT(3).
Make limited 2- or 3-D choices with KEYOPT(4).

Real Constants

D1, F1, D2, F2, D3, F3,
D4, F4, ..., D20, F20
See *Table 1, "COMBIN39 Real Constants"* for a description of the real constants

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Stress stiffening
Large displacement

KEYOPT(1)
Unloading path:

0 --
Unload along same loading curve

1 --
Unload along line parallel to slope at origin of loading curve

KEYOPT(2)
Element behavior under compressive load:

0 --
Compressive loading follows defined compressive curve (or reflected tensile curve if not defined)

1 --
Element offers no resistance to compressive loading

2 --
Loading initially follows tensile curve then follows compressive curve after buckling (zero or negative stiffness)

KEYOPT(3)
Element degrees of freedom (1-D) (KEYOPT(4) overrides KEYOPT(3)):

0, 1 --
UX (Displacement along nodal X axes)

2 --
UY (Displacement along nodal Y axes)

3 --
UZ (Displacement along nodal Z axes)

4 --
ROTX (Rotation about nodal X axes)

5 --
ROTY (Rotation about nodal Y axes)

6 --
ROTZ (Rotation about nodal Z axes)

7 --
PRES

8 --
TEMP

KEYOPT(4)
Element degrees of freedom (2-D or 3-D):

0 --
Use any KEYOPT(3) option

1 --
3-D longitudinal element (UX, UY and UZ)

2 --
3-D torsional element (ROTX, ROTY and ROTZ)

3 --
2-D longitudinal element. (UX and UY) Element must lie in an X-Y plane

KEYOPT(6)
Element output:

0 --
Basic element printout

1 --
Also print force-deflection table for each element (only at first iteration of problem)

Table 1 COMBIN39 Real Constants

No.	Name	Description
1	D1	D value for the first point on force-deflection curve
2	F1	F value for the first point on force-deflection curve
3	D2	D value for the second point on force-deflection curve
4	F2	F value for the second point on force-deflection curve
5, ... 40	D3, F3, etc.	Continue input of D and F values up to a maximum of 20 points on the force-deflection curve

COMBIN39 Output Data

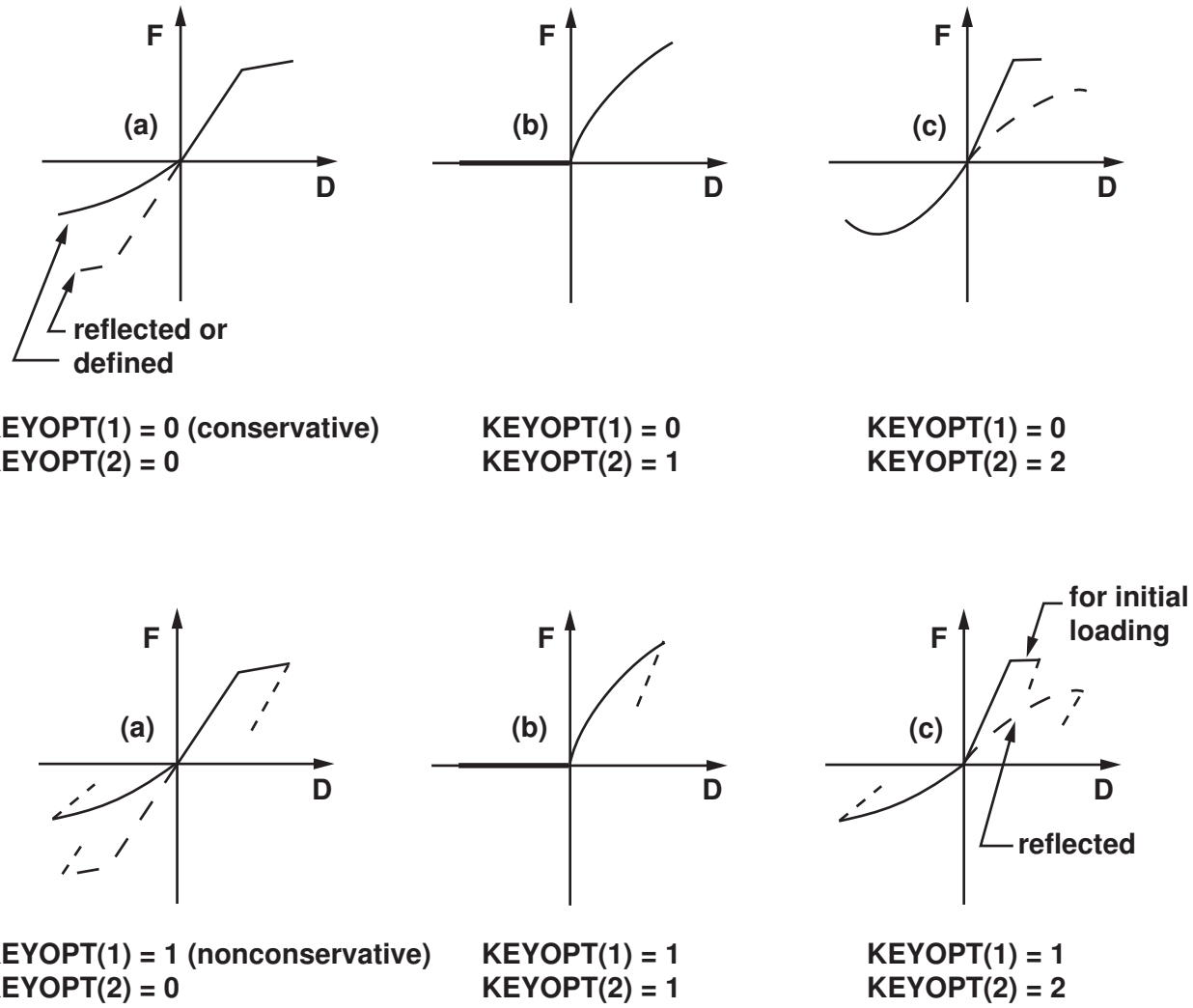
The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "COMBIN39 Element Output Definitions"*

The nodal displacements and forces correspond to the degrees of freedom selected with KEYOPT(3). For an axisymmetric analysis, the element forces are expressed on a full 360° basis. The element value STRETCH is the relative deflection at the end of the substep (e.g., UX(J) - UX(I) - UORIG, etc.). STAT and OLDST describe the curve segment number at the end of the current and previous substeps, respectively. STAT or OLDST = 0 indicates nonconservative unloading (KEYOPT(1) = 1). A status of 99 or -99 (as shown in *Figure 1, "COMBIN39 Geometry"*) indicates that the active load point on the curve is outside of the supplied data. The slope of the last segment that is provided is simply continued beyond the last data point.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 COMBIN39 Force-Deflection Curves



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 COMBIN39 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*4*
UORIG	Origin shift upon reversed loading	*1*	*1*
FORCE	Force in element	Y	Y
STRETCH	Relative displacement (includes origin shift)	Y	Y
STAT	Status at end of this time step	*2*	*2*

Name	Definition	O	R
OLDST	Same as STAT except status assumed at beginning of this time step	*2*	*2*
UI	Displacement of node I	Y	Y
UJ	Displacement of node J	Y	Y
CRUSH	Status of the force deflection curve after buckling	*3*	-
SLOPE	Current slope	Y	-

1. If KEYOPT(1) = 1
2. If the value of STAT is:

   0 - Indicates nonconservative unloading

   1-20 - Curve segment number at end of time step

   99 - Beyond last segment (last segment is extrapolated) (negative STAT values indicate compressive segments)
3. If KEYOPT(2) = 2 and if the value of CRUSH is:

   0 - Use defined tensile curve

   1 - Use reflected compressive curve in tension (element has been compressed)
4. Available only at centroid as a ***GET** item.

Table 3, "COMBIN39 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "COMBIN39 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "COMBIN39 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 COMBIN39 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCE	SMISC	1
STRETCH	NMISC	1
UI	NMISC	2
UJ	NMISC	3
UORIG	NMISC	4
STAT	NMISC	5
OLDST	NMISC	6

COMBIN39 Assumptions and Restrictions

- If you specify KEYOPT(4) = 0, the element has only one degree of freedom per node. This degree of freedom defined by KEYOPT(3), is specified in the nodal coordinate system and is the same for both nodes (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*). KEYOPT(3) also defines the direction of the force.
- The element assumes only a 1-D action. Nodes I and J may be anywhere in space (preferably coincident).
- The element is defined such that a positive displacement of node J relative to node I tends to put the element in tension.
- If you specify KEYOPT(4) $\neq$ 0, the element has two or three displacement degrees of freedom per node. Nodes I and J should not be coincident, since the line joining the nodes defines the direction of the force.
- The element is nonlinear and requires an iterative solution.
- The nonlinear behavior of the element operates only in static and nonlinear transient dynamic analyses.
- As with most nonlinear elements, loading and unloading should occur gradually.
- When the element is also nonconservative, loads should be applied along the actual load history path and in the proper sequence.
- The element can not be deactivated with the **EKILL** command.
- The real constants for this element can not be changed from their initial values.
- Whenever the force that the element carries changes sign, UORIG is reset, and the origin of the force-deflection curve effectively shifts over to the point where the force changed sign. If KEYOPT(2) = 1 and the force tends to become negative, the element "breaks" and no force is transmitted until the force tends to become positive again.
- When KEYOPT(1) = 1, the element is both nonlinear and nonconservative.
- In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

COMBIN39 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(3) = 8 (temperature DOF) is not allowed.

COMBIN40

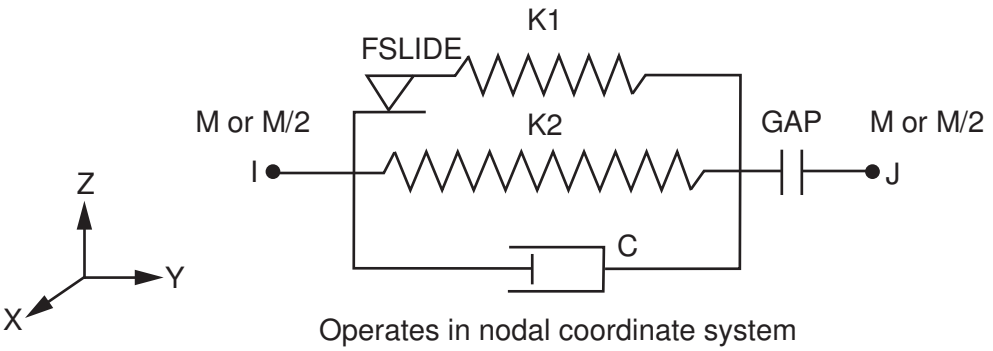
Combination

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

COMBIN40 Element Description

COMBIN40 is a combination of a spring-slider and damper in parallel, coupled to a gap in series. A mass can be associated with one or both nodal points. The element has one degree of freedom at each node, either a nodal translation, rotation, pressure, or temperature. The mass, springs, slider, damper, and/or the gap may be removed from the element. The element may be used in any analysis. See COMBIN40 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other elements having damper, slider, or gap capabilities are COMBIN7, LINK10, CONTAC12, COMBIN14, MATRIX27, COMBIN37, COMBIN39, and CONTAC52.

Figure 1 COMBIN40 Geometry



COMBIN40 Input Data

The combination element is shown in *Figure 1, "COMBIN40 Geometry"*. The element is defined by two nodes, two spring constants K1 and K2 (Force/Length), a damping coefficient C (Force*Time/Length), a mass M (Force*Time^2/Length), a gap size GAP (Length), and a limiting sliding force FSLIDE (Force). (Units listed here apply only to KEYOPT(3) = 0, 1, 2, or 3.)

If the element is used in an axisymmetric analysis, these values (except GAP) should be on a full 360° basis. A spring constant of 0.0 (for either K1 or K2, but not both) or a damping coefficient of 0.0 will remove these capabilities from the element. The mass, if any, may be applied at node I or node J or it may be equally distributed between the nodes.

The gap size is defined by the fourth element real constant. If positive, a gap of this size exists. If negative, an initial interference of this amount exists. If GAP = 0.0, the gap capability is removed from the element. The FSLIDE value represents the absolute value of the spring force that must be exceeded before sliding occurs. If FSLIDE is 0.0, the sliding capability of the element is removed, that is, a rigid connection is assumed.

A "breakaway" feature is available to allow the element stiffness (K1) to drop to zero once a limiting force |FSLIDE| has been reached. The limit is input as -|FSLIDE| and is applicable to both tensile breaking and compressive crushing. A "lockup" feature may be selected with KEYOPT(1). This feature removes the gap opening capability once the gap has closed.

The force-deflection relationship for the combination element is as shown in *Figure 2, "COMBIN40 Behavior"* (for no damping). If the initial gap is identically zero, the element responds as a spring-damper-slider element having both tension and compression capability. If the gap is not initially zero, the element responds as follows: when

the spring force (F1+F2) is negative (compression), the gap remains closed and the element responds as a spring-damper parallel combination. As the spring force (F1) increases beyond the FSLIDE value, the element slides and the F1 component of the spring force remains constant. If FSLIDE is input with a negative sign, the stiffness drops to zero and the element moves with no resisting F1 spring force. If the spring force becomes positive (tension), the gap opens and no force is transmitted. In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

The element has only the degrees of freedom selected with KEYOPT(3). The KEYOPT(3) = 7 and 8 options allow the element to be used in the thermal analysis (with thermal equivalent real constants).

A summary of the element input is given in *COMBIN40 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBIN40 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, or TEMP (depending on KEYOPT(3) below)

Real Constants
Units for real constants will depend on the KEYOPT(3) setting.

K1 - Spring constant
C - Damping coefficient
M - Mass
GAP - Gap size
FSLIDE - Limiting sliding force
K2 - Spring constant (par to slide)

Note

If GAP is exactly zero, the interface cannot open. If GAP is negative, there is an initial interference. If FSLIDE is exactly zero, the sliding capability is removed. If FSLIDE is negative, the "breakaway" feature is used.

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features

Nonlinear (unless both GAP and FSLIDE equal zero)
Adaptive descent

KEYOPT(1)
Gap behavior:

0 --
Standard gap capability

1 --
Gap remains closed after initial contact ("lockup")

KEYOPT(3)
Element degrees of freedom:

0, 1 --
UX (Displacement along nodal X axes)

2 --
UY (Displacement along nodal Y axes)

3 --
UZ (Displacement along nodal Z axes)

4 --
ROTX (Rotation about nodal X axes)

5 --
ROTY (Rotation about nodal Y axes)

6 --
ROTZ (Rotation about nodal Z axes)

7 --
PRES

8 --
TEMP

KEYOPT(4)
Element output:

0 --
Produce element printout for all status conditions

1 --
Suppress element printout if gap is open (STAT = 3)

KEYOPT(6)
Mass location:

0 --
Mass at node I

1 --
Mass equally distributed between nodes I and J

2 --
Mass at node J

COMBIN40 Output Data

The solution output associated with the element is in two forms:

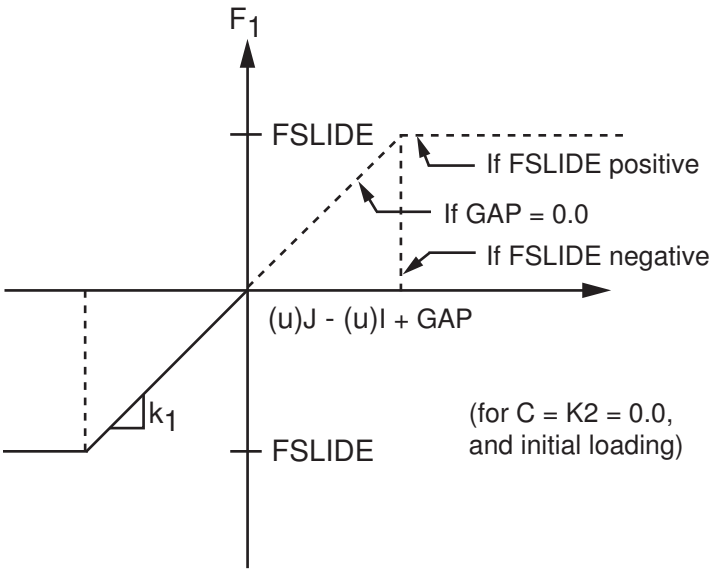
- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Figure 2, "COMBIN40 Behavior"*

Several items are illustrated in *Figure 2, "COMBIN40 Behavior"*. The displacement direction corresponds to the nodal coordinate direction selected with KEYOPT(3). The value STR is the spring displacement at the end of this substep, STR = U(J)-U(I)+GAP-SLIDE. This value is used in determining the spring force. For an axisymmetric

analysis, the element forces are expressed on a full 360° basis. The value SLIDE is the accumulated amount of sliding at the end of this substep relative to the starting location.

STAT describes the status of the element at the end of this substep for use in the next substep. If STAT = 1, the gap is closed and no sliding occurs. If STAT = 3, the gap is open. If STAT = 3 at the end of a substep, an element stiffness of zero is being used. A value of STAT = +2 indicates that node J moves to the right of node I. STAT = -2 indicates a negative slide. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 COMBIN40 Behavior



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 COMBIN40 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
SLIDE	Amount of sliding	Y	Y
F1	Force in spring 1	Y	Y
STR1	Relative displacement of spring 1	Y	Y
STAT	Element status	*1*	*1*
OLDST	STAT value of the previous time step	*1*	*1*
UI	Displacement of node I	Y	Y
UJ	Displacement of node J	Y	Y
F2	Force in spring 2	Y	Y

Name	Definition	O	R
STR2	Relative displacement of spring 2	Y	Y

1. If the value of STAT is:

   1 - Gap closed (no sliding)

   2 - Sliding right (node J moving to right of node I)

   -2 - Sliding left (node J moving to left of node I)

   3 - Gap open

2. Available only at centroid as a ***GET** item.

Table 2, "COMBIN40 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "COMBIN40 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "COMBIN40 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 COMBIN40 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
F1	SMISC	1
F2	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
STR1	NMISC	3
STR2	NMISC	4
UI	NMISC	5
UJ	NMISC	6
SLIDE	NMISC	7

COMBIN40 Assumptions and Restrictions

- The element has only one degree of freedom per node which is specified in the nodal coordinate system (see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*).
- The element assumes only a 1-D action.
- Nodes I and J may be anywhere in space (preferably coincident).

- The element is defined such that a positive displacement of node J relative to node I tends to open the gap. If, for a given set of conditions, nodes I and J are interchanged, the gap element acts as a hook element, i.e., the gap closes as the nodes separate.
- The real constants for this element can not be changed from their initial values.
- The element can not be deactivated with the **EKILL** command.
- The nonlinear options of the element operate only in static and nonlinear transient dynamic (**TRNOPT**,FULL) analyses.
- If used in other analysis types, the element maintains its initial status throughout the analysis.
- A 0.0 value for GAP or FSLIDE removes the gap or sliding capability, respectively, from the element.
- The mass, if any, is 1-D.
- The element requires an iterative solution if GAP and/or FSLIDE are nonzero.
- A stiffness (K1 or K2) must be defined if the gap capability is used. Unreasonably high stiffness values should be avoided.
- The rate of convergence may decrease as the stiffness increases. If FSLIDE is not equal to zero, the element is nonconservative as well as nonlinear. Nonconservative elements require that the load be applied very gradually, along the actual load history path, and in the proper sequence (if multiple loadings exist).
- Only the lumped mass matrix is available.

COMBIN40 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; CV1 and CV2 are not allowed.
- Only stress stiffening and large deflections are allowed.
- KEYOPT(3) = 7 or 8 is not allowed.
- The DAMP material property is not allowed.
- FSLIDE and K2 not allowed.

ANSYS Structural

- KEYOPT(3) = 8 (temperature DOF) is not allowed.

SHELL41

Membrane Shell

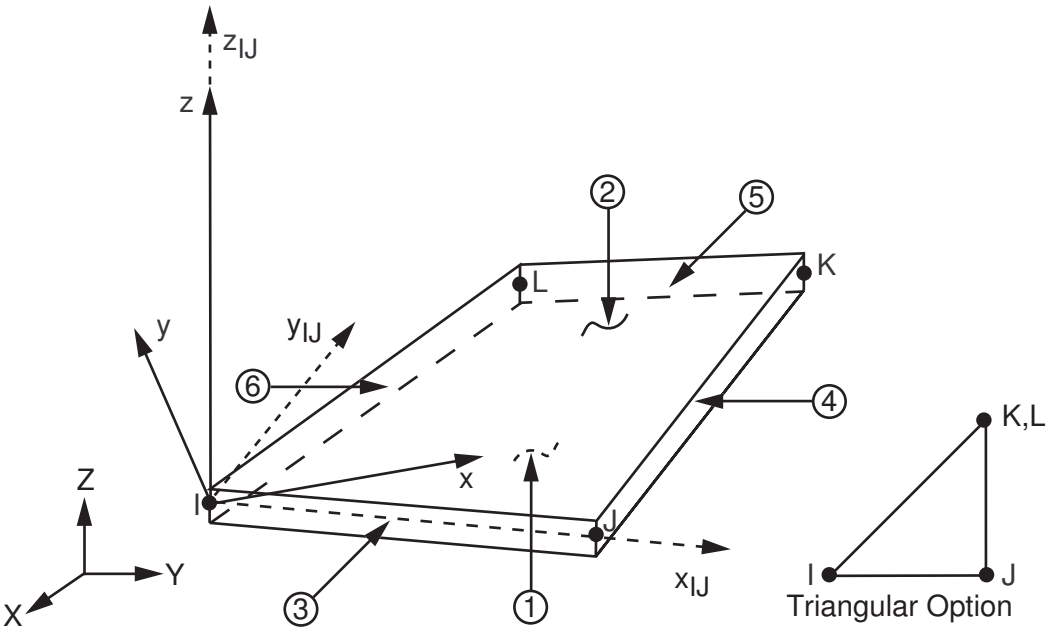
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL41 Element Description

SHELL41 is a 3-D element having membrane (in-plane) stiffness but no bending (out-of-plane) stiffness. It is intended for shell structures where bending of the elements is of secondary importance. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has variable thickness, stress stiffening, large deflection, and a cloth option. See SHELL41 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Another element having "membrane only" capability as an option is SHELL63.

Figure 1 SHELL41 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL41 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL41 Geometry"*. The element is defined by four nodes, four thicknesses, a material direction angle and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element x-axis may be rotated by an angle THETA (in degrees).

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input. The elastic foundation stiffness (EFS) is defined

as the pressure required to produce a unit normal deflection of the foundation. The elastic foundation capability is bypassed if EFS is less than, or equal to, zero. ADMSUA is the added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL41 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length. The pressure loading is converted to equivalent element loads applied at the nodes. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Use KEYOPT(1) for a tension-only option. This nonlinear option acts like a cloth in that tension loads will be supported but compression loads will cause the element to wrinkle. This capability is a shell version of LINK10, the tension-only spar.

You should *not* use this "cloth" option to model cloth materials, since real cloth materials do contain some bending stiffness. You can use the cloth option to efficiently model regions where wrinkling is to be approximated, such as for shear panels in aircraft structures. Wrinkling for this type of application may be in one (or both) orthogonal directions. If you do need to model a real cloth material, you can use the cloth option to simulate the tension part of the loading, but you will need to superimpose a very thin regular shell element to include a bending stiffness for the material. Superimposing a thin shell may also aid solution stability.

Any out-of-planeness within the element or round off-error in nodal location may cause an instability in the displacement solution. To counteract this, a slight normal stiffness may be added to the element with the EFS real constant. KEYOPT(2) is used to include or suppress the extra displacement shapes. KEYOPT(4) provides various element printout options (see *Section 2.2.2: Element Solution*).

A summary of the element input is given in *SHELL41 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL41 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, EFS,
ADMSUA
See *Table 1, "SHELL41 Real Constants"* for a description of the real constants

Material Properties
EX, EY, PRXY or NUXY, ALPX, ALPY (or CTEX, CTEY *or* THSX,THSY), DENS, GXY, DAMP (X-direction defined by THETA real constant)

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features

Stress Stiffening
Large Deflection
Nonlinear (if KEYOPT(1) = 2)
Birth and death
Adaptive descent

KEYOPT(1)
Element stiffness behavior:

0 --
Stiffness acts in both tension and compression

2 --
Stiffness acts in tension, collapses in compression ("cloth" option)

KEYOPT(2)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shape

KEYOPT(4)
Extra stress output:

0 --
Basic element printout

1 --
Repeat basic printout at integration points

2 --
Nodal stress printout

KEYOPT(5)
Member force output:

0 --
No member force printout

1 --
Print member forces in the element coordinate system

KEYOPT(6)
Edge output (isotropic material):

0 --
No edge printout

1 --
Edge printout for midpoint of side I-J

2 --
Edge printout for midpoints of both sides I-J and K-L

Note

Edge printout valid only for isotropic materials

Table 1 SHELL41 Real Constants

No.	Name	Description
1	TK(I)	Shell thickness at node I
2	TK(J)	Shell thickness at node J (defaults to TK(I))
3	TK(K)	Shell thickness at node K (defaults to TK(I))
4	TK(L)	Shell thickness at node L (defaults to TK(I))
5	THETA	Element x-axis rotation
6	EFS	Elastic foundation stiffness
7	ADMSUA	Added mass/unit area

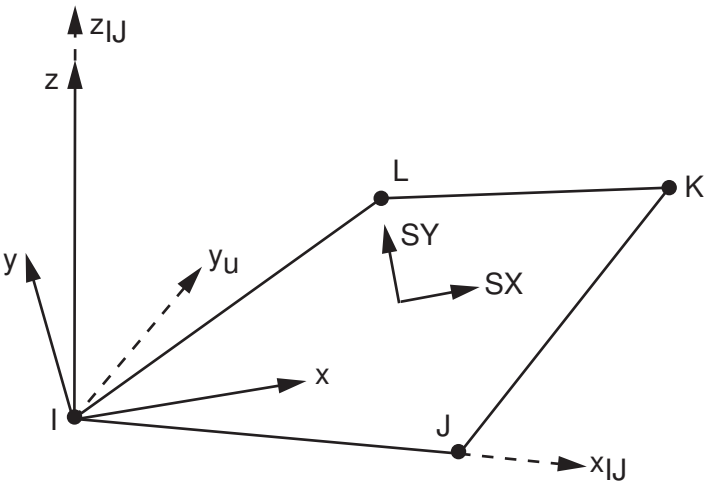
SHELL41 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL41 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL41 Stress Output"*. The element stress directions correspond to the element coordinate directions. Edge stresses are defined parallel and perpendicular to the IJ edge (and the KL edge). A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL41 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL41 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
XC, YC, ZC	Location where results are reported	Y	*4*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
S:X, Y, Z, XY	Stresses	Y	Y
S:1, 2, 3	Principal stress	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Average elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain	Y	Y
EPTH:X, Y, Z, XY	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain	Y	Y
ANGLES	Diagonal tension angles (degrees) between element x-axis and tensile stress directions	*1*	*1*
CURRENT STATS.	Element statuses at end of this time step	*2*	*2*
OLD STATUSES	Element statuses at end of previous time step	*2*	*2*
TEMP	Edge average temperature	*3*	*3*
EPEL(PAR, PER, Z)	Edge elastic strains (parallel, perpendicular, Z)	*3*	*3*
S(PAR, PER, Z)	Edge stresses (parallel, perpendicular, Z)	*3*	*3*
SINT	Edge stress intensity	*3*	*3*
SEQV	Edge equivalent stress	*3*	*3*
FX, FY, FZ	Nodal forces	-	Y

1. Output at the integration points only if KEYOPT(1) = 2 (meaningful only if STAT = 1)
2. Output at the integration points only if KEYOPT(1) = 2.The element status is given by the following values:

   0 - Tension in both (orthogonal) directions

   1 - Tension in one direction, collapse in other direction

   2 - Collapse in both directions
3. Edge I-J output, if KEYOPT(6) is greater than zero.
4. Available only at centroid as a ***GET** item.

Table 3 SHELL41 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Stress Solution	TEMP, S(X, Y, Z, XY), SINT, SEQV	*1*	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY), SINT, SEQV	*2*	-

Description	Names of Items Output	O	R
Edge K-L	TEMP, EPEL(PAR, PER, Z), S(PAR, PER, Z), SINT, SEQV	*3*	-
Member Forces	FX, FY, FZ	*4*	-

1. Output at each integration point, if KEYOPT(4) = 1
2. Output at each node, if KEYOPT(4) = 2
3. Output if KEYOPT(6) = 2
4. Output at each node (in the element coordinate system) if KEYOPT(5) = 1

Table 4, "SHELL41 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 4, "SHELL41 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SHELL41 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 4 SHELL41 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
FX	SMISC	1	4	7	10
FY	SMISC	2	5	8	11
FZ	SMISC	3	6	9	12
P1	SMISC	13	14	15	16
P2	SMISC	17	18	19	20
P3	SMISC	22	21	-	-
P4	SMISC	-	24	23	-
P5	SMISC	-	-	26	25
P6	SMISC	27	-	-	28
S:1	NMISC	1	6	11	16
S:2	NMISC	2	7	12	17
S:3	NMISC	3	8	13	18
S:INT	NMISC	4	9	14	19
S:EQV	NMISC	5	10	15	20

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	Corner Location			
		1	2	3	4
ANGLE	NMISC	21	23	25	27
STAT	NMISC	22	24	26	28

SHELL41 Assumptions and Restrictions

- The four nodes defining the element should lie in an exact flat plane; however, a small out-of-plane tolerance is permitted so that the element may have a slightly warped shape.
- A slightly warped element will produce a warning message. If the warping is too severe, a fatal message results and a triangular element should be used (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- Zero area elements are not allowed.
- TK(I) must not be zero.
- The element must not taper down to a zero thickness at any corner.
- A triangular element may be formed by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- The extra shapes are automatically deleted for triangular elements so that a constant strain element results.
- The triangular shape is required for large deflection analyses since a four-node element may warp during deflection.
- Edge stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met.

Modeling hints:

- An assembly of SHELL41 elements describing a flat plane should be *exactly* flat; otherwise singularities may develop in the direction perpendicular to the plane.
- Very weak spar elements (LINK8) tied to the nodes in the plane and to a common ground point may be added to provide a small normal stiffness, or the EFS real constant may be used to counteract the singularity problem.
- Stress stiffening will help stabilize the solution after the first substep if the membrane element is in a tension field.
- An assemblage of flat elements can produce an approximation to a curved surface, but each flat element should not extend over more than a 15° arc.

SHELL41 Product Restrictions

There are no product-specific restrictions for this element.

PLANE42

2-D Structural Solid

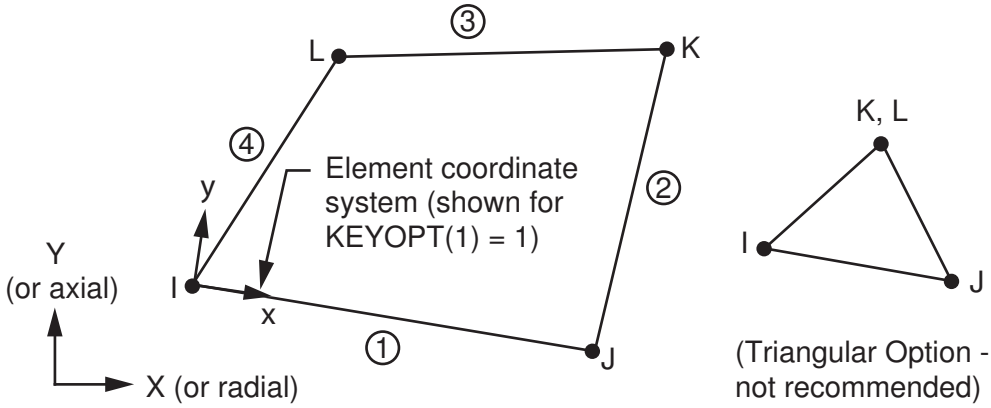
MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

PLANE42 Element Description

PLANE42 is used for 2-D modeling of solid structures. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. The element is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

An option is available to suppress the extra displacement shapes. See PLANE42 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PLANE82 for a multi-node version of this element. See PLANE25 for an axisymmetric version that accepts nonaxisymmetric loading.

Figure 1 PLANE42 Geometry



PLANE42 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE42 Geometry"*. The element input data includes four nodes, a thickness (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE42 Geometry"*. Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis. KEYOPT(2) is used to include or suppress the extra displacement shapes.

KEYOPT(5) and KEYOPT(6) provide various element printout options (see *Section 2.2.2: Element Solution*).

Initial state conditions previously handled via the **ISTRESS** command will be discontinued for this element. The **INISTATE** command will provide increased functionality, but only via the Current Technology elements (180,181,

etc.). To continue using Initial State conditions in future versions of ANSYS, consider switching to the appropriate Current Technology element. For more information on setting Initial State values see the **INISTATE** command and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. For more information on current -vs- legacy element technologies see *Section 2.17: Legacy vs. Current Element Technologies* in the *Elements Reference*

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *PLANE42 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE42 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Thickness if KEYOPT(3) = 3

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Fluences --
FL(I), FL(J), FL(K), FL(L)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO, CHABOCHE, HILL)
Creep (CREEP, RATE)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent
Initial stress import

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(2)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness input

KEYOPT(5)
Extra stress output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points

2 --
Nodal stress solution

KEYOPT(6)
Extra surface output:

0 --
Basic element solution

1 --
Surface solution for face I-J also.

2 --
Surface solution for both faces I-J and K-L also. (Surface solution available for linear materials only)

3 --
Nonlinear solution at each integration point also.

4 --
Surface solution for faces with nonzero pressure

KEYOPT(9)
Initial stress subroutine option (available only through direct input of the **KEYOPT** command):

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)

PLANE42 Output Data

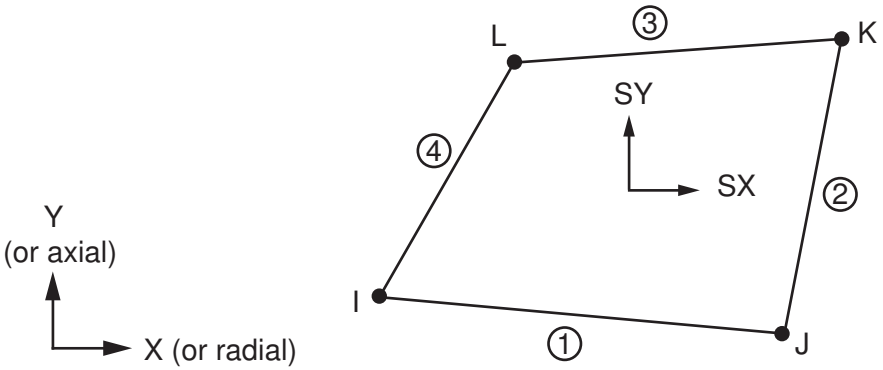
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE42 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE42 Stress Output"*.

The element stress directions are parallel to the element coordinate system. Surface stresses are available on any face. Surface stresses on face IJ, for example, are defined parallel and perpendicular to the IJ line and along the Z axis for a plane analysis or in the hoop direction for an axisymmetric analysis. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE42 Stress Output



Stress directions shown are for KEYOPT(1) = 0

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE42 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
THICK	Average thickness	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L)	Y	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	Y	-
S:INT	Stress intensity	Y	-
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strain	Y	-
EPEL:EQV	Equivalent elastic strain [*4*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
EPPL:X, Y, Z, XY	Plastic strain	*1*	*1*
EPPL:EQV	Equivalent plastic strain [*4*]	-	*1*
EPCR:X, Y, Z, XY	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*4*]	-	*1*
EPSW:	Swelling strain	*1*	*1*
NL:EPEQ	Equivalent plastic strain	*1*	*1*
NL:SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
NL:SEPL	Equivalent stress on stress-strain curve	*1*	*1*
NL:HPRES	Hydrostatic pressure	-	*1*
FACE	Face label	*2*	*2*
EPEL(PAR, PER, Z)	Surface elastic strains (parallel, perpendicular, Z or hoop)	*2*	*2*
TEMP	Surface average temperature	*2*	*2*
S(PAR, PER, Z)	Surface stresses (parallel, perpendicular, Z or hoop)	*2*	*2*
SINT	Surface stress intensity	*2*	*2*
SEQV	Surface equivalent stress	*2*	*2*
LOCI:X, Y, Z	Integration point locations	-	Y

1. Nonlinear solution, output only if the element has a nonlinear material.
2. Surface output (if KEYOPT(6) is 1,2, or 4)
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2 PLANE42 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution (KEYOPT(5) = 1)	TEMP, SINT, SEQV, EPEL(1, 2, 3), S(X, Y, Z, XY), S(1, 2, 3)	Y	-
Nodal Stress Solution (KEYOPT(5) = 2)	TEMP, S(X, Y, Z, XY), S(1, 2, 3), SINT, SEQV	Y	-
Nonlinear Integration Point Solution (KEYOPT(6) = 3)	EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR, EPSW	*1*	-

1. Valid if the element has a nonlinear material and KEYOPT(6) = 3

Note

For axisymmetric solutions with KEYOPT(1) = 0, the X, Y, Z, and XY stress and strain outputs correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively.

Table 3, "PLANE42 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PLANE42 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE42 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 3 PLANE42 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P1	SMISC	-	2	1	-	-
P2	SMISC	-	-	4	3	-
P3	SMISC	-	-	-	6	5
P4	SMISC	-	7	-	-	8
S:1	NMISC	-	1	6	11	16
S:2	NMISC	-	2	7	12	17
S:3	NMISC	-	3	8	13	18
S:INT	NMISC	-	4	9	14	19
S:EQV	NMISC	-	5	10	15	20
FLUEN	NMISC	-	21	22	23	24
THICK	NMISC	25	-	-	-	-

See *Section 2.2.2.5: Surface Solution* of this manual for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE42 Assumptions and Restrictions

- The area of the element must be nonzero.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE42 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- The extra shapes are automatically deleted for triangular elements so that a constant strain element results.
- Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met.

PLANE42 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads are not applicable.
- The only special feature allowed is stress stiffening.
- KEYOPT(6) = 3 is not applicable.

SHELL43

4-Node Plastic Large Strain Shell

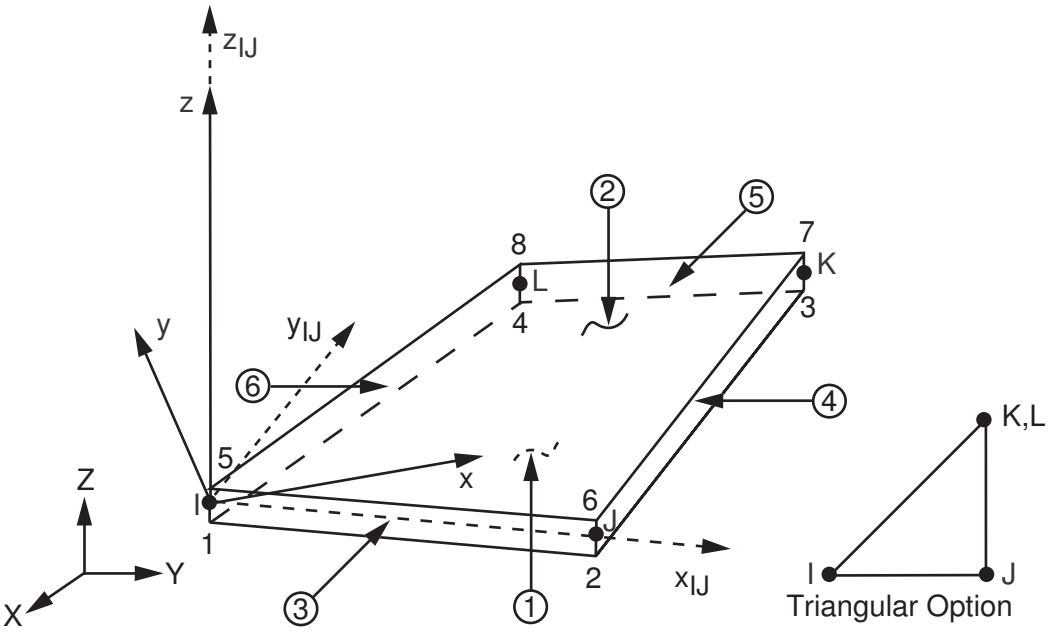
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SHELL43 Element Description

SHELL43 is well suited to model linear, warped, moderately-thick shell structures. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. The deformation shapes are linear in both in-plane directions. For the out-of-plane motion, it uses a mixed interpolation of tensorial components.

The element has plasticity, creep, stress stiffening, large deflection, and large strain capabilities. See SHELL43 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. For a thin shell capability or if plasticity or creep is not needed, the elastic quadrilateral shell (SHELL63) may be used. If convergence difficulties are encountered and large strain capability is needed, use SHELL181. Also, we recommend using SHELL181 for nonlinear structures.

Figure 1 SHELL43 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL43 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL43 Geometry"*. The element is defined by four nodes, four thicknesses, and the orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K and L as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element x axis may be rotated an angle THETA (in degrees) from the element x axis toward the element y axis.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the corner nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

A nominal in-plane rotational stiffness about the element z axis is used for KEYOPT(3) = 0 or 1. A more realistic rotational stiffness (Allman rotation) may alternately be defined (KEYOPT(3) = 2). In this case, real constants ZSTIF1 and ZSTIF2 are used to control the two spurious zero energy modes usually introduced by the Allman rotation. Default values of 1.0E-6 and 1.0E-3 are provided for ZSTIF1 and ZSTIF2, respectively. ADMSUA is the added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL43 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length. Temperatures may be input as element body loads at the "corner" locations (1-8) shown in *Figure 1, "SHELL43 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T1 is used for T1, T2, T3, and T4, while T2 (as input) is used for T5, T6, T7, and T8. For any other input pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *SHELL43 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL43 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, ZSTIF1
ZSTIF2, ADMSUA
See *Table 1, "SHELL43 Real Constants"* for a description of the real constants

Material Properties

EX, EY, EZ, (or PRXY, PRYZ, PRXZ *or* NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction),

face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --
T1, T2, T3, T4, T5, T6, T7, T8

Fluences --
FL1, FL2, FL3, FL4, FL5, FL6, FL7, FL8

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(3)

Extra displacement shapes:

0 --
Include in-plane extra displacement shapes

1 --
Suppress extra displacement shapes

2 --
Include Allman rotational stiffness (use real constants ZSTIF1 and ZSTIF2)

KEYOPT(4)

Element coordinate system defined:

0 --
No user subroutine to define element coordinate system

4 --
Element x-axis located by user subroutine USERAN

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(5)

Extra element output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points and top, middle and bottom surfaces

2 --
Nodal Stress Solution

KEYOPT(6)

Nonlinear integration point output:

0 --
Basic element solution

1 --
Nonlinear integration point solution

Table 1 SHELL43 Real Constants

No.	Name	Description
1	TK(I)	Shell thickness at node I
2	TK(J)	Shell thickness at node J
3	TK(K)	Shell thickness at node K
4	TK(L)	Shell thickness at node L
5	THETA	Element x-axis rotation
6	ZSTIF1	Allman rotation control constant (only available if KEYOPT(3) = 2)
7	ZSTIF2	Allman rotation control constant (only available if KEYOPT(3) = 2)
8	ADMSUA	Added mass/unit area

SHELL43 Output Data

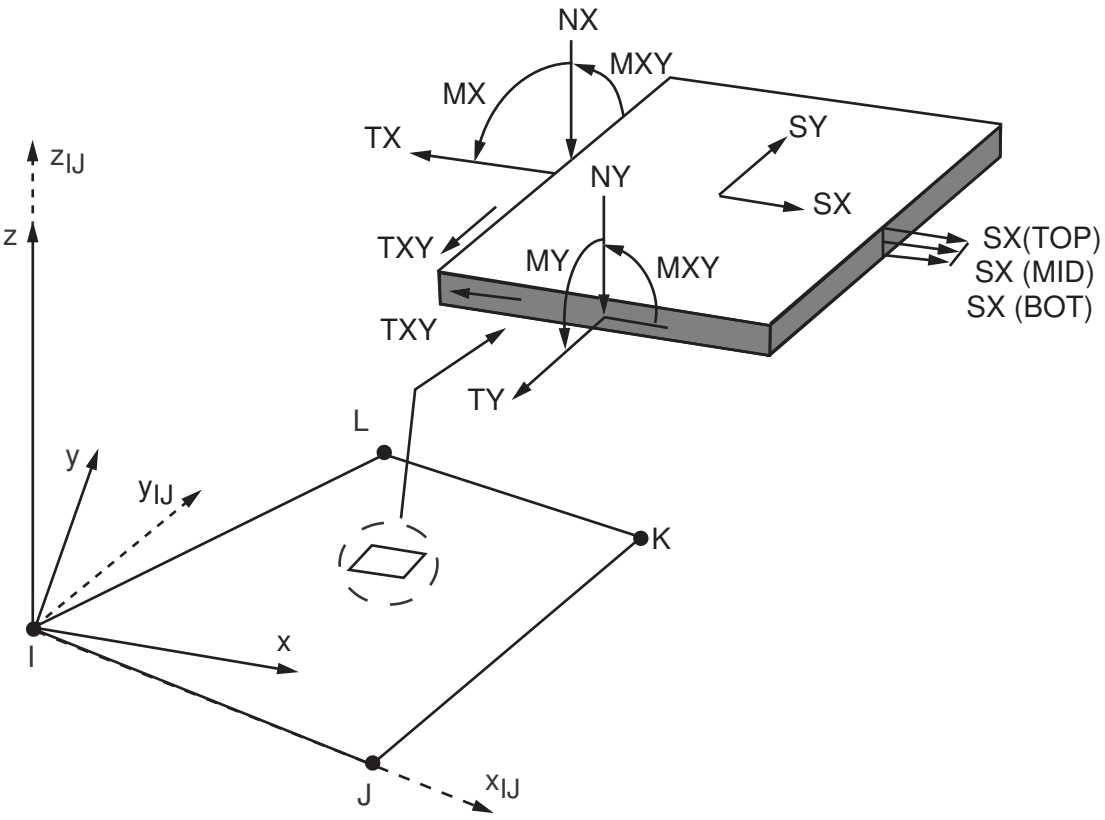
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL43 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL43 Stress Output"*.

The element stress directions and force resultants (NX, MX, TX, etc.) are parallel to the element coordinate system. The basic element printout is given at the center of the top of surface I-J-K-L, the element centroid, and at the center of the bottom of surface I-J-K-L. For triangular element configurations, the face centers and the element centroid are averaged values. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL43 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL43 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
THICK	Average thickness	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T1, T2, T3, T4, T5, T6, T7, T8	Y	Y

Name	Definition	O	R
LOC	TOP, MID, BOT, or integration point location	*1*	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*1*	*1*
S:1, 2, 3	Principal stress	*1*	*1*
S:INT	Stress intensity	*1*	*1*
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal elastic strain	*1*	*1*
EPEL:EQV	Equivalent elastic strain [*4*]	*1*	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	*2*	*2*
EPPL:EQV	Equivalent plastic strain [*4*]	-	*2*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains (X, Y, Z, XY, YZ, XZ)	*2*	*2*
EPCR:EQV	Equivalent creep strain [*4*]	-	*2*
NL:EPEQ	Average equivalent plastic strain	*2*	*2*
NL:SRAT	Ratio of trial stress to stress on yield surface	*2*	*2*
NL:SEPL	Average equivalent stress from stress-strain curve	*2*	*2*
T(X, Y, XY)	In-plane element X, Y, and XY forces	Y	Y
M(X, Y, XY)	Element X, Y, and XY moments	Y	Y
N(X, Y)	Out-of-plane element X and Y shear forces	Y	Y

1. The following stress solution repeats for top, middle, and bottom surfaces (and also for all integration points if KEYOPT(5) = 1)
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 3 SHELL43 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Point Solution	EPPL, EPEQ, SRAT, SEPL, EPCR	*1*	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV	*2*	-

1. Output at each integration point, if the element has a nonlinear material and KEYOPT(6) = 1
2. Output at each node, if KEYOPT(5) = 2, repeats each location

Table 4, "SHELL43 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 4, "SHELL43 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SHELL43 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 4 SHELL43 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
TX	SMISC	1	-	-	-	-
TY	SMISC	2	-	-	-	-
TXY	SMISC	3	-	-	-	-
MX	SMISC	4	-	-	-	-
MY	SMISC	5	-	-	-	-
MXY	SMISC	6	-	-	-	-
NX	SMISC	7	-	-	-	-
NY	SMISC	8	-	-	-	-
P1	SMISC	-	9	10	11	12
P2	SMISC	-	13	14	15	16
P3	SMISC	-	18	17	-	-
P4	SMISC	-	-	20	19	-
P5	SMISC	-	-	-	22	21
P6	SMISC	-	23	-	-	24
THICK	NMISC	49	-	-	-	-
Top						
S:1	NMISC	-	1	6	11	16
S:2	NMISC	-	2	7	12	17
S:3	NMISC	-	3	8	13	18
S:INT	NMISC	-	4	9	14	19
S:EQV	NMISC	-	5	10	15	20
Bottom						
S:1	NMISC	-	21	26	31	36
S:2	NMISC	-	22	27	32	37
S:3	NMISC	-	23	28	33	38
S:INT	NMISC	-	24	29	34	39
S:EQV	NMISC	-	25	30	35	40

		Corner Location							
		1	2	3	4	5	6	7	8
FLUEN	NMISC	41	42	43	44	45	46	47	48

SHELL43 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- Under bending loads, tapered elements produce inferior stress results and refined meshes may be required.
- Use of this element in triangular form produces results of inferior quality compared to the quadrilateral form. However, under thermal loads, when the element is doubly curved (warped), triangular SHELL43 elements produce more accurate stress results than do quadrilateral shaped elements.
- Quadrilateral SHELL43 elements may produce inaccurate stresses under thermal loads for doubly curved or warped domains.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness.
- The out-of-plane (normal) stress for this element varies linearly through the thickness.
- The transverse shear stresses (SYZ and SXZ) are assumed to be constant through the thickness.
- Shear deflections are included.
- Elastic rectangular elements without membrane loads give constant curvature results, i.e., nodal stresses are the same as the centroidal stresses.
- For linearly varying results use SHELL63 (no shear deflection) or SHELL93 (with midside nodes).
- Triangular elements are not geometrically invariant and the element produces a constant curvature solution.
- Only the lumped mass matrix is available.

SHELL43 Product Restrictions

There are no product-specific restrictions for this element.

BEAM44

3-D Elastic Tapered Unsymmetric Beam

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

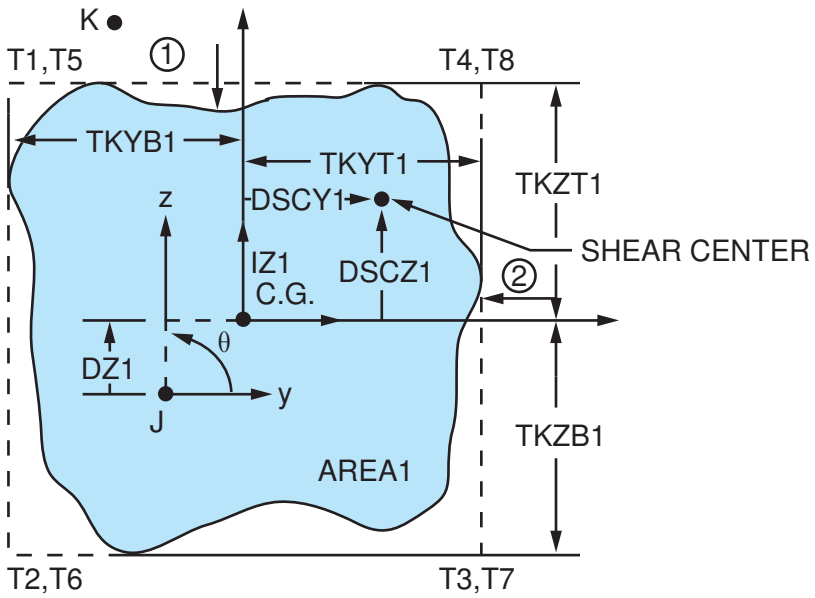
BEAM44 Element Description

BEAM44 is a uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. This element allows a different unsymmetrical geometry at each end and permits the end nodes to be offset from the centroidal axis of the beam. If these features are not desired, the uniform symmetrical beam BEAM4 may be used. A 2-D version of this element (BEAM54) is also available. For nonlinear materials, use BEAM188 or BEAM189 instead of BEAM44.

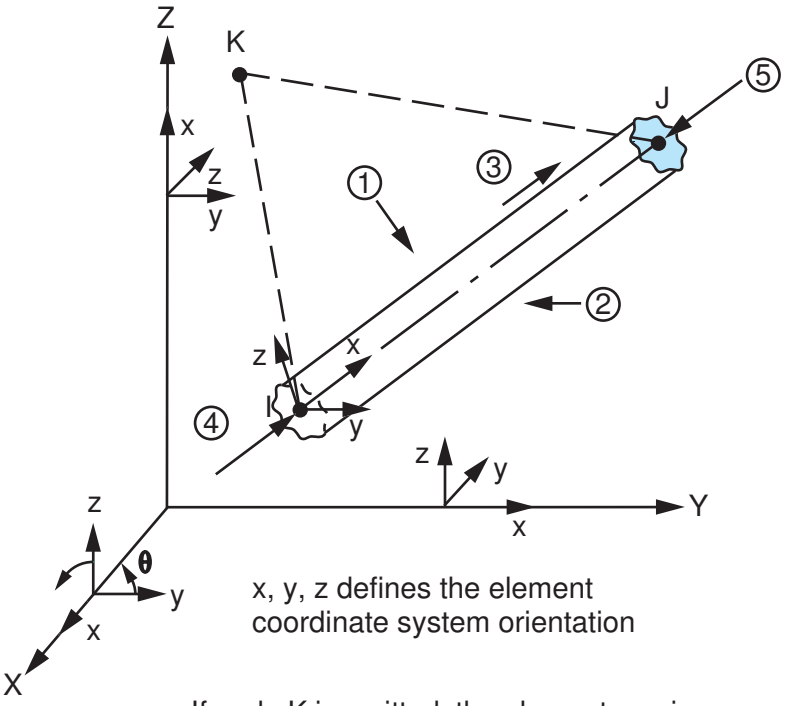
The effect of shear deformation is available as an option. Another option is available for printing the forces acting on the element in the element coordinate directions. Stress stiffening and large deflection capabilities are also included. See BEAM44 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

BEAM44 can be used with any cross section that was defined using **SECTYPE**, **SECDATA**, **SECOFFSET**, **SECWRITE**, and **SECREAD**. However, a section defined with these commands will be used *only* if there is no real constant set defined.

Figure 1 BEAM44 Geometry



Use subscript 2 with node J



x, y, z defines the element coordinate system orientation

If node K is omitted, the element y-axis is parallel to the global X-Y plane

BEAM44 Input Data

The geometry, node locations, and coordinate system for this element are shown in *Figure 1, "BEAM44 Geometry"*. The element is located by a reference coordinate system (x', y', z') and offsets. The reference system is defined by nodes I, J, and K, or an orientation angle, as shown in *Figure 1, "BEAM44 Geometry"*. The principal axes of the beam are in the element coordinate system (x, y, z) with x along the cross-section centroid (C.G.).

The element x-axis is oriented from node I (end 1) toward node J (end 2). For the two-node option, the default ($\theta = 0°$) orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in *Figure 1, "BEAM44 Geometry"*. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element x-axis, use the θ angle (THETA) or the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large deflection analysis, it should be noted that the location of the third node (K), or the angle (THETA), is used only to *initially* orient the element. For information about orientation nodes and beam meshing, see Meshing Your Solid Model in the *Modeling and Meshing Guide*. See the **LMESH** and **LATT** command descriptions for details on generating the K node automatically.

The element real constants describe the beam in terms of the cross-sectional area, the area moments of inertia, the extreme fiber distances from the centroid, the centroid offset, and the shear constants. The moments of inertia (IZ and IY) are about the lateral principal axes of the beam. The torsional moment of inertia at end 1 (IX1), if not specified, is assumed equal to the polar moment of inertia at end 1 (IZ1 + IY1). The moment of inertia values at end 2 (IX2, IY2, and IZ2), if blank, default to the corresponding end 1 values. The element torsional stiffness decreases with decreasing values of IX.

The offset constants (DX, DY, DZ) define the centroid location of the section relative to the node location. Offset distances are measured positive from the node in the positive element coordinate directions. All real constants (except the centroidal offset constants DX, DY, and DZ) for end 2 of the beam, default to the corresponding end 1 values, if zero. The "top" thicknesses at end 1, TKZT1 and TKYT1, default to the "bottom" thicknesses at end 1, TKZB1 and TKYB1, respectively. Also the "top" thicknesses at end 2, TKZT2 and TKYT2, default to the "top" thicknesses at end 1, TKZT1 and TKYT1, respectively. The thicknesses are measured from the centroid of the section.

The shear deflection constants (SHEARZ and SHEARY) are used only if shear deflection is to be included. A zero value of SHEAR_ may be used to neglect shear deflection in a particular direction. See *Section 2.14: Shear Deflection* for details.

If no real constants are defined, the cross-section details are provided separately using the **SECTYPE** and **SECDATA** commands (see Beam Analysis and Cross Sections in the *Structural Analysis Guide* for details). Note that a beam section defined using **SECTYPE** and **SECDATA** may be referenced by any combination of BEAM44, BEAM188, and BEAM189 elements in the same model. A section is associated with the beam elements by specifying the section ID number (**SECNUM**). A section number is an independent element attribute.

KEYOPT(2) allows a reduced mass matrix formulation (rotational degrees of freedom terms deleted). This option is useful for improved bending stresses in long, slender members under mass loading.

KEYOPT(7) and KEYOPT(8) allow element stiffness releases at the nodes in the element coordinate system. Stiffnesses should not be released such that free-body motion could occur, usually indicated by pivot warning or error messages. Also, translational degrees of freedom of stress stiffness matrices should not be released. Loads applied in the direction of released stiffness will be ignored. For large deflection, note that the element stiffness release follows the element orientation, whereas release by nodal coupling does not. Solution stability may be enhanced by superimposing weak (low value of EX) beam elements with no stiffness releases on the model.

The shear areas (ARES_ _) and the torsional stress factors (TSF_) are also used if they are nonzero. The shear areas are used for shear stress computation only and are generally less than the actual cross-sectional area. The torsional moment is multiplied by the torsional stress factor to calculate the torsional shear stress. Torsional stress factors may be found in structural handbooks. For circular sections, TSF = diameter/(2*IX).

For some beam cross sections, the shear center may be offset from the centroid location. Nonzero shear center offsets (DSC_ _) may be input as shown in *Figure 1, "BEAM44 Geometry"*. Offset distances are measured positive from the centroid in the positive element axes directions. End 2 offsets default to end 1 values, if zero. If constants

Y1 through Z4 are provided, additional stress printout is given at up to four user specified output points at each end of the beam as shown in *Figure 2, "BEAM44 Stress Output"*.

The elastic foundation stiffnesses (EFS_) are defined as the pressure required to produce unit normal deflections of the foundation. This capability is bypassed if the EFS_ values are zero. The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero strain length. An added mass per unit length may be input with the ADDMAS value.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM44 Geometry"*. The circled number represents the load key for the indicated face. Positive pressures act into the element. Lateral pressures are input as a force per unit length. End "pressures" are input as a force. KEYOPT(10) allows tapered lateral pressures to be offset from the nodes. Temperatures may be input as element body loads at the eight "corner" locations shown in *Figure 1, "BEAM44 Geometry"*. Temperatures 1-4 are at node I and 5-8 are at node J. Note that the temperature input points are different from the stress output points shown in *Figure 2, "BEAM44 Stress Output"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. If only T1 and T4 are input, T2 defaults to T1 and T3 defaults to T4. In both cases, T5 through T8 default to T1 through T4. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(9) is used to request output at intermediate locations. It is based on equilibrium (free body of a portion of the element) considerations and is not valid if:

- stress stiffening is turned on [**SSTIF**,ON], or
- more than one component of angular velocity is applied [**OMEGA**], or
- *any* angular velocities or accelerations are applied with the **CGOMGA**, **DOMEGA**, or **DCGOMG** commands.

A summary of the element input is given in *BEAM44 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

BEAM44 Input Summary

Nodes
I, J, K (K orientation node is optional)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

AREA1, IZ1, IY1, TKZB1, TKYB1, IX1,
AREA2, IZ2, IY2, TKZB2, TKYB2, IX2,
DX1, DY1, DZ1, DX2, DY2, DZ2,
SHEARZ, SHEARY, TKZT1, TKYT1, TKZT2, TKYT2,
ARESZ1, ARESY1, ARESZ2, ARESY2, TSF1, TSF2,
DSCZ1, DSCY1, DSCZ2, DSCY2, EFSZ, EFSY,
Y1, Z1, Y2, Z2, Y3, Z3,
Y4, Z4, Y1, Z1, Y2, Z2,
Y3, Z3, Y4, Z4, THETA, ISTRN,
ADDMAS
See *Table 1, "BEAM44 Real Constants"* for descriptions of the real constants.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J) (-Z normal direction)
face 2 (I-J) (-Y normal direction)
face 3 (I-J) (+X tangential direction)
face 4 (I) (+X axial direction)
face 5 (J) (-X axial direction)
(use negative value for opposite loading)

Body Loads

Temperatures --
T1, T2, T3, T4, T5, T6, T7, T8

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(2)
Mass matrix:

0 --
Consistent

1 --
Reduced

KEYOPT(6)
Member force and moment output:

0 --
No member force printout

1 --
Print out member forces and moments in the element coordinate system

KEYOPT(7)
Stiffness release at node I:

1 --
Release element rotational Z-stiffness

10 --
Release element rotational Y-stiffness

100 --
Release element rotational X-stiffness

1000 --
Release element translational Z-stiffness

10000 --
Release element translational Y stiffness

100000 --
Release element translational X-stiffness

To combine releases, input the sum of the number keys (such as 11 for rotational Z and Y).

KEYOPT(8)
Same as KEYOPT(7) but used for node J:

KEYOPT(9)
Output at intermediate points between ends I and J:

N --
Output at *N* intermediate locations (*N* = 0, 1, 3, 5, 7, 9)

KEYOPT(10)
Load offset (used only for tapered surface loads with the **SFBEAM** command):

0 --
Offset is in terms of length units

1 --
Offset is in terms of a length ratio (0.0 to 1.0)

Note

SHEARZ goes with IZ. If SHEARZ = 0.0, there is no shear deflection in the element Y direction.

SHEARY goes with IY. If SHEARY = 0.0, there is no shear deflection in the element Z-direction

Table 1 BEAM44 Real Constants

No.	Name	Description
1	AREA1	Cross-sectional area at end 1 (node I)
2, 3	IZ1, IY1	Moments of inertia at end 1 about the Z and Y axes
4, 5	TKZB1, TKYB1	Bottom thickness at end 1 in the Z and Y directions
6	IX1	Torsional moment of inertia at end 1
7	AREA2	Cross-sectional area at end 2 (node J)
8, 9	IZ2, IY2	Moments of inertia at end 2 about the Z and Y axes
10, 11	TKZB2, TKYB2	Bottom thickness at end 2 in the Z and Y directions
12	IX2	Torsional moment of inertia at end 2
13, 14, 15	DX1, DY1, DZ1	X, Y, and Z offsets at end 1 (node I)
16, 17, 18	DX2, DY2, DZ2	X, Y, and Z offsets at end 2 (node J)
19, 20	SHEARZ, SHEARY	Shear deflection constant in the Z and Y directions
21,22	TKZT1, TKYT1	Top thickness at end 1 (node I) in the Z and Y directions
23, 24	TKZT2, TKYT2	Top thickness at end 2 (node J) in the Z and Y directions
25, 26	ARESZ1, ARESY1	Shear areas at end 1 (node I) in the Z and Y directions
27, 28	ARESZ2, ARESY2	Shear areas at end 2 (node J) in the Z and Y directions
29. 30	TSF1, TSF2	Torsional stress factor at each end about the Z and Y axes
31, 32	DSCZ1, DSCY1	Shear center offset at end 1 (node I) in the Z and Y directions
33, 34	DSCZ2, DSCY2	Shear center offset at end 2 (node J) in the Z and Y directions
35, 36	EFSZ, EFSY	Foundation stiffnesses in the Z and Y directions
37, 38	Y1, Z1	Y, Z coordinate set 1 for additional stress output at end 1 (node I)
39, 40	Y2, Z2	Y, Z coordinate set 2 for additional stress output at end 1 (node I)
41, 42	Y3, Z3	Y, Z coordinate set 3 for additional stress output at end 1 (node I)
43, 44	Y4, Z4	Y, Z coordinate set 4 for additional stress output at end 1 (node I)
45, 46	Y1, Z1	Y, Z coordinate set 1 for additional stress output at end 2 (node J)

No.	Name	Description
47, 48	Y2, Z2	Y, Z coordinate set 2 for additional stress output at end 2 (node J)
49, 50	Y3, Z3	Y, Z coordinate set 3 for additional stress output at end 2 (node J)
51, 52	Y4, Z4 (at end J)	Y, Z coordinate set 4 for additional stress output at end 2 (node J)
53	THETA	Element X-axis rotation
54	ISTRN	Initial strain in element
55	ADDMAS	Added mass/unit length

BEAM44 Output Data

The solution output associated with the element is in two forms:

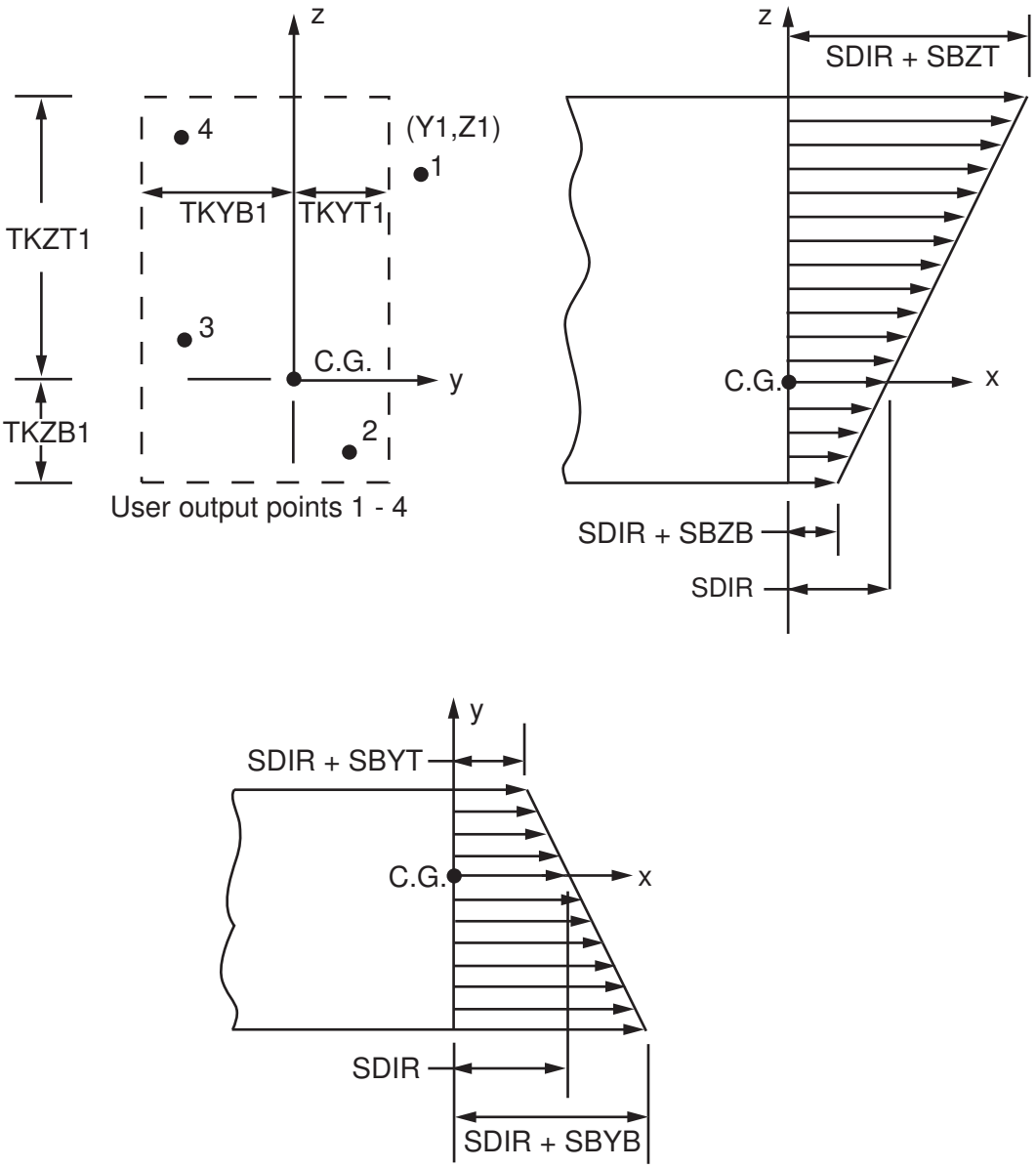
- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "BEAM44 Element Output Definitions"*

Several items are illustrated in *Figure 2, "BEAM44 Stress Output"*. At each cross-section, the computed output consists of the direct (axial) stress and four bending components. Then these five values are combined to evaluate maximum and minimum stresses, assuming a rectangular cross section. If constants Y1 through Z4 are provided, the combined stresses at the specified locations shown in *Figure 2, "BEAM44 Stress Output"* will also be computed. If KEYOPT(6) = 1 for this element, the 12 member forces and moments (6 at each end) are also printed (in the element coordinate directions). The element x-axis is defined through the center of gravity of the cross section. If real constants 25 through 30 (ARES_ _, TSF_) are provided, the average shear stresses and the torsional stresses are printed. If they are all zero, the shear printout is suppressed. Additional results at intermediate locations between the ends may be output with KEYOPT(9). A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Note

If **/ESHAPE**,1 has been specified, 3-D plotting of BEAM44 elements is supported in the ANSYS preprocessor *only*. 3-D plotting of BEAM44 elements is *not* supported in the ANSYS postprocessor.

Figure 2 BEAM44 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 BEAM44 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y

Name	Definition	O	R
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*5*
TEMP	Temperatures T1, T2, T3, T4, T5, T6, T7, T8	Y	Y
PRES	Pressures P1 at nodes I, J; OFFST1 at I, J; P2 at I, J; OFFST2 at I, J; P3 at I, J; OFFST3 at I, J; P4 at I; P5 at J	Y	Y
SDIR	Axial direct stress	*1*	*1*
SBYT	Bending stress on the element +Y side of the beam	*1*	*1*
SBYB	Bending stress on the element -Y side of the beam	*1*	*1*
SBZT	Bending stress on the element +Z side of the beam	*1*	*1*
SBZB	Bending stress on the element -Z side of the beam	*1*	*1*
SMAX	Maximum stress (direct stress + bending stress)	*1*	*1*
SMIN	Minimum stress (direct stress - bending stress)	*1*	*1*
EPELDIR	Axial elastic strain at the end	*1*	*1*
EPELBYT	Bending elastic strain on the element +Y side of the beam	*1*	*1*
EPELBYB	Bending elastic strain on the element -Y side of the beam	*1*	*1*
EPELBZT	Bending elastic strain on the element +Z side of the beam	*1*	*1*
EPELBZB	Bending elastic strain on the element -Z side of the beam	*1*	*1*
EPTHDIR	Axial thermal strain at the end	*1*	*1*
EPTHBYT	Bending thermal strain on the element +Y side of the beam	*1*	*1*
EPTHBYB	Bending thermal strain on the element -Y side of the beam	*1*	*1*
EPTHBZT	Bending thermal strain on the element +Z side of the beam	*1*	*1*
EPTHBZB	Bending thermal strain on the element -Z side of the beam	*1*	*1*
EPINAXL	Initial axial strain in the element	*1*	*1*
S(XY, XZ, YZ)	Average shear (Y-direction), average shear (Z-direction), torsion stresses	*2*	*2*
S(AXL1, AXL2, AXL3, AXL4)	Combined stresses at user points 1, 2, 3 and 4	*3*	*3*
MFOR(X, Y, Z)	Member forces in the element coordinate system X, Y, Z directions	*4*	Y
MMOM(X, Y, Z)	Member moments in the element coordinate system X, Y, Z directions	*4*	Y

1. The item repeats for end I, intermediate locations (see KEYOPT(9)), and end J
2. Output only if real constants 25-30 are provided
3. Output only if real constants 37-52 are provided
4. If KEYOPT(6) = 1
5. Available only at centroid as a ***GET** item.

Table 3, "BEAM44 Item and Sequence Numbers (KEYOPT(9) = 0)" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "BEAM44 Item and Sequence Numbers (KEYOPT(9) = 0)"*:

Name
output quantity as defined in the *Table 2, "BEAM44 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

IL*n*
sequence number for data at Intermediate Location *n*

SP*n*
solution items for Stress Point *n*

Table 3 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 0)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDIR	LS	-	1	6
SBYT	LS	-	2	7
SBYB	LS	-	3	8
SBZT	LS	-	4	9
SBZB	LS	-	5	10
EPELDIR	LEPEL	-	1	6
EPELBYT	LEPEL	-	2	7
EPELBYB	LEPEL	-	3	8
EPELBZT	LEPEL	-	4	9
EPELBZB	LEPEL	-	5	10
EPTHDIR	LEPTH	-	1	6
EPTHBYT	LEPTH	-	2	7
EPTHBYB	LEPTH	-	3	8
EPTHBZT	LEPTH	-	4	9
EPTHBZB	LEPTH	-	5	10
EPINAXL	LEPTH	11	-	-
SMAX	NMISC	-	1	3
SMIN	NMISC	-	2	4
MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MFORZ	SMISC	-	3	9
MMOMX	SMISC	-	4	10
MMOMY	SMISC	-	5	11
MMOMZ	SMISC	-	6	12
SXY	SMISC	-	13	16
SXZ	SMISC	-	14	17
SYZ	SMISC	-	15	18
P1	SMISC	-	27	28
OFFST1	SMISC	-	29	30
P2	SMISC	-	31	32

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
OFFST2	SMISC	-	33	34
P3	SMISC	-	35	36
OFFST3	SMISC	-	37	38
P4	SMISC	-	39	
P5	SMISC	-		40
SAXL (SP1)	SMISC	-	19	23
SAXL (SP2)	SMISC	-	20	24
SAXL (SP3)	SMISC	-	21	25
SAXL (SP4)	SMISC	-	22	26

		Corner Location							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 4 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 1)

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	IL1	J
SDIR	LS	-	1	6	11
SBYT	LS	-	2	7	12
SBYB	LS	-	3	8	13
SBZT	LS	-	4	9	14
SBZB	LS	-	5	10	15
EPELDIR	LEPEL	-	1	6	11
EPELBYT	LEPEL	-	2	7	12
EPELBYB	LEPEL	-	3	8	13
EPELBZT	LEPEL	-	4	9	14
EPELBZB	LEPEL	-	5	10	15
EPTHDIR	LEPTH	-	1	6	11
EPTHBYT	LEPTH	-	2	7	12
EPTHBYB	LEPTH	-	3	8	13
EPTHBZT	LEPTH	-	4	9	14
EPTHBZB	LEPTH	-	5	10	15
EPINAXL	LEPTH	16	-	-	-
SMAX	NMISC	-	1	3	5
SMIN	NMISC	-	2	4	6
MFORX	SMISC	-	1	7	13
MFORY	SMISC	-	2	8	14
MFORZ	SMISC	-	3	9	15
MMOMX	SMISC	-	4	10	16
MMOMY	SMISC	-	5	11	17
MMOMZ	SMISC	-	6	12	18

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	IL1	J
SXY	SMISC	-	19	22	25
SXZ	SMISC	-	20	23	26
SYZ	SMISC	-	21	24	27
P1	SMISC	-	40	-	41
OFFST1	SMISC	-	42	-	43
P2	SMISC	-	44	-	45
OFFST2	SMISC	-	46	-	47
P3	SMISC	-	48	-	49
OFFST3	SMISC	-	50	-	51
P4	SMISC	-	52	-	-
P5	SMISC	-	-	-	53
SAXL (SP1)	SMISC	-	28	32	36
SAXL (SP2)	SMISC	-	29	33	37
SAXL (SP3)	SMISC	-	30	34	38
SAXL (SP4)	SMISC	-	31	35	39

		Corner Location							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 5 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 3)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
SDIR	LS	-	1	6	11	16	21
SBYT	LS	-	2	7	12	17	22
SBYB	LS	-	3	8	13	18	23
SBZT	LS	-	4	9	14	19	24
SBZB	LS	-	5	10	15	20	25
EPELDIR	LEPEL	-	1	6	11	16	21
EPELBYT	LEPEL	-	2	7	12	17	22
EPELBYB	LEPEL	-	3	8	13	18	23
EPELBZT	LEPEL	-	4	9	14	19	24
EPELBZB	LEPEL	-	5	10	15	20	25
EPTHDIR	LEPTH	-	1	6	11	16	21
EPTHBYT	LEPTH	-	2	7	12	17	22
EPTHBYB	LEPTH	-	3	8	13	18	23
EPTHBZT	LEPTH	-	4	9	14	19	24
EPTHBZB	LEPTH	-	5	10	15	20	25
EPINAXL	LEPTH	26	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9
SMIN	NMISC	-	2	4	6	8	10

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
MFORX	SMISC	-	1	7	13	19	25
MFORY	SMISC	-	2	8	14	20	26
MFORZ	SMISC	-	3	9	15	21	27
MMOMX	SMISC	-	4	10	16	22	28
MMOMY	SMISC	-	5	11	17	23	29
MMOMZ	SMISC	-	6	12	18	24	30
SXY	SMISC	-	31	34	37	40	43
SXZ	SMISC	-	32	35	38	41	44
SYZ	SMISC	-	33	36	39	42	45
P1	SMISC	-	66	-	-	-	67
OFFST1	SMISC	-	68	-	-	-	69
P2	SMISC	-	70	-	-	-	71
OFFST2	SMISC	-	72	-	-	-	73
P3	SMISC	-	74	-	-	-	75
OFFST3	SMISC	-	76	-	-	-	77
P4	SMISC	-	78	-	-	-	-
P5	SMISC	-	-	-	-	-	79
SAXL (SP1)	SMISC	-	46	50	54	58	62
SAXL (SP2)	SMISC	-	47	51	55	59	63
SAXL (SP3)	SMISC	-	48	52	56	60	64
SAXL (SP4)	SMISC	-	49	53	57	61	65

		Corner Location							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 6 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 5)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	I	IL1	IL2	IL3	IL4	IL5	J
SDIR	LS	-	1	6	11	16	21	26	31
SBYT	LS	-	2	7	12	17	22	27	32
SBYB	LS	-	3	8	13	18	23	28	33
SBZT	LS	-	4	9	14	19	24	29	34
SBZB	LS	-	5	10	15	20	25	30	35
EPELDIR	LEPEL	-	1	6	11	16	21	26	31
EPELBYT	LEPEL	-	2	7	12	17	22	27	32
EPELBYB	LEPEL	-	3	8	13	18	23	28	33
EPELBZT	LEPEL	-	4	9	14	19	24	29	34
EPELBZB	LEPEL	-	5	10	15	20	25	30	35
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	I	IL1	IL2	IL3	IL4	IL5	J
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35
EPINAXL	LEPTH	36	-	-	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9	11	13
SMIN	NMISC	-	2	4	6	8	10	12	14
MFORX	SMISC	-	1	7	13	19	25	31	37
MFORY	SMISC	-	2	8	14	20	26	32	38
MFORZ	SMISC	-	3	9	15	21	27	33	39
MMOMX	SMISC	-	4	10	16	22	28	34	40
MMOMY	SMISC	-	5	11	17	23	29	35	41
MMOMZ	SMISC	-	6	12	18	24	30	36	42
SXY	SMISC	-	43	46	49	52	55	58	61
SXZ	SMISC	-	44	47	50	53	56	59	62
SYZ	SMISC	-	45	48	51	54	57	60	63
P1	SMISC	-	92	-	-	-	-	-	93
OFFST1	SMISC	-	94	-	-	-	-	-	95
P2	SMISC	-	96	-	-	-	-	-	97
OFFST2	SMISC	-	98	-	-	-	-	-	99
P3	SMISC	-	100	-	-	-	-	-	101
OFFST3	SMISC	-	102	-	-	-	-	-	103
P4	SMISC	-	104	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	105
SAXL (SP1)	SMISC	-	64	68	72	76	80	84	88
SAXL (SP2)	SMISC	-	65	69	73	77	81	85	89
SAXL (SP3)	SMISC	-	66	70	74	78	82	86	90
SAXL (SP4)	SMISC	-	67	71	75	79	83	87	91

		Corner Location							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 7 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 7)

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
SDIR	LS	-	1	6	11	16	21	26	31	36	41
SBYT	LS	-	2	7	12	17	22	27	32	37	42
SBYB	LS	-	3	8	13	18	23	28	33	38	43
SBZT	LS	-	4	9	14	19	24	29	34	39	44
SBZB	LS	-	5	10	15	20	25	30	35	40	45
EPELDIR	LEPEL	-	1	6	11	16	21	26	31	36	41

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
EPELBYT	LEPEL	-	2	7	12	17	22	27	32	37	42
EPELBYB	LEPEL	-	3	8	13	18	23	28	33	38	43
EPELBZT	LEPEL	-	4	9	14	19	24	29	34	39	44
EPELBZB	LEPEL	-	5	10	15	20	25	30	35	40	45
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31	36	41
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32	37	42
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33	38	43
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34	39	44
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35	40	45
EPINAXL	LEPTH	46	-	-	-	-	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50
MFORZ	SMISC	-	3	9	15	21	27	33	39	45	51
MMOMX	SMISC	-	4	10	16	22	28	34	40	46	52
MMOMY	SMISC	-	5	11	17	23	29	35	41	47	53
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54
SXY	SMISC	-	55	58	61	64	67	70	73	76	79
SXZ	SMISC	-	56	59	62	65	68	71	74	77	80
SYZ	SMISC	-	57	60	63	66	69	72	75	78	81
P1	SMISC	-	118	-	-	-	-	-	-	-	119
OFFST1	SMISC	-	120	-	-	-	-	-	-	-	121
P2	SMISC	-	122	-	-	-	-	-	-	-	123
OFFST2	SMISC	-	124	-	-	-	-	-	-	-	125
P3	SMISC	-	126	-	-	-	-	-	-	-	127
OFFST3	SMISC	-	128	-	-	-	-	-	-	-	129
P4	SMISC	-	130	-	-	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	-	-	131
SAXL (SP1)	SMISC	-	82	86	90	94	98	102	106	110	114
SAXL (SP2)	SMISC	-	83	87	91	95	99	103	107	111	115
SAXL (SP3)	SMISC	-	84	88	92	96	100	104	108	112	116
SAXL (SP4)	SMISC	-	85	89	93	97	101	105	109	113	117

		Corner Location							
		1	2	3	4	5	6	7	8
TEMP	LBFE	1	2	3	4	5	6	7	8

Table 8 BEAM44 Item and Sequence Numbers (KEYOPT(9) = 9)

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SDIR	LS	-	1	6	11	16	21	26	31	36	41	46	51
SBYT	LS	-	2	7	12	17	22	27	32	37	42	47	52
SBYB	LS	-	3	8	13	18	23	28	33	38	43	48	53
SBZT	LS	-	4	9	14	19	24	29	34	39	44	49	54
SBZB	LS	-	5	10	15	20	25	30	35	40	45	50	55
EPELDIR	LEPEL	-	1	6	11	16	21	26	31	36	41	46	51
EPELBYT	LEPEL	-	2	7	12	17	22	27	32	37	42	47	52
EPELBYB	LEPEL	-	3	8	13	18	23	28	33	38	43	48	53
EPELBZT	LEPEL	-	4	9	14	19	24	29	34	39	44	49	54
EPELBZB	LEPEL	-	5	10	15	20	25	30	35	40	45	50	55
EPTHDIR	LEPTH	-	1	6	11	16	21	26	31	36	41	46	51
EPTHBYT	LEPTH	-	2	7	12	17	22	27	32	37	42	47	52
EPTHBYB	LEPTH	-	3	8	13	18	23	28	33	38	43	48	53
EPTHBZT	LEPTH	-	4	9	14	19	24	29	34	39	44	49	54
EPTHBZB	LEPTH	-	5	10	15	20	25	30	35	40	45	50	55
EPINAXL	LEPTH	56	-	-	-	-	-	-	-	-	-	-	-
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17	19	21
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18	20	22
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49	55	61
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50	56	62
MFORZ	SMISC	-	3	9	15	21	27	33	39	45	51	57	63
MMOMX	SMISC	-	4	10	16	22	28	34	40	46	52	58	64
MMOMY	SMISC	-	5	11	17	23	29	35	41	47	53	59	65
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54	60	66
SXY	SMISC	-	67	70	73	76	79	82	85	88	91	94	97
SXZ	SMISC	-	68	71	74	77	80	83	86	89	92	95	98
SYZ	SMISC	-	69	72	75	78	81	84	87	90	93	96	99
P1	SMISC	-	144	-	-	-	-	-	-	-	-	-	145
OFFST1	SMISC	-	146	-	-	-	-	-	-	-	-	-	147
P2	SMISC	-	148	-	-	-	-	-	-	-	-	-	149
OFFST2	SMISC	-	150	-	-	-	-	-	-	-	-	-	151
P3	SMISC	-	152	-	-	-	-	-	-	-	-	-	153
OFFST3	SMISC	-	154	-	-	-	-	-	-	-	-	-	155
P4	SMISC	-	156	-	-	-	-	-	-	-	-	-	-
P5	SMISC	-	-	-	-	-	-	-	-	-	-	-	157
SAXL (SP1)	SMISC	-	100	104	108	112	116	120	124	128	132	136	140
SAXL (SP2)	SMISC	-	101	105	109	113	117	121	125	129	133	137	141
SAXL (SP3)	SMISC	-	102	106	110	114	118	122	126	130	134	138	142
SAXL (SP4)	SMISC	-	103	107	111	115	119	123	127	131	135	139	143

		Corner Location							
		1	**2**	**3**	**4**	**5**	**6**	**7**	**8**
TEMP	LBFE	1	2	3	4	5	6	7	8

BEAM44 Assumptions and Restrictions

- The beam must not have a zero length, area, or moment of inertia.
- Because shear area is not calculated when using section properties to create BEAM44, no shear stresses will be output. Use of BEAM188 or BEAM189 to output and visualize shear stresses.
- The element thicknesses are used for locating the extreme fibers for the stress calculations and for computing the thermal gradient.
- Tapers within an element, if any, should be gradual. If AREA2/AREA1 or I_2/I_1 is not between 0.5 and 2.0, a warning message is output. If the ratio is outside of the range of 0.1 to 10.0, an error message is output. The element should not taper to a point (zero thickness).
- The applied thermal gradients are assumed to be linear across the thickness in both directions and along the length of the element.
- The flexible length of the beam is adjusted to account for the effect of the offsets. The offset lengths may be regarded as rigid portions of the beam. Unequal lateral offsets, which rotate the beam, also cause a corresponding shortening of the flexible length. The difference between the lateral offsets should not exceed the length of the element. Rotational body forces resulting from an angular velocity are based upon the node locations (as if zero offsets).
- The shear stresses are calculated based on the shear force rather than the shear deflection
- BEAM44 can be used with any cross section that was defined using **SECTYPE**, **SECDATA**, **SECOFFSET**, **SECWRITE**, and **SECREAD**. However, a section defined with these commands will be used *only* if there is no real constant set defined.
- An unsymmetric section, defined using **SECTYPE**, **SECDATA**, **SECOFFSET**, **SECWRITE**, and **SECREAD**, must be transformed into a principal coordinate system. This system is defined by the real constant THETA. Real Constants and Output Data are calculated in the principal coordinate system. Note, a small peturbation of a regular polygon may result in a large rotation of the principal axes.
- Unlike BEAM188 and BEAM189, numerical integration is not performed through the cross section when BEAM44 elements are used.
- If you have issued an **/ESHAPE**,1 command: With the exception of displaced shape and expanded element plots, 3-D plots of BEAM44 elements with section definition (instead of real constants) do not support contour data in the ANSYS postprocessor.
- A lumped mass matrix formulation [**LUMPM**,ON] is not allowed for this element when using member releases in the element translational Y or Z directions. In addition, the effect of offsets on the mass matrix is ignored if the lumped mass formulation is on.

BEAM44 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special features allowed are stress stiffening and large deflection.

SOLID45

3-D Structural Solid

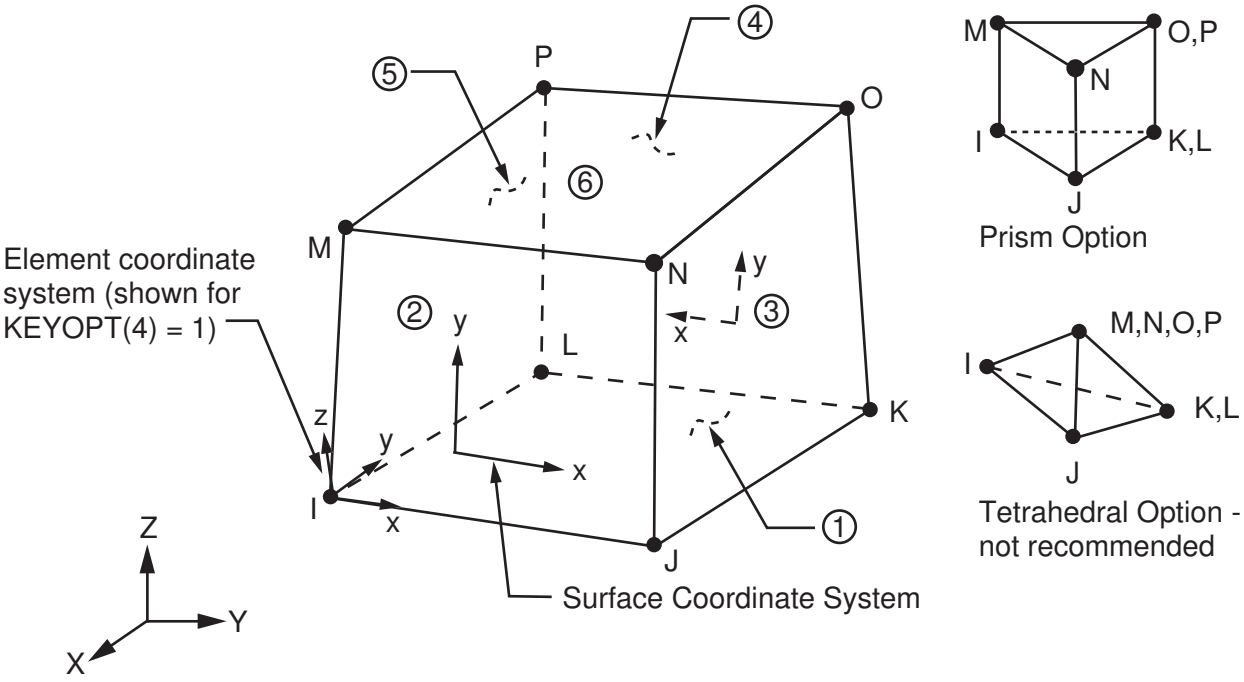
MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

SOLID45 Element Description

SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. A reduced integration option with hourglass control is available. See SOLID45 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A higher-order version of the SOLID45 element is SOLID95.

Figure 1 SOLID45 Geometry



SOLID45 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID45 Geometry"*. The element is defined by eight nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID45 Geometry"*. Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

KEYOPT(1) is used to include or suppress the extra displacement shapes. KEYOPT(5) and KEYOPT(6) provide various element printout options (see *Section 2.2.2: Element Solution*).

This element also supports uniform reduced (1 point) integration with hourglass control when KEYOPT(2) = 1. Using uniform reduced integration provides the following advantages when running a nonlinear analysis:

- Less cpu time is required for element stiffness formation and stress/strain calculations to achieve a comparable accuracy to the FULL integration option.
- The length of the element history saved record (.ESAV and .OSAV) is about 1/7th as much as when the full integration (2 X 2 X 2) is used for the same number of elements.
- Nonlinear convergence characteristic of the option is generally far superior to the default full integration with extra displacement shape; that is, KEYOPT(1) = 0, KEYOPT(2) = 0.
- The analysis will not suffer from volumetric locking which can be caused by plasticity or other incompressible material properties.

An analysis using uniform reduced integration can have the following disadvantages:

- The analysis is not as accurate as the full integration method, which is apparent in the linear analysis for the same mesh.
- The analysis cannot capture the bending behavior with a single layer of elements; for example, in the case of a fixed-end cantilever with a lateral point load, modeled by one layer of elements laterally. Instead, four elements are usually recommended.

When the uniform reduced integration option is used (KEYOPT(2) = 1 - this option is the same as SOLID185 with KEYOPT(2) = 1), you can check the accuracy of the solution by comparing the total energy (SENE label in **ETABLE**) and the artificial energy (AENE label in **ETABLE**) introduced by hourglass control. If the ratio of artificial energy to total energy is less than 5%, the solution is generally acceptable. If the ratio exceeds 5%, refine the mesh. The total energy and artificial energy can also be monitored by using the **OUTPR**,VENG command in the solution phase. For more details, see the *Theory Reference for ANSYS and ANSYS Workbench*.

Initial state conditions previously handled via the **ISTRESS** command will be discontinued for this element. The **INISTATE** command will provide increased functionality, but only via the Current Technology elements (180,181, etc.). To continue using Initial State conditions in future versions of ANSYS, consider switching to the appropriate Current Technology element. For more information on setting Initial State values see the **INISTATE** command and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. For more information on current -vs- legacy element technologies see *Section 2.17: Legacy vs. Current Element Technologies* in the *Elements Reference*

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID45 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID45 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
HGSTF - Hourglass control factor needed only when KEYOPT(2) = 1.

Note

The valid value for this real constant is any positive number; default = 1.0. We recommend that you use a value between 1 and 10.

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Fluences --
FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO, CHABOCHE, HILL)
Creep (CREEP, RATE)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent
Initial stress import

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)

Include or suppress extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(2)

Integration option:

0 --
Full integration with or without extra displacement shapes, depending on the setting of KEYOPT(1)

1 --
Uniform reduced integration with hourglass control; suppress extra displacement shapes (KEYOPT(1) is automatically set to 1).

KEYOPT(4)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points

2 --
Nodal Stress Solution

KEYOPT(6)
Extra surface output:

0 --
Basic element solution

1 --
Surface solution for face I-J-N-M also

2 --
Surface solution for face I-J-N-M and face K-L-P-O (Surface solution available for linear materials only)

3 --
Include nonlinear solution at each integration point

4 --
Surface solution for faces with nonzero pressure

KEYOPT(9)
Initial stress subroutine option (available only through direct input of the **KEYOPT** command):

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)

SOLID45 Output Data

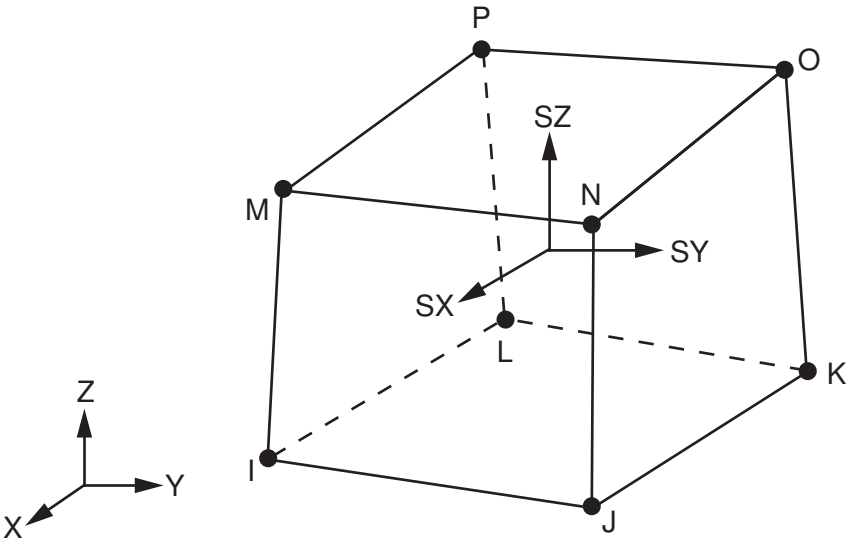
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID45 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID45 Stress Output"*. The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate systems and are available for any face (KEYOPT(6)). The coordinate systems for faces IJNM and KLPO are shown in *Figure 1, "SOLID45 Geometry"*. The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are

met. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID45 Stress Output



Stress directions shown are for KEYOPT(4) = 0

When KEYOPT(2) = 1 (the element is using uniform reduced integration), all the outputs for the element integration points are output in the same style as the full integration outputs. The number of points for full integration is used for consistency of output within the same element type.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID45 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y

Name	Definition	O	R
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [*4*]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strains	-	*5*
EPTH:EQV	Equivalent thermal strain [*4*]	-	*5*
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	*1*	*1*
EPPL:EQV	Equivalent plastic strain [*4*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strain [*4*]	*1*	*1*
EPSW:	Average swelling strain	*1*	*1*
NL:EPEQ	Average equivalent plastic strain	*1*	*1*
NL:SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
NL:SEPL	Average equivalent stress from stress-strain curve	*1*	*1*
NL:HPRES	Hydrostatic pressure		*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
TEMP	Surface average temperature	*2*	*2*
EPEL	Surface elastic strains (X ,Y, XY)	*2*	*2*
PRESS	Surface pressure	*2*	*2*
S(X, Y, XY)	Surface stresses (X-axis parallel to line defined by first two nodes which define the face)	*2*	*2*
S(1, 2, 3)	Surface principal stresses	*2*	*2*
SINT	Surface stress intensity	*2*	*2*
SEQV	Surface equivalent stress	*2*	*2*
LOCI:X, Y, Z	Integration point locations	-	Y

1. Nonlinear solution, output only if the element has a nonlinear material
2. Surface output (if KEYOPT(6) is 1, 2, or 4)
3. Available only at centroid as a ***GET** item
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
5. Output only if element has a thermal load.

Table 2 SOLID45 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Pt. Solution	EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR, EPSW	*1*	-
Integration Point Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV, EPEL	*2*	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV, EPEL	*3*	-

1. Output at each of eight integration points, if the element has a nonlinear material and KEYOPT(6) = 3

2. Output at each integration point, if KEYOPT(5) = 1
3. Output at each node, if KEYOPT(5) = 2

Table 3, "SOLID45 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. SeeThe General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "SOLID45 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID45 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I,J,...,P

Table 3 SOLID45 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40
FLUEN	NMISC	41	42	43	44	45	46	47	48

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for the **ETABLE** command.

SOLID45 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1, "SOLID45 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
  - A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
  - A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

SOLID45 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads are not applicable.
- The only special feature allowed is stress stiffening.
- KEYOPT(6) = 3 is not applicable.

SOLID46

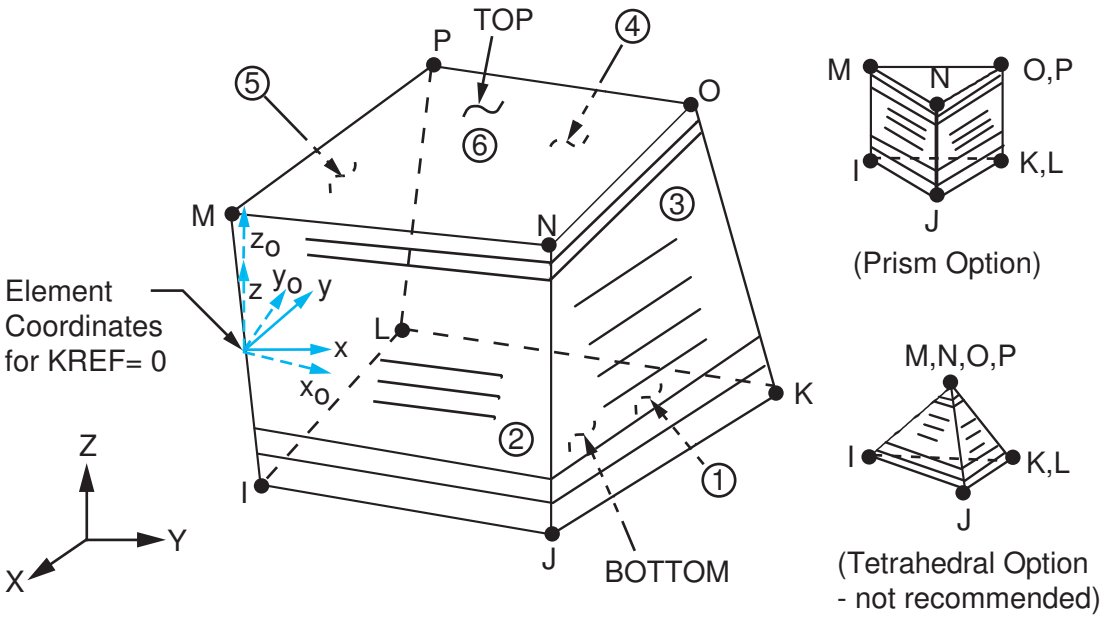
3-D 8-Node Layered Structural Solid

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SOLID46 Element Description

SOLID46 is a layered version of the 8-node structural solid (SOLID45) designed to model layered thick shells or solids. The element allows up to 250 different material layers. If more than 250 layers are required, a user-input constitutive matrix option is available. The element may also be stacked as an alternative approach. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions. See SOLID46 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A similar element for shells is SHELL99 .

Figure 1 SOLID46 Geometry



x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID46 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID46 Geometry"*. The element is defined by eight nodes, layer thicknesses, layer material direction angles, and orthotropic material properties. Shear moduli GXZ and GYZ must be within a factor of 10,000 of each other.

The element z-axis is defined to be normal to a flat reference plane, using real constant KREF as shown in *Figure 2, "SOLID46 Stress Output"*. KREF may have values of 0 (midplane), 1 (bottom), or 2 (top). If the nodes imply a warped surface, an averaged flat plane is used. The default element x-axis is the projection of side I-J, side M-N, or their average (depending on KREF) onto the reference plane. The orientation within the plane of the layers may be changed using **ESYS** in the same way it is used for shell elements as described in *Section 2.3: Coordinate Systems*. To reorient the elements (after automatic meshing) you should use **EORIENT**. With **EORIENT**, you can make

SOLID46 elements match an element whose orientation is as desired, or set the orientation to be as parallel as possible to a defined axis.

The input may be either in matrix form or layer form, depending upon KEYOPT(2). For matrix form, the matrices must be computed outside of ANSYS. The force-strain and moment-curvature relationships defining the matrices for a quadratic variation of strain through the thickness (KEYOPT(2) = 3) may be defined as described in *SHELL99 Input Data* for the 8-node linear layered shell (SHELL99). Also, references to midside nodes should be ignored for this element. Thermal strains, most stresses, and failure criteria are not available with matrix input.

For layer (non-matrix) input, the total number of layers must be specified (NL). If KEYOPT(2) = 0, the maximum number of layers is 250; if KEYOPT(2) = 1, the maximum is 125. The properties of all layers should be entered (LSYM = 0). If the properties of the layers are symmetrical about the midthickness of the element (LSYM = 1), only half of properties of the layers, up to and including the middle layer (if any), need to be entered. While all layers may be printed, two layers may be specifically selected to be output (LP1 and LP2, with LP1 usually less than LP2).

Each layer of the layered solid element may have a variable thickness (TK). The thickness is assumed to vary bilinearly over the area of the layer, with the thickness input at the corner node locations. If a layer has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four corner thicknesses must be input using positive values. Zero thickness layers may be used to model dropped plies. The layer thicknesses used are computed by scaling the input real constant thicknesses to be consistent with the thicknesses between the nodes.

The node locations may imply that the layers are tilted or warped. However, the local coordinate system for each layer is effectively reoriented parallel to the reference plane, as shown in *Figure 2, "SOLID46 Stress Output"*. In this local right-handed system, the x'-axis is rotated an angle THETA(LN) (in degrees) from the element x-axis toward the element y-axis.

The material properties of each layer may be orthotropic in the plane of the element. The real constant MAT is used to define the layer material number instead of the element material number applied with **MAT**. MAT defaults to 1 if not input. The material X direction corresponds to the local layer x' direction.

Use **TREF** and **BETAD** to supply global values for reference temperature and damping, respectively. Alternatively, use **MAT** to specify element-dependent values for reference temperature (**MP**,REFT) or damping (**MP**,DAMP); layer material numbers are ignored for this purpose.

The total number of layers must be specified with the NL real constant as described in *SHELL99 Input Data* for SHELL99. The real constants, material properties, layer thicknesses, and failure criteria are also described in *SHELL99 Input Data* for SHELL99.

The failure criteria selection is input in the data table [**TB**], as described in *Table 2.2, "Orthotropic Material Failure Criteria Data"*. Three predefined criteria are available and up to six user-defined criteria may be entered with user subroutines. See Failure Criteria in the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the three predefined failure criteria. See *Guide to ANSYS User Programmable Features* for an explanation of user subroutines. Failure criteria may also be computed in POST1 (using the **FC** commands). All references to failure criteria as part of element output data are based only on the **TB** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID46 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If only T(I) and T(J) are input, T(I) is used for T(I), T(J), T(K), and T(L), while T(J) (as input) is used for T(M), T(N), T(O), and T(P). For any other input pattern, unspecified temperatures default to TUNIF.

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID46 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For more information on Failure Criteria, see Composites in the *Structural Analysis Guide*.

SOLID46 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
The real constants vary, depending on the KEYOPT(2) setting. For descriptions of the real constants, see:

Table 1, "SOLID46 Real Constants (KEYOPT(2) = 0 or 1)"
Table 2, "SOLID46 Real Constants (KEYOPT(2) = 3)"

Material Properties
If KEYOPT(2) = 0 or 1, supply the following 13*NM properties where NM is the number of materials (maximum is NL):

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
(PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, for each of the NM materials

If KEYOPT(2) = 3, supply none of the above.

Supply DAMP and REFT only once for the element (use **MAT** to assign material property set). See the discussion in *SOLID46 Input Data* for more details.

Surface Loads

Pressure --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P) if KEYOPT(2) = 0 or 1, or none if KEYOPT(2) = 3

Special Features

Stress stiffening
Large deflection

KEYOPT(1)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(2)
Form of input:

0 --
Constant thickness layer input (250 layers maximum)

1 --
Tapered layer input (125 layers maximum)

3 --
Matrix input using quadratic logic (see *SHELL99 Input Data*)

KEYOPT(3)
Extra element output:

0 --
Basic element printout

1 --
Integration point strain printout at bottom and top surfaces of element

2 --
Nodal force printout in element coordinates

4 --
Combination of both options

KEYOPT(4)
Element coordinate system:

0 --
No user subroutines to define element coordinate system

4 --
Element x-axis located by user subroutine USERAN

5 --
Element x-axis located by user subroutine USERAN and layer x-axes located by user subroutine USANLY

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(5)
Determines whether strains or stresses will be used with KEYOPT(6):

0 --
Use strain results

1 --
Use stress results

2 --
Use both strain and stress results

KEYOPT(6)
Printout control:

0 --
Basic element printout, as well as the summary of the maximum of all the failure criteria

1 --
Same as 0, and also print the summary of all the failure criteria, average transverse shear stresses, and the summary of the maximum interlaminar shear stress

2 --
Same as 1, and also print the layer solution at the integration points in the bottom layer (or LP1) and the top layer (or LP2)

3 --
Same as 1, and also print the layer solution at the element centroid for all layers

4 --
Same as 1, and also print the layer solution at the corners for all layers

5 --
Same as 1, and also print the layer solution with the failure criterion values at the integration points for all layers

Note

Thermal strains, most stresses, and failure criteria are not available with matrix input.

KEYOPT(8)
Storage of layer data:

0 --
Store data for bottom of bottom layer (or LP1), for top of top layer (or LP2), and data for maximum failure criteria layer.

1 --
Store data for all layers.

Caution

Volume of data stored may be excessive.

KEYOPT(9)
Determines where strains, stresses, and failure criteria are evaluated (available only if KEYOPT(2) = 0 or 1 with NL > 1):

0 --
Evaluate strains and stresses at top and bottom of each layer

1 --
Evaluate at midthickness of each layer

KEYOPT(10)
Determines whether material property matrices are printed:

0 --
No material property matrices printed

1 --
Print material property matrices integrated through thickness for first element, if it is a SOLID46 element

For more information on real constants and other input data, see *SHELL91* . For more information on failure criteria, please refer to *Section 2.2.2.12: Failure Criteria*.

Table 1 SOLID46 Real Constants (KEYOPT(2) = 0 or 1)

No.	Name	Description
Basic constants for KEYOPT(2) = 0 or 1		

No.	Name	Description
1	NL	Number of layers (250 maximum)
2	LSYM	Layer symmetry key
3	LP1	First layer for output
4	LP2	Second layer for output
5, 6	(blank)	
7	KREF	Location of reference plane
8, ..., 12	(blank)	
KEYOPT(2) = 0, add these:		
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK	Layer thickness for layer 1
16, ... (12+3*NL)	MAT, THETA, TK, etc.	Repeat MAT, THETA, and TK for each layer specified (up to NL layers)
For KEYOPT(2) = 1, add these:		
13	MAT	Material number for layer 1
14	THETA	X-axis rotation for layer 1
15	TK(I)	Layer thickness at node I for layer 1
16	TK(J)	Layer thickness at node J for layer 1
17	TK(K)	Layer thickness at node K for layer 1
18	TK(L)	Layer thickness at node L for layer 1
19, ... (12+6*NL)	MAT, THETA, TK(I), etc.	Repeat MAT, THETA, TK(I), TK(J), TK(K), and TK(L) for each layer specified (up to NL layers)

Table 2 SOLID46 Real Constants (KEYOPT(2) = 3)

For KEYOPT(2) = 3, use these:		
1, ..., 21	A(1), ..., A(21)	Submatrix A
22, ..., 42	B(1), ..., B(21)	Submatrix B
43, ..., 63	D(1), ..., D(21)	Submatrix D
64, ..., 84	E(1), ..., E(21)	Submatrix E
85, ..., 105	F(1), ..., F(21)	Submatrix F
106, ..., 111	MT(1), ..., MT(6)	MT array
112, ..., 117	BT(1), ..., BT(6)	BT array
118, ..., 123	QT(1), ..., QT(6)	QT array
124	AVDENS	Element average density
125, 126, 127	(blank)	
128	KREF	Reference plane factor

SOLID46 Output Data

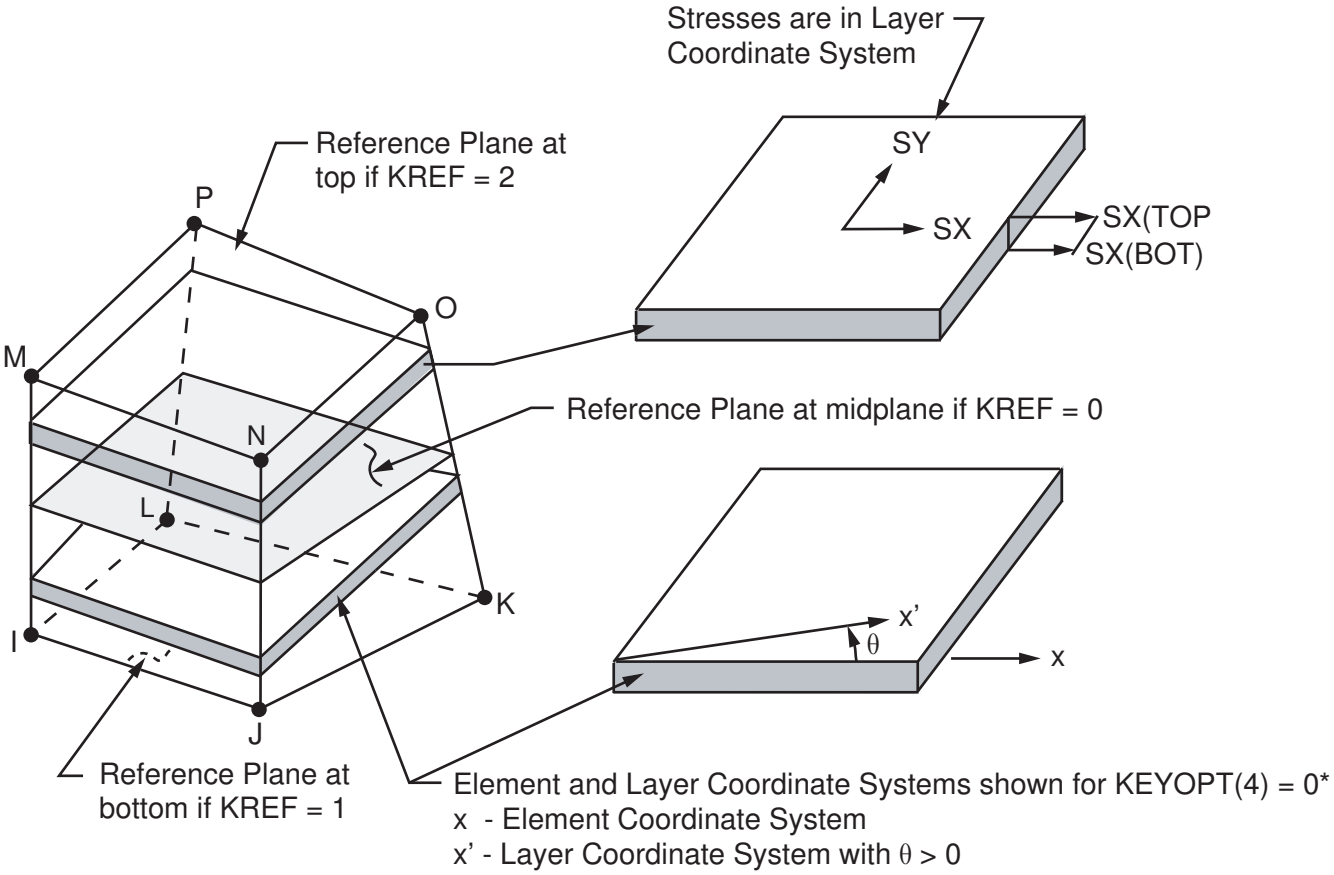
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 3, "SOLID46 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID46 Stress Output"*.

The element stress directions correspond to the layer local coordinate directions. Various layer printout options are available. For integration point output, integration point 1 is nearest Node I, 2 nearest J, 3 nearest K, and 4 nearest L. Failure criterion output is evaluated only at the in-plane integration points. (See the *Theory Reference for ANSYS and ANSYS Workbench*.) If KEYOPT(3) = 2 or 4 for this element, the three member forces and moments are also printed for each node (in the element coordinate system). KEYOPT(8) controls the amount of data output on the postdata file for processing with **LAYER** or **LAYERP26**. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID46 Stress Output



*Note: Layer Coordinate System x-y plane is parallel to the reference plane (KREF)

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 SOLID46 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
VOLU:	Volume	Y	Y
TTOP, TBOT	Average temperatures at top and bottom faces	*1*	-
XC, YC, ZC	Element centroid	Y	*11*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
INT	In-plane integration point number	*2*	-
POS	Top (TOP), bottom (BOT), midthickness (MID) of element	*2*	-
XI, YI, ZI	Global X, Y, Z location of integration point	*2*	-
NUMBER	Layer number	*1*, *3*	-
MAT	Material number of this layer	*3*	-
THETA	Material direction angle for layer (THETA)	*1*, *3*	-
AVE THICK	Average thickness of layer	*3*	-
ACC AVE THICK	Accumulative average thickness (Thickness of element from layer 1 to this layer)	*1*, *3*	-
AVE TEMP	Average temperature of layer	*3*	-
POS	Top (TOP), bottom (BOT), midthickness (MID) of layer (See KEYOPT(9) for control options)	*3*	-
LOC	Center location (AVG)	*1*, *4*	-
NODE	Corner node number	*1*, *5*	-
INT	Integration point number	*1*, *6*	-
S:X, Y, Z, XY, YZ, XZ	Stresses (in layer local coordinates)	*1*, *7*	*1*
S:1, 2, 3	Principal stresses	*1*, *7*	*1*
S:INT	Stress intensity	*1*, *7*	*1*
S:EQV	Equivalent stresses (in layer local coordinates)	*1*, *7*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains (in layer local coordinates). Total elastic strain if KEYOPT(2) = 2 or 3	*7*	Y
EPEL:EQV	Equivalent elastic strains (in layer local coordinates) [*12*]	*7*	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains (in layer local coordinates). Total thermal strain if KEYOPT(2) = 2 or 3	*7*	Y
EPTH:EQV	Equivalent thermal strains (in layer local coordinates) [*12*]	*7*	Y
FC1, ..., FC6, FCMAX	Failure criterion values and maximum at each integration point	*1*, *8*	-
FC	Failure criterion number (FC1 to FC6, FCMAX)	*1*, *9*	*1*
VALUE	Maximum value for this criterion (if value exceeds 9999.999, 9999.999 will be printed)	*1*, *9*	*1*
LN	Layer number where maximum occurs	*9*	*1*
EPELF(X, Y, Z, XY, YZ, XZ)	Elastic strains (in layer local coordinates) causing the maximum value for this criterion in the element.	*1*, *9*	*1*
SF(X, Y, Z, XY, YZ, XZ)	Stresses (in layer local coordinates) causing the maximum value for this criterion in the element	*1*, *9*	*1*

Name	Definition	O	R
ILSXZ	SXZ interlaminar shear stress	-	*1*
ILSYZ	SXZ interlaminar shear stress	-	*1*
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	*1*
ILSUM	Interlaminar shear stress vector sum	-	*1*
LN1, LN2	Layer numbers which define location of maximum interlaminar shear stress (ILMAX)	*1*, *10*	*1*
ILMAX	Maximum interlaminar shear stress (occurs between LN1 and LN2)	*1*, *10*	*1*

1. If KEYOPT(2) = 0 or 1
2. Integration point strain solution (printed only if KEYOPT(3) = 1 or 4)
3. Layer solution (printed only if KEYOPT(2) = 0 or 1 and KEYOPT(6) > 1)
4. If KEYOPT(6) = 3
5. If KEYOPT(6) = 4
6. If KEYOPT(6) = 2 or 5
7. The strain and stress output is controlled with KEYOPT(5)
8. Output only if KEYOPT(6) = 5
9. Summary of failure criteria calculation (only if KEYOPT(2) = 0 or 1). If KEYOPT(6) = 0, only the maximum of all failure criteria (FCMAX) in the element is output. Output of the elastic strains and/or stresses (depending on KEYOPT(5)) for each failure criterion and the maximum of all criteria (FCMAX).
10. Printed only if KEYOPT(2) = 0 or 1; KEYOPT(6) $\neq$ 0; and significant shear stress. If there is no clear maximum location, LN1 and LN2 are set to NL and zero, respectively.
11. Available only at centroid as a ***GET** item.
12. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).
13. If KEYOPT(2) = 2 or 3

Table 4 SOLID46 Miscellaneous Element Output

Description	Names of Items Output	O	R
Member Forces	FX, FY, FZ	*1*	-
Average Transverse Shear Stress	Components and sum	*2*	-
Normal stress along edges	-	*3*	-

1. Output at each node in the element coordinate system if KEYOPT(3) = 2 or 4
2. Output if KEYOPT(6) $\neq$ 0 (calculated from nodal forces)
3. Output at edges I-M, J-N, etc. (calculated from nodal forces). Output only if KEYOPT(2) = 3 and KEYOPT(6) $\neq$ 0

Table 5, "SOLID46 Item and Sequence Numbers" lists the output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 5, "SOLID46 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 3, "SOLID46 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I,J,...,P

Table 5 SOLID46 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	(2*i)-1	(2*NL)+1
ILSYZ	SMISC	(2*i)	(2*NL)+2
ILSUM	NMISC	(2*i)+5	(2*NL)+7
ILANG	NMISC	(2*i)+6	(2*NL)+8

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	(2*NL)+4	(2*NL)+3	(2*NL)+6	(2*NL)+5
P2	SMISC	(2*NL)+7	(2*NL)+8	-	-
P3	SMISC	-	(2*NL)+11	(2*NL)+12	-
P4	SMISC	-	-	(2*NL)+15	(2*NL)+16
P5	SMISC	(2*NL)+20	-	-	(2*NL)+19
P6	SMISC	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	M	N	O	P
P1	SMISC	-	-	-	-
P2	SMISC	(2*NL)+10	(2*NL)+9	-	-
P3	SMISC	-	(2*NL)+14	(2*NL)+13	-
P4	SMISC	-	-	(2*NL)+18	(2*NL)+17
P5	SMISC	(2*NL)+21	-	-	(2*NL)+22
P6	SMISC	(2*NL)+23	(2*NL)+24	(2*NL)+25	(2*NL)+26

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FCMAX (over all layers)	NMISC	1
VALUE	NMISC	2
LN	NMISC	3
ILMAX	NMISC	4
LN1	NMISC	5
LN2	NMISC	6

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
ILSUM (between layers i-1 and i)	NMISC	2i+7
ILANG	NMISC	2i+8
FCMAX (at layer i)	NMISC	(2*(NL+i))+7
VALUE (at layer i)	NMISC	(2*(NL+i))+8
FC	NMISC	(4*NL)+8+15(N-1)+1
VALUE	NMISC	(4*NL)+8+15(N-1)+2
LN	NMISC	(4*NL)+8+15(N-1)+3
EPELFX	NMISC	(4*NL)+8+15(N-1)+4
EPELFY	NMISC	(4*NL)+8+15(N-1)+5
EPELFZ	NMISC	(4*NL)+8+15(N-1)+6
EPELFXY	NMISC	(4*NL)+8+15(N-1)+7
EPELFYZ	NMISC	(4*NL)+8+15(N-1)+8
EPELFXZ	NMISC	(4*NL)+8+15(N-1)+9
SFX	NMISC	(4*NL)+8+15(N-1)+10
SFY	NMISC	(4*NL)+8+15(N-1)+11
SFZ	NMISC	(4*NL)+8+15(N-1)+12
SFXY	NMISC	(4*NL)+8+15(N-1)+13
SFYZ	NMISC	(4*NL)+8+15(N-1)+14
SFXZ	NMISC	(4*NL)+8+15(N-1)+15

Note

The i in *Table 5, "SOLID46 Item and Sequence Numbers"* (where i = 1, 2, 3 ..., NL) refers to the layer number of the element. NL is the maximum layer number as input for real constant NL ($1 \leq NL \leq 250$). N is the failure number as stored on the results file in compressed form, e.g., only those failure criteria requested will be written to the results file. For example, if only the maximum strain and the Tsai-Wu failure criteria are requested, the maximum strain criteria will be stored first (N = 1) and the Tsai-Wu failure criteria will be stored second (N = 2). In addition, if more than one criteria is requested, the maximum value over all criteria is stored last (N = 3 for this example).

SOLID46 Assumptions and Restrictions

- Zero volume elements are not allowed. Usually, this occurs if the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.
- Zero thickness layers are allowed only if a zero thickness is defined at all corners. Tapering down to zero is not allowed. No slippage is assumed between the element layers.
- All material orientations are parallel to the reference plane. Further, any warped layers act as if they are flat and parallel to the reference plane.
- The matrix input option (KEYOPT(2) = 3) assumes a uniform thickness of the element. This thickness is computed based on the nodal locations and on KREF.

- It has been observed that large differences (factors greater than 1000) between different moduli of the same material can cause large differences between the equation solver maximum and minimum pivots, and can even cause "NEGATIVE PIVOT..." messages to appear. If this occurs, you should consider whether the material properties are realistic. Enhanced solution stability for such cases also occurs by suppressing the extra displacement shapes (KEYOPT(1) = 1).
- The element matrices are reformed every iteration unless option 1 of **KUSE** is active.
- Interlaminar shear stresses for SHELL91 and SHELL99 shell elements are based on the premise that there are no interlaminar (transverse) shear stresses at the outer surface of the shell. This assumption cannot be used for a solid element. Thus, SOLID46 has two forms of shear stress calculations:
  - Those based on nodal forces (labeled "average transverse shear stress components").
  - Those based on the strain-displacement relationships, averaged across layers when applicable (labeled "maximum interlaminar shear stress").

  Neither one of these is exact, but ideally they will agree with each other. In both situations, the given values are averages, which will be less than the peak value. The differences between the average and the peak will be small in most cases; however, differences up to a factor of two have been seen.
- Additional elements in the thickness direction will improve the interlaminar shear stress calculation.
- When brick (rectangular prism) elements are used, both calculations result in constant stresses over the volume of the element. In all cases, the values are constant in the plane of the layer and may, therefore, be thought of as centroidal values. Hence, one should consider using solid-to-solid submodeling to get accurate shear stress values at a free edge.
- These shear stresses are discussed further in the *Theory Reference for ANSYS and ANSYS Workbench*. The *Structural Analysis Guide* contains additional information on composite elements.

SOLID46 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element is limited to 20 constant thickness layers, or 10 tapered layers, and does not allow the user-input constitutive matrix option (that is, KEYOPT(2) = 3 is not valid).
- The DAMP material property is not allowed.
- KEYOPT(4) can only be set to 0 (default).
- The six user-defined failure criteria (subroutines USRFC1 through USRFC6) are not allowed.
- The only special feature allowed is stress stiffening.

INFIN47

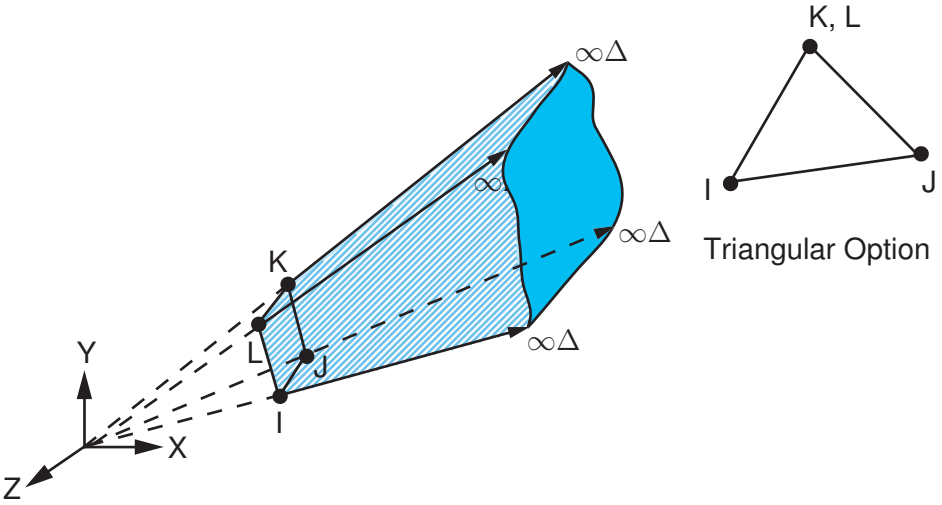
3-D Infinite Boundary

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

INFIN47 Element Description

INFIN47 is used to model an open boundary of a 3-D unbounded field problem. The element may be a 4-node quadrilateral or a 3-node triangle with a magnetic potential or temperature degree of freedom at each node. The enveloped (or enclosed) element types may be the SOLID5, SOLID96, or SOLID98 magnetic elements or the SOLID70, SOLID90 or SOLID87 thermal solid elements. With the magnetic degree of freedom the analysis may be linear or nonlinear static. With the thermal degree of freedom steady-state or transient analyses (linear or nonlinear) may be done. See INFIN47 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 INFIN47 Geometry



INFIN47 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "INFIN47 Geometry"*. The element is defined by 4 nodes, and the material properties. Nonzero material properties must be defined. A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. The element x-axis is parallel to the I-J side of the element.

The coefficient matrix of this boundary element is, in general, unsymmetric. The matrix is made symmetric by averaging the off-diagonal terms to take advantage of a symmetric solution with a slight decrease in accuracy. KEYOPT(2) can be used to keep an unsymmetric matrix from being made symmetric.

A summary of the element input is given in *INFIN47 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

INFIN47 Input Summary

Nodes
I, J, K, L

Degrees of Freedom

MAG if KEYOPT(1) = 0
TEMP if KEYOPT(1) = 1

Real Constants
None

Material Properties

MUZERO if KEYOPT(1) = 0, (has default value for MKS units or can be set with the **EMUNIT** command).
KXX if KEYOPT(1) = 1

Surface Loads
None

Body Loads
None

Element Printout
None

Special Features
None

KEYOPT(1)
Element degree(s) of freedom:

0 --
Magnetic option

1 --
Thermal option

KEYOPT(2)
Coefficient matrix:

0 --
Make the coefficient matrix symmetric

1 --
Coefficient matrix is used as generated (symmetric or unsymmetric, depending on the problem)

INFIN47 Output Data

The boundary element has no output of its own since it is used only to provide a semi-infinite boundary condition to a model consisting of other elements.

INFIN47 Assumptions and Restrictions

- The 4 nodes defining the element should lie as close as possible to a flat plane; however, a moderate out-of-plane tolerance is permitted so that the element may have a somewhat warped shape.
- An excessively warped element will produce a warning message. In the case of warping errors, triangular elements should be used (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- Shell element warping tests are described in detail in the tables of Applicability of Warping Tests and Warping Factor Limits in the *Theory Reference for ANSYS and ANSYS Workbench*.
- Zero area elements are not allowed.

- The semi-infinite volume is assumed to be bound on five sides (four, if triangular) by the boundary element and by four semi-infinite radial surfaces (three, if triangular) defined from the global coordinate system origin through nodes I and J, J and K, K and L, and L and I (nodes I and J, J and K, and K and I if triangular).
- The boundary element should be as normal as possible to the radial surfaces.
- Acute or wide intersection angles should be avoided by "filling-in" the model with the other elements so that the line of boundary elements around the model is smooth and concave when viewed from the global coordinate system origin.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.
- The boundary element must lie "against" an enclosed element (that is, share the same nodes).
- The exterior semi-infinite domain is assumed to be homogeneous, isotropic, and linear without containing any sources or sinks.
- The origin of the global coordinate system must be inside the model and as centrally located as possible.
- The surface of boundary elements should be located away from the region of interest of the enclosed elements for better accuracy. The surface of boundary elements need not totally surround the model.
- The element may not be deactivated with the **EKILL** command.
- When used in a model with higher order elements SOLID90, SOLID87, and SOLID98, the midside nodes of these elements must be removed at the interface with INFIN47 [**EMID**].
- If KEYOPT(2) = 1, the matrices are presumed to be unsymmetric.

INFIN47 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic field capability.
- The MAG degree of freedom is not active.
- KEYOPT(1) defaults to 1 (TEMP) instead of 0 and cannot be changed.
- The material property MUZERO is not allowed.

ANSYS Emag

- This element has only magnetic field capability, and does not have thermal capability.
- The only active degree of freedom is MAG.
- The only allowable material property is MUZERO.
- KEYOPT(1) can only be set to 0 (default).

MATRIX50

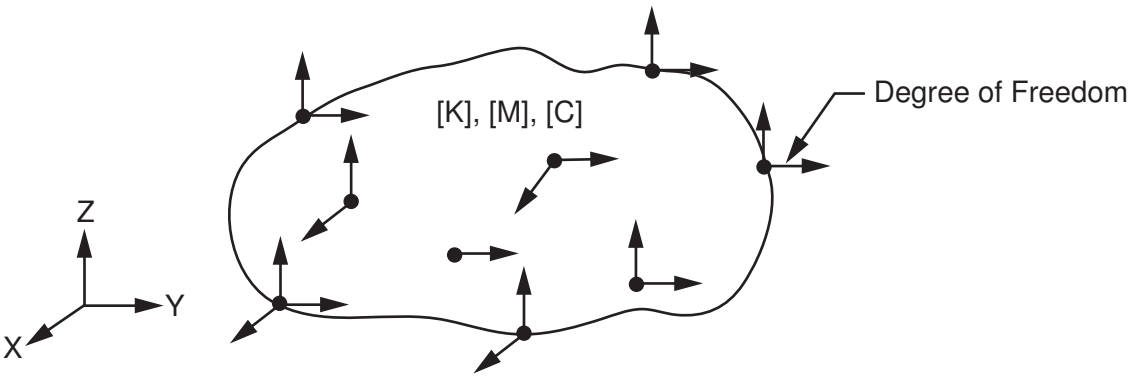
Superelement (or Substructure)

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MATRIX50 Element Description

MATRIX50 is a group of previously assembled ANSYS elements that is treated as a single element. The superelement, once generated, may be included in any ANSYS model and used in any analysis type for which it is applicable. The superelement can greatly decrease the cost of many analyses. Once the superelement matrices have been formed, they are stored in a file and can be used in other analyses the same way any other ANSYS elements are used. Multiple load vectors may also be stored with the superelement matrices, thereby allowing various loading options. See MATRIX50 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 MATRIX50 Schematic



MATRIX50 Input Data

The superelement, which is a mathematical matrix representation of an arbitrary structure, has no fixed geometrical identity and is conceptually shown in *Figure 1, "MATRIX50 Schematic"*. Any analysis using a superelement as one of its element types is called a superelement use pass (or run). The degrees of freedom are the master degrees of freedom specified during the generation pass.

The element name is MATRIX50 (the number 50 or the name MATRIX50 should be input for the variable *ENAME* on the **ET** command). The **SE** command is used to define a superelement. **SE** reads the superelement from `Jobname.SUB` (defaults to `File.SUB`) in the working directory. The material number [**MAT**] is only used when material dependent damping [**MP**,DAMP] or electrical permittivity [**MP**,PERX] is an input. The real constant table number [**REAL**] is not used. However, the appropriate element type number [**TYPE**] must be entered.

An element load vector is generated along with the element at each load step of the superelement generation pass. Up to 31 load vectors may be generated. Load vectors may be proportionately scaled in the use pass. The scale factor is input on the element surface load command [**SFE**]. The load label is input as SELV, the load key is the load vector number, *KVAL* determines whether the load vector is real or imaginary, and the load value is the scale factor. The load vector number is determined from the load step number associated with the superelement generation. If a superelement load vector has a zero scale factor (or is not scaled at all), this load vector is not included in the analysis. Any number of load vector-scale factor combinations may be used in the use pass.

In a large rotation analysis (**NLGEOM**,ON), you can use KEYOPT(3) to specify whether the load vectors associated with this element type rotates with the element (as you would for a pressure load) or remains in the original (unrotated) direction (as you would for a non-follower force load); all load vectors (if multiple load vectors) are

rotated or left unrotated. You can use KEYOPT(4) to indicate that the superelement was generated with constraints (**D**) so that it cannot translate or rotate freely in the use pass as expected (although you can apply constraints in the use pass to the master degrees of freedom to prevent such motion.)

The KEYOPT(1) option is for the special case where the superelement is to be used with a T^4 nonlinearity, such as for radiation. The `File.` SUB for this case may be constructed directly by the user or may be generated by AUX12, the radiation matrix generator.

A summary of the element input is given in *MATRIX50 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

MATRIX50 Input Summary

Nodes
None input (supplied by element)

Degrees of Freedom
As determined from the included element types (a mixture of multi-field degrees of freedom is not allowed)

Real Constants
None

Material Properties
DAMP, PERX

Surface Loads
Surface load effects may be applied through a generated load vector and scale factors. Use the **SFE** command to supply scale factors with *LAB* = SELV, *LKEY* = load vector number (31 maximum), *KVAL* = real or imaginary, and *VAL1* = scale factor.

Body Loads
Body loads may be applied through a generated load vector and scale factors as described for surface loads.

Special Features
Radiation (if KEYOPT(1) = 1), Large rotation

KEYOPT(1)
Element behavior:

0 --
Normal substructure

1 --
Special radiation substructure

KEYOPT(3)
Load vector update with large rotations (**NLGEOM**,ON):

0 --
Load vector(s) rotate with the substructure as it rotates

1 --
Load vector(s) do not rotate and remain in their original direction

KEYOPT(4)
Constrained substructure with large rotations (**NLGEOM**,ON):

0 --
Substructure was unconstrained in the generation pass

1 --
Substructure was constrained in the generation pass

KEYOPT(6)
Nodal force output:

0 --
Do not print nodal forces

1 --
Print nodal forces

MATRIX50 Output Data

Displacements and forces may be printed for each (master) degree of freedom in a structural superelement in the "use" pass. The nodal forces may be output if KEYOPT(6) = 1. The stress distribution within the superelement and the expanded nodal displacements can be obtained from a subsequent stress pass. In addition to the database and substructure files from the generation run, `File.DSUB` must be saved from the superelement "use" pass and input to the expansion pass (if an expansion pass is desired). A general description of solution output is given in *Section 2.2: Solution Output*.

MATRIX50 Assumptions and Restrictions

- A superelement may contain elements of any type except Lagrange multiplier-based elements (such as MPC184, PLANE182 with KEYOPT(6) = 1, and CONTA171 with KEYOPT(2) = 3).
- See the **D** command for degree of freedom field groups.
- Superelements of different field types may be mixed within the use run.
- The nonlinear portion of any element included in a superelement will be ignored and any bilinear element will maintain its initial status throughout the analysis.
- Superelements may contain other superelements.
- The PCG solver does not support MATRIX50 elements.
- The relative locations of the superelement attachment points in the nonsuperelement portion of the model (if any) should match the initial superelement geometry.
- If the superelement contains a mass matrix, acceleration [**ACEL**] defined in the use run will be applied to the superelement.
- If a load vector containing acceleration effects is also applied in the use run, *both* accelerations (the **ACEL** command and the load vector) will be applied to the superelement.
- Similarly, if the superelement contains a damping matrix (as specified in the generation run) and α and β damping multipliers [**ALPHA** and **BETA**] are defined in the use run, additional damping effects will be applied to the superelement.
- You should be careful to avoid duplicating acceleration and damping effects.
- Pressure and thermal effects may be included in a superelement only through its load vectors.
- The dimensionality of the superelement corresponds to the maximum dimensionality of any element used in its generation. A 2-D superelement should only be used in 2-D analyses, and 3-D superelements in 3-D analyses.
- See the *Theory Reference for ANSYS and ANSYS Workbench* for a discussion of the substructure matrix assembly procedure.

MATRIX50 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(1) = 0
- The PERX material property is not applicable.

ANSYS Professional

- This element may be used as a radiation substructure only. KEYOPT(1) defaults to 1 instead of 0 and cannot be changed.
- The DAMP material property, PERX material property, surface loads, and body loads are not applicable.
- The large rotation special feature is not applicable.

ANSYS Emag

- This element may be used as a Trefftz substructure only.
- The DAMP material property is not applicable.
- The large rotation special feature is not applicable.

CONTAC52

3-D Point-to-Point Contact

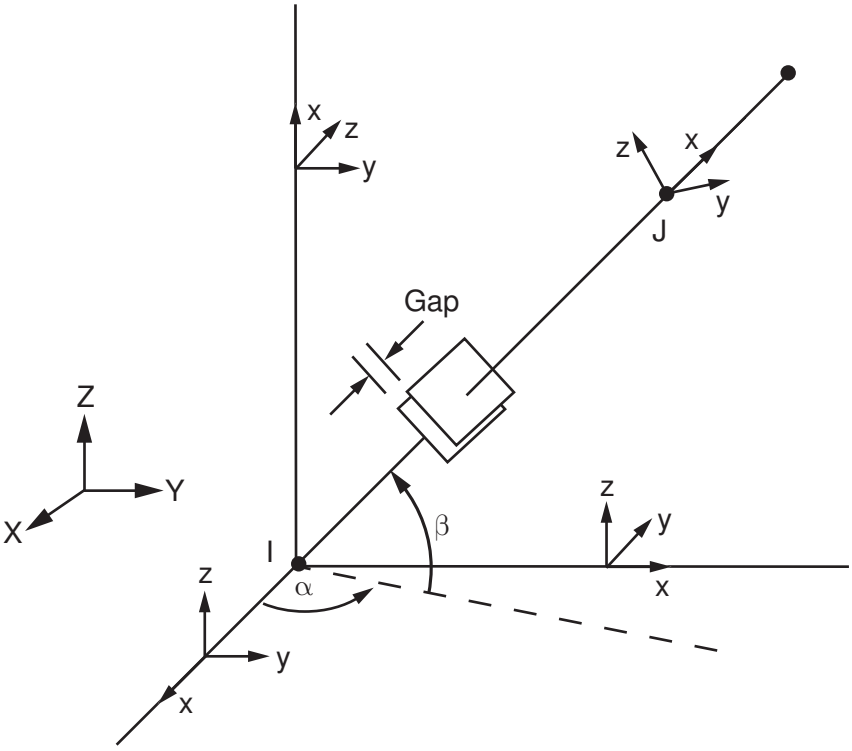
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

CONTAC52 Element Description

CONTAC52 represents two surfaces which may maintain or break physical contact and may slide relative to each other. The element is capable of supporting only compression in the direction normal to the surfaces and shear (Coulomb friction) in the tangential direction. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element may be initially preloaded in the normal direction or it may be given a gap specification. A specified stiffness acts in the normal and tangential directions when the gap is closed and not sliding. See CONTAC52 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other contact elements, such as CONTAC12 and COMBIN40, are also available.

Figure 1 CONTAC52 Geometry



CONTAC52 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "CONTAC52 Geometry"*. The element is defined by two nodes, two stiffnesses (KN and KS), an initial gap or interference (GAP), and an initial element status (START). The orientation of the interface is defined by the node locations, or by a user-specified gap direction. The interface is assumed to be perpendicular to the I-J line or to the specified gap direction. The element coordinate system has its origin at node I and the x-axis is directed toward node J or in the user-specified gap direction. The interface is parallel to the element y-z plane.

The normal stiffness, KN, should be based upon the stiffness of the surfaces in contact. See Nonlinear Structural Analysis in the *Structural Analysis Guide* for guidelines on choosing a value for KN. In some cases (such as initial

interference analyses, nonconvergence, or over penetration), it may be useful to change the KN value between load steps or in a restart in order to obtain an accurate, converged solution. The sticking stiffness, KS, represents the stiffness in the tangential direction when elastic Coulomb friction is selected ($\mu > 0.0$ and KEYOPT(1) = 0). The coefficient of friction μ is input as material property MU and is evaluated at the average of the two node temperatures. Stiffnesses may also be computed from the maximum expected force divided by the maximum allowable surface displacement. KS defaults to KN.

The initial gap defines the gap size (if positive) or the displacement interference (if negative). This input is the opposite of that used for CONTAC12. If you do not specify the gap direction (by means of real constants NX, NY, and NZ), an interference causes the nodes to separate. The gap size may be input as a real constant (GAP) or automatically calculated from the input node locations (as the distance between node I and node J) if KEYOPT(4) = 1. Interference must be input as a real constant. Stiffness is associated with a zero or negative gap. The initial element status (START) is used to define the "previous" condition of the interface to be used at the start of the first substep. This input is used to override the condition implied by the interference specification and is useful in anticipating the final interface configuration and in reducing the number of iterations required for convergence.

You can specify the gap direction by means of real constants NX, NY, and NZ (the global Cartesian X, Y, and Z components of the gap direction vector). If you do not specify the gap direction, the program will calculate the direction based on the initial positions of the I and J nodes, such that a positive normal displacement (in the element coordinate system) of node J relative to node I tends to open the gap. You should always specify the gap direction if nodes I and J have the same initial coordinates, if the model has an initial interference condition in which the underlying elements' geometry overlaps, or if the initial open gap distance is very small. If the gap is initially geometrically open, the correct normal (NX, NY, NZ) usually points from node I toward node J.

The only material property used is the interface coefficient of friction μ. A zero value should be used for frictionless surfaces. Temperatures may be specified at the element nodes (for material property evaluation only). The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I).

The force deflection relationships for the interface element can be separated into the normal and tangential (sliding) directions as shown in *Figure 2, "CONTAC52 Force-Deflection Relationship"*. The element condition at the beginning of the first substep is determined from the START parameter. If the interface is closed and sticking, KN is used in the gap resistance and KS is used for sticking resistance. If the interface is closed but sliding, KN is used in the gap resistance and the constant friction force μFN is used for the sliding resistance.

In the normal direction, when the normal force (FN) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted.

KEYOPT(3) can be used to specify a "weak spring" across an open interface, which is useful for preventing rigid body motion that could occur in a static analysis. The weak spring stiffness is computed by multiplying the normal stiffness KN by a reduction factor. The default reduction factor of 1E-6 can be overridden with real constant REDFACT.

This "weak spring" capability is not analogous to overlaying an actual spring element (such as COMBIN14) with a low stiffness value. The REDFACT capability will not limit gap separation when a tensile force is applied.

In the tangential direction, for $FN < 0$ and the absolute value of the tangential force (FS) less than $\mu|FN|$, the interface sticks and responds as a linear spring. For $FN < 0$ and $FS = \mu|FN|$, sliding occurs. If contact is broken, FS = 0.

If KEYOPT(1) = 1, rigid Coulomb friction is selected, KS is not used, and the elastic sticking capability is removed. This option is useful for displacement controlled problems or for certain dynamic problems where sliding dominates.

For analyses involving friction, using **NROPT**,UNSYM is useful (and, in fact, sometimes required) for problems where the normal and tangential (sliding) motions are strongly coupled, such as in a wedge insertion problem.

A summary of the element input is given in *CONTAC52 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTAC52 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

KN, GAP, START, KS, REDFACT, NX,
NY, NZ

See *Table 1, "CONTAC52 Real Constants"* for details on these real constants.

Material Properties
DAMP, MU

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Special Features

Nonlinear
Adaptive descent

KEYOPT(1)
Sticking stiffness if MU > 0.0:

0 --
Elastic Coulomb friction (KS used for sticking stiffness)

1 --
Rigid Coulomb friction (resisting force only)

KEYOPT(3)
Weak spring across open gap:

0 --
No weak spring across an open gap

1 --
Use a weak spring across an open gap

KEYOPT(4)
Basis for gap size:

0 --
Gap size based on gap real constant

1 --
Gap size determined from initial node locations (ignore gap real constant)

KEYOPT(7)

Element-level time incrementation control. Note that this option should be activated first at the procedure level if **SOLCONTROL** is ON. **SOLCONTROL**,ON,ON is the most frequent usage with this element. If **SOLCONTROL**,ON,OFF, this keyoption is not activated.

0 --

Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

1 --

Change in contact predictions made to maintain a reasonable time/load increment (recommended)

Table 1 CONTAC52 Real Constants

No.	Name	Description
1	KN	Normal stiffness
2	GAP	Initial gap size; a negative value assumes an initial interference condition.
3	START	Initial condition: If = 0.0 or blank, initial status of element is determined from gap input If = 1.0, gap is initially closed and not sliding (if MU $\neq$ 0.0), or sliding (if MU = 0.0) If = 2.0, gap is initially closed and sliding If = 3.0, gap initially open
4	KS	Sticking stiffness
5	REDFACT	Default reduction factor 1E-6
6	NX	Defined gap normal - X component
7	NY	Defined gap normal - Y component
8	NZ	Defined gap normal - Z component

CONTAC52 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTAC52 Element Output Definitions"*.

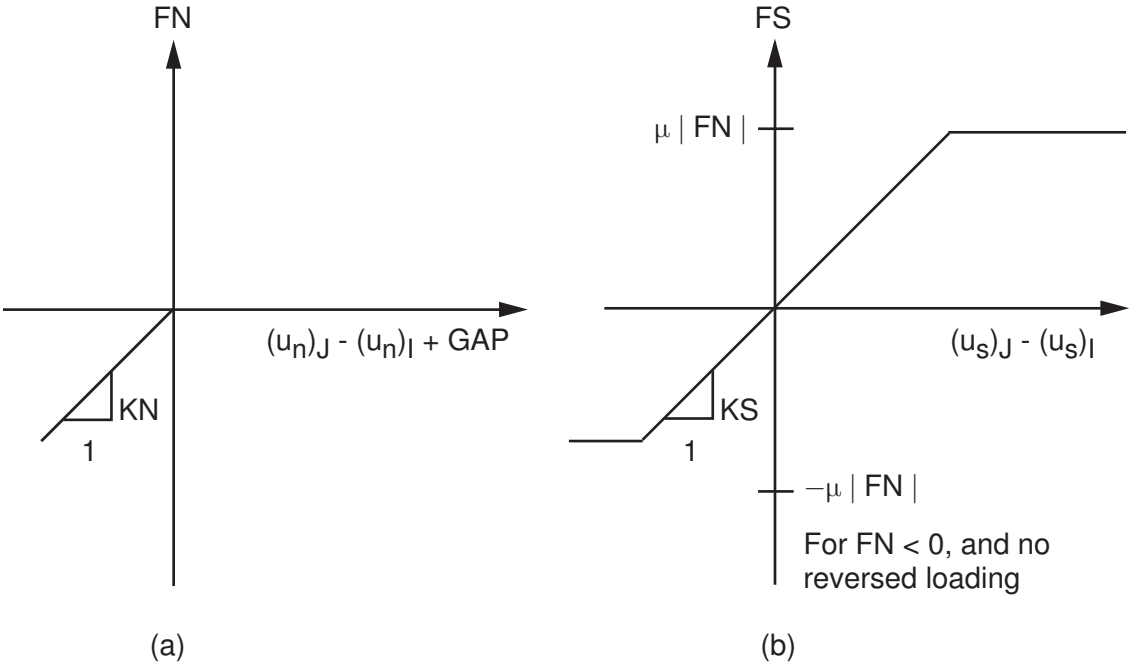
Force-deflection curves are illustrated in *Figure 2, "CONTAC52 Force-Deflection Relationship"*.

The value of USEP is determined from the normal displacement (u_n) (in the element x-direction) between the interface nodes at the end of a substep, that is: USEP = $(u_n)_J$ - $(u_n)_I$ + GAP. This value is used in determining the normal force, FN. The values represented by UT(Y, Z) are the total translational displacements in the element y and z directions. The maximum value printed for the sliding force, FS, is μ|FN|. Sliding may occur in both the element y and z directions. STAT describes the status of the element at the end of a substep. If STAT = 1, the gap is closed and no sliding occurs. If STAT = 3, the gap is open. A value of STAT = 2 indicates the node J slides relative to node I. For a frictionless surface (μ = 0.0), the converged element status is either STAT = 2 or 3.

The element coordinate system orientation angles α and β (shown in *Figure 1, "CONTAC52 Geometry"*) are computed by the program from the node locations. These values are printed as ALPHA and BETA respectively. α ranges from 0° to 360° and β from -90° to +90°. Elements lying along the Z-axis are assigned values of α = 0°, β = ± 90°, respectively. Elements lying off the Z-axis have their coordinate system oriented as shown for the general α, β position. Note, for α = 90°, $\beta \rightarrow$ 90°, the element coordinate system flips 90° about the Z-axis. The value

of ANGLE represents the principal angle of the friction force in the element y-z plane. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 CONTAC52 Force-Deflection Relationship



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTAC52 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	T(I), T(J)	Y	Y
USEP	Gap size	Y	Y
FN	Normal force (along I-J line)	Y	Y
STAT	Element status	*1*	*1*
ALPHA, BETA	Element orientation angles	Y	Y
MU	Coefficient of friction	*2*	*2*
UT(Y, Z)	Displacement (node J - node I) in element y and z directions	*2*	*2*
FS	Tangential (friction) force (vector sum)	*2*	*2*
ANGLE	Principal angle of friction force in element y-z plane	*2*	*2*

1. If the value of STAT is:

1 - Contact, no sliding

2 - Sliding contact

3 - Gap open

2. If MU > 0.0
3. Available only at centroid as a ***GET** item.

Table 3, "CONTAC52 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "CONTAC52 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTAC52 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 CONTAC52 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FN	SMISC	1
FS	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
USEP	NMISC	3
ALPHA	NMISC	4
BETA	NMISC	5
UTY	NMISC	6
UTZ	NMISC	7
MU	NMISC	8
ANGLE	NMISC	9

CONTAC52 Assumptions and Restrictions

- The element operates bilinearly only in the static and the nonlinear transient dynamic analyses. If used in other analysis types, the element maintains its initial status throughout the analysis.
- The element is nonlinear and requires an iterative solution. Nonconverged substeps are not in equilibrium.
- Unless the gap direction is specified (NX, NY, NZ), nodes I and J may not be coincident since the nodal locations define the interface orientation. The element maintains is original orientation in either a small or a large deflection analysis.
- The element coordinate system is defined by the initial I and J node locations or by the specified gap direction.
- The gap value may be specified independent of the node locations.

- The element may have rotated nodal coordinates since a displacement transformation into the element coordinate system is included.
- The element stiffness KN should not be exactly zero, and unreasonably high stiffness values also should be avoided. The rate of convergence decreases as the stiffness increases.
- Although it is permissible to change KN, it is not permissible to change any other real constants between load steps. Therefore, if you plan to change KN, you cannot allow the value of KS to be defined by default, because the program would then attempt to redefine KS as KN changed. You must explicitly define KS whenever KN changes, to maintain a consistent value throughout all load steps.
- The element may not be deactivated with the **EKILL** command.
- If μ is not equal to zero, the element is nonconservative as well as nonlinear. Nonconservative elements require that the load be applied very gradually, along the actual load history path, and in the proper sequence (if multiple loadings exist).

CONTAC52 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element is frictionless. MU is not allowed as a material property and KS is not allowed as a real constant.
- Temperature body loads are not applicable in a structural analysis.
- KEYOPT(1) is not applicable.
- The DAMP material property is not allowed.

PLANE53

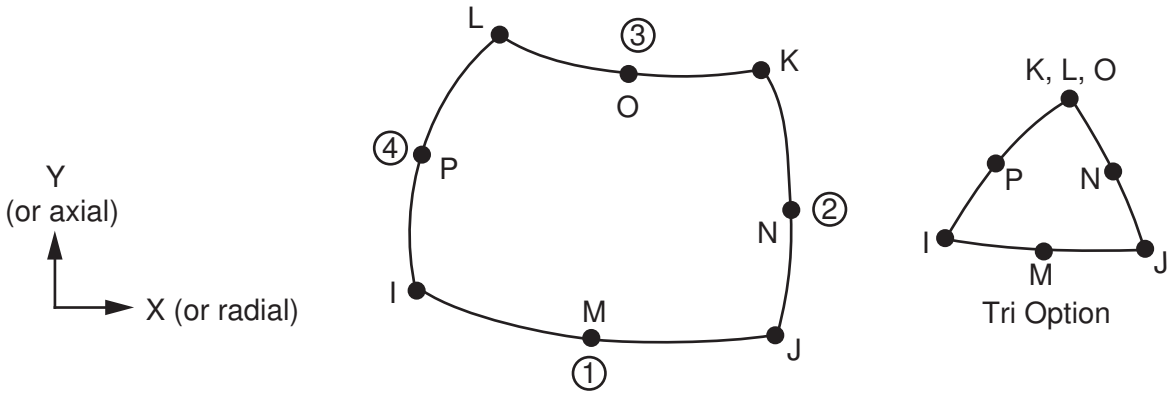
2-D 8-Node Magnetic Solid

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

PLANE53 Element Description

PLANE53 models 2-D (planar and axisymmetric) magnetic fields. The element is defined by 8 nodes and has up to 4 degrees of freedom per node: z component of the magnetic vector potential (AZ), time-integrated electric scalar potential (VOLT), electric current (CURR), and electromotive force (EMF). PLANE53 is based on the magnetic vector potential formulation and is applicable to the following low-frequency magnetic field analyses: magnetostatics, eddy currents (AC time harmonic and transient analyses), voltage forced magnetic fields (static, AC time harmonic and transient analyses), and electromagnetic-circuit coupled fields (static, AC time harmonic and transient analyses). The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. See PLANE53 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A similar 4 node element (without voltage forced and magnetic-circuit coupled capability) is PLANE13.

Figure 1 PLANE53 Geometry



PLANE53 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE53 Geometry"*. The element input data includes 8 nodes and the magnetic material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4 \pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is specified through the MURX and MURY material property labels.

MGXX and MGYY represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX and MGYY. Permanent magnet polarization and orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*. Nonlinear magnetic B-H properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element, depending upon the KEYOPT(1) value. Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (VOLT or AZ) and *VALUE* corresponds to the value (time-integrated electric scalar potential or vector magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (AMPS or CSGZ) and *VALUE* corresponds to the value (current or magnetic current segment). The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in *Section 2.8: Node and Element Loads*. Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "PLANE53 Geometry"* using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Lorentz and Maxwell forces may be made available for a subsequent structural analysis with companion elements [**LDREAD**].

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. Source current density and voltage body loads may be applied to an area [**BFA**] or input as an element value [**BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

Air elements in which local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

A summary of the element input is given in *PLANE53 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE53 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

AZ if KEYOPT(1) = 0
VOLT, AZ if KEYOPT(1) = 1 AZ
CURR if KEYOPT(1) = 2
AZ, CURR, EMF if KEYOPT(1) = 3 or 4

Real Constants

CARE, TURN, LENG, DIRZ, FILL, VELOX,
VELOY, OMEGAZ, XLOC, YLOC

See *Table 1, "PLANE53 Real Constants"* for descriptions of the real constants.

Material Properties

MUZERO, MURX, MURY, RSVX, MGXX, MGYY, plus BH data table (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Maxwell Force flag --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Magnetic Virtual Displacement --

VD(I), VD(J), VD(K), VD(L), VD(M), VD(N),
VD(O), VD(P) Source Current Density, if KEYOPT(1) = 0 or 1:
spare, spare, JSZ(I), PHASE(I), spare, spare,
JSZ(J), PHASE(J), spare, spare, JSZ(K), PHASE(K),
spare, spare, JSZ(L), PHASE(L) spare, spare,
JSZ(M), PHASE(M), spare, spare, JSZ(N), PHASE(N),
spare, spare, JSZ(O), PHASE(O), spare, spare,
JSZ(P), PHASE(P)
Voltage Loading, if KEYOPT(1) = 2:
VLTG(I), PHASE(I), VLTG(J), PHASE(J), VLTG(K), PHASE(K),
VLTG(L), PHASE(L), VLTG(M), PHASE(M), VLTG(N), PHASE(N),
VLTG(O), PHASE(O), VLTG(P), PHASE(P)

Special Features

Birth and death
Adaptive descent

KEYOPT(1)
Element degrees of freedom:

0 --
AZ degree of freedom: static domain, induced eddy current domain

1 --
VOLT, AZ degrees of freedom: current-fed massive conductor

2 --
AZ, CURR degrees of freedom: voltage-fed stranded coil

3 --
AZ, CURR, EMF degrees of freedom: circuit-coupled stranded coil

4 --
AZ, CURR, EMF degrees of freedom: circuit-coupled massive conductor

KEYOPT(2)
Element conventional velocity:

0 --
Velocity effects ignored

1 --
Conventional velocity formulation (not available if KEYOPT(1) = 2, 3, or 4)

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(4)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

KEYOPT(7)
Store magnetic forces for coupling with elements:

0 --
Midside node (higher-order) structural elements

1 --
Non-midside node structural elements

Table 1 PLANE53 Real Constants

No.	Name	Description
KEYOPT(1) ≥ 2 - voltage forced or electromagnetic-circuit coupled analyses (coils or massive conductors)		
1	CARE	Coil cross-sectional area; required when KEYOPT(1) = 2, 3, 4
2	TURN	Total number of coil turns (stranded coil only), default is 1; KEYOPT(1) = 2, 3
3	LENG	Coil length in Z-direction, (required for planar models only), default is 1 meter; KEYOPT(1) = 2, 3, 4
4	DIRZ	1 for current in positive z-axis, -1 for current in negative z-axis; KEYOPT(1) = 2, 3, 4
5	FILL	Coil fill factor; KEYOPT(1) = 2, 3
KEYOPT(2) = 1 (and KEYOPT(1) = 0 or 1) - Velocity effects of a conducting body		
6	VELOX	Velocity component in X-direction (global Cartesian)
7	VELOY	Velocity component in Y-direction (global Cartesian)
8	OMEGAZ	Angular (rotational) velocity (Hz, cycles/sec) about the Z-axis (global Cartesian), at the pivot point
9	XLOC	Pivot point X-location (global Cartesian coordinate)
10	YLOC	Pivot point Y-location (global Cartesian coordinate)

PLANE53 Output Data

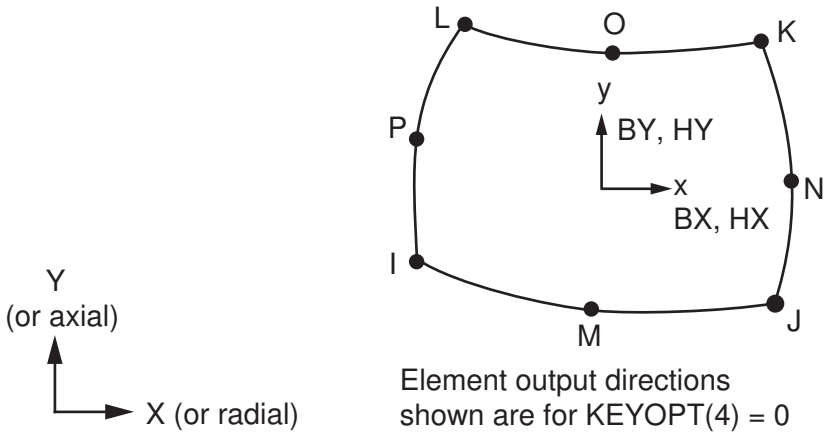
The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution

- Additional element output as shown in *Table 2, "PLANE53 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE53 Magnetic Element Output"*. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE53 Magnetic Element Output



Because of different sign conventions for Cartesian and polar coordinate systems, magnetic flux density vectors point in opposite directions for planar (KEYOPT(3) = 0) and axisymmetric (KEYOPT(3) = 1) analyses.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 PLANE53 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
CENT:X, Y	Global location XC, YC	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	*1*	-
MUX, MUY	Magnetic secant permeability	*1*	*1*
H:X, Y	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y	Magnetic flux density components (X, Y)	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
JSZ	Source current density, valid for static analysis only	*1*	*1*

Name	Definition	O	R
JTZ	Total current density	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
FJB(X, Y)	Lorentz force components	*1*	*1*
FMX(X, Y)	Maxwell force components	*1*	*1*
FVW(X, Y, SUM)	Virtual work force components	*1*	*1*
FMAG:X, Y	Combined (FJB or FMX) force components	-	*1*
ERES	Element resistance value (for stranded coils only)	-	*1*
EIND	Element inductance value (for stranded coils only)	-	*1*
DMUXX, DMUYY	Differential permeability	*1*	*1*
V:X, Y, SUM	Velocity components	*1*	*1*
MRE	Magnetic Reynolds number	*1*	*1*
TJB(Z)	Lorentz torque about global Cartesian +Z-axis	*1*	*1*
TMX(Z)	Maxwell torque about global Cartesian +Z-axis	*1*	*1*
TVW(Z)	Virtual work torque about global Cartesian +Z-axis	*1*	*1*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively.

For harmonic analysis, joule losses (JHEAT), forces (FJB(X, Y), FMX(X, Y), FVW(X, Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in both the "Real" and "Imaginary" data sets. The macros **POWERH**, **FMAGSUM**, and **TORQSUM** can be used to retrieve this data.

Inductance values (EIND) obtained for KEYOPT(1) = 2, 3, or 4 are only valid under the following conditions: the problem is linear (constant permeability), there are no permanent magnets in the model, and only a single coil exists in the model.

2. Available only at centroid as a ***GET** item.

Table 3 PLANE53 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Solution	H, HSUM, B, BSUM, FJB, FMX, V, VSUM	*1*	-

1. Output at each node, if KEYOPT(5) = 2

Table 4, "PLANE53 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 4, "PLANE53 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "PLANE53 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 4 PLANE53 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JSZ	SMISC	1
MUX	NMISC	1
MUY	NMISC	2
FVWX	NMISC	3
FVWY	NMISC	4
FVWSUM	NMISC	5
JTZ	NMISC	7
ERES	NMISC	8
EIND	NMISC	9
DMUXX	NMISC	10
DMUYY	NMISC	11
VX	NMISC	12
VY	NMISC	13
MRE	NMISC	15
TJB(X Y)	NMISC	16
TMX(X, Y)	NMISC	17
TVW(X, Y)	NMISC	18

PLANE53 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE53 Geometry"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- Current density loading (**BFE**,,JS) is only valid for the AZ option (KEYOPT(1) = 0). For the VOLT, AZ option (KEYOPT(1) = 1) use **F**,,AMPS.
- When this element does not have the VOLT degree of freedom (KEYOPT(1) = 0), for a harmonic or transient analysis, its behavior depends on the applied load. For a **BFE**,,JS load, the element acts as a stranded conductor. Without **BFE**,,JS loads, it acts as a solid conductor modeling eddy current effects.
  - In this respect, PLANE53 (and PLANE13) are not like the 3-D elements SOLID97 and SOLID117. When SOLID97 and SOLID117 do not have the VOLT degree of freedom, they act as stranded conductors.

- Permanent magnets are not permitted in a harmonic analysis.
- For magnetostatic analyses, the VOLT, AZ option is not allowed.
- For harmonic and transient (time-varying) analyses, the ANSYS product does not support the analysis of coupled velocity and circuit effects.
- Reduced transient methods cannot be used. A 2-D planar or axisymmetric skin-effect analysis (where eddy current formation is permitted in conducting regions with impressed current loading) is performed by setting KEYOPT(1) = 1, specifying a resistivity, and coupling all VOLT degrees of freedom for elements in each of such regions.
- For voltage forced magnetic field (KEYOPT(1) = 2) and circuit coupled problems (KEYOPT(1) = 3,4), note the following additional restrictions:
  - Only MKS units are allowed.
  - The permeability and conductivity are isotropic and constant.
  - All CURR degrees of freedom in a coil region must be coupled (**CP** command).
  - All EMF degrees of freedom in a coil region must be coupled (**CP** command).
- For circuit coupled transient analyses, use THETA = 1.0, the default value, on the **TINTP** command to specify the backward Euler method. For more information, refer to the *Theory Reference for ANSYS and ANSYS Workbench*, as well as the description of the **TINTP** command in the *Commands Reference*.

For velocity effects (KEYOPT(2) = 1), note the following restrictions:

- Velocity effects are valid only for AZ or AZ-VOLT DOF options.
- Isotropic resistivity.
- Solution accuracy may degrade if the element magnetic Reynolds number is much greater than 1.0. (See the discussion of magnetic field analysis in the *Low-Frequency Electromagnetic Analysis Guide*.)
- If KEYOPT(1) $\geq$ 2 or KEYOPT(2) $\geq$ 1, unsymmetric matrices are produced.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

PLANE53 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

BEAM54

2-D Elastic Tapered Unsymmetric Beam

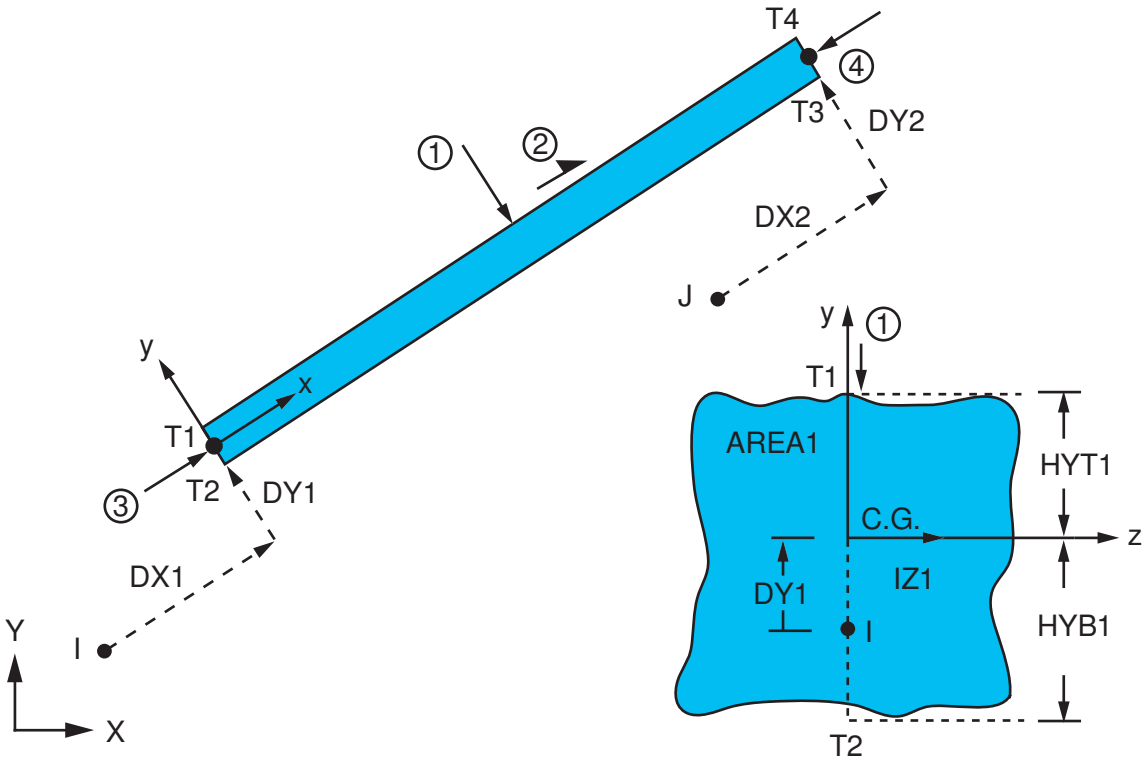
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

BEAM54 Element Description

BEAM54 is a uniaxial element with tension, compression, and bending capabilities. The element has three degrees of freedom at each node: translations in the nodal x and y directions and rotation about the nodal z-axis. This element allows a different unsymmetrical geometry at each end and permits the end nodes to be offset from the centroidal axis of the beam. If these features are not desired, use the uniform symmetrical beam element, BEAM3. This element does not have plastic, creep, or swelling capabilities. These effects are included in BEAM23, the 2-D, untapered, plastic beam element. Stress stiffening capability is also included. See BEAM44 for a 3-D tapered unsymmetrical beam.

Shear deformation and elastic foundation effects are available as options. Another option is available for printing the forces acting on the element in the element coordinate directions. See BEAM54 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 BEAM54 Geometry



Note - use subscript 2 with node J

BEAM54 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "BEAM54 Geometry"*. The element x-axis is oriented in the global X-Y plane (or a parallel plane) from node I (end 1) toward node J (end 2). The element real constants describe the beam in terms of the cross-sectional areas, the area moments of inertia, the extreme fiber distances from the centroid, the offset distances, and the shear deflection constant. The moments of inertia (IZ_) are about the principal axis of the beam. The element may be used in

axisymmetric analyses if hoop effects are negligible, such as for bolts, slotted cylinders, etc. The areas and moments of inertia must be input on a full 360° basis for an axisymmetric analysis.

The shear deflection constant (SHEARZ) is optional. A zero value of SHEARZ may be used to neglect shear deflection. The shear modulus (GXY) is used only with shear deflection. See *Section 2.14: Shear Deflection* for details. The offset constants (DX_, DY_) define the centroid location of the section relative to the node location. Offset distances are measured positive from the node in the positive element coordinate directions. The shear areas (AREAS_) are used only for the shear stress computation. The shear areas are generally less than the actual cross-sectional area.

The AREA_, IZ_, HY__, and AREAS_ real constants for end 2 of the beam default to the corresponding end 1 values if zero. Furthermore, the "top" height at end 1, HYT1, defaults to the "bottom" height at end 1, HYB1, and the "top" height at end 2, HYT2, defaults to the "top" height at end 1, HYT1. The heights are measured from the centroid of the section. The elastic foundation stiffness (EFS) is defined as the pressure required to produce a unit normal deflection of the foundation. This capability is bypassed if EFS equals zero. The initial strain in the element (ISTRN) is given by Δ/L, where Δ is the difference between the element length, L, (as defined by the I and J node locations) and the zero strain length. The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration. An added mass per unit length may be input with the ADDMAS value.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM54 Geometry"*. The circled number represents the load key for the indicated face. Positive pressures act into the element. Lateral pressures are input as a force per unit length. End "pressures" are input as a force. KEYOPT(10) allows tapered lateral pressures to be offset from the nodes. Temperatures may be input as element body loads at the four "corner" locations shown in *Figure 1, "BEAM54 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(9) is used to request output at intermediate locations. It is based on equilibrium (free body of a portion of the element) considerations and is not valid if:

- Stress stiffening is turned on [**SSTIF**,ON], or
- More than one component of angular velocity is applied [**OMEGA**], or
- *Any* angular velocities or accelerations are applied with the **CGOMGA**, **DOMEGA**, or **DCGOMG** commands.

A summary of the element input is given in *BEAM54 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

BEAM54 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, ROTZ

Real Constants

AREA1, IZ1, HYT1, HYB1, AREA2, IZ2,
HYT2, HYB2, DX1, DY1, DX2, DY2,
SHEARZ, AREAS1, AREAS2, EFS, ISTRN, ADDMAS
See *Table 1, "BEAM54 Real Constants"* for descriptions of the real constants.

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J) (-Y normal direction),
face 2 (I-J) (+X tangential direction),
face 3 (I) (+X axial direction),
face 4 (J) (-X axial direction) (use negative value for loading in opposite direction)

Body Loads

Temperatures --
T1, T2, T3, T4

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(6)
Member force and member moment output:

0 --
No member force printout

1 --
Print member forces and moments in the element coordinate system

KEYOPT(9)
Additional output at points between ends I and J:

N --
Output at N intermediate locations (N = 0, 1, 3, 5, 7, or 9)

KEYOPT(10)
Load location, used in conjunction with the offset values input on the **SFBEAM** command):

0 --
Offset is in terms of length units

1 --
Offset is in terms of a length ratio (0.0 to 1.0)

Note

If SHEARZ = 0.0, there is no shear deflection in the element y direction. AREAS1 and AREAS2 are used only for the shear stress calculation.

In the following table, the I-end of the beam corresponds to end 1, and the J-end of the beam corresponds to end 2.

Table 1 BEAM54 Real Constants

No.	Name	Description
1	AREA1	Cross-sectional area at I end of beam
2	IZ1	Moment of inertia about Z at I end of beam

No.	Name	Description
3	HYT1	Distance from CG to top of Y surface at I end of beam
4	HYB1	Distance from CG to bottom of Y surface at I end of beam
5	AREA2	Cross-sectional area at J end of beam
6	IZ2	Moment of inertia about Z at J end of beam
7	HYT2	Distance from CG to top of Y surface at J end of beam
8	HYB2	Distance from CG to bottom of Y surface at J end of beam
9	DX1	X offset at CG at I end of beam
10	DY1	Y offset at CG at I end of beam
11	DX2	X offset at CG at J end of beam
12	DY2	Y offset at CG at J end of beam
13	SHEARZ	Shear deflection constant
14	AREAS1	Shear area at I end of beam
15	AREAS2	Shear area at J end of beam
16	EFS	Elastic foundation stiffness
17	ISTRN	Initial strain in element
18	ADDMAS	Added mass/unit length

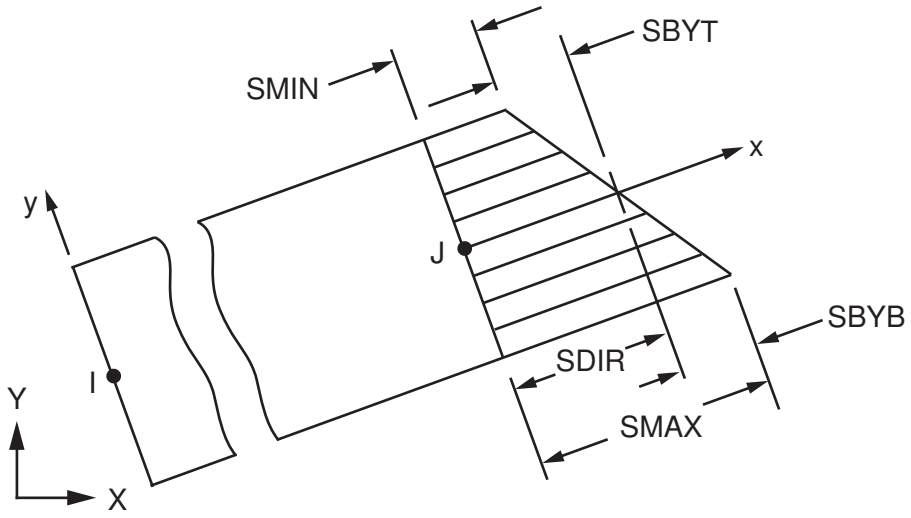
BEAM54 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "BEAM54 Element Output Definitions"*

Several items are illustrated in *Figure 2, "BEAM54 Stress Output"*. At each cross-section, the computed output consists of the direct (axial) stress and two bending components. Then these three values are combined to evaluate the maximum and minimum stresses. If KEYOPT(6) = 1 for this element, the 6 member forces and moments (3 at each end) are also printed (in the element coordinate system). The element x-axis is defined through the center of gravity of the cross-section. Additional results at intermediate locations between the ends may be output with KEYOPT(9). A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 BEAM54 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 BEAM54 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*4*
TEMP	Temperatures T1, T2, T3, T4	Y	Y
PRES	Pressures P1 at nodes I,J; OFFST1 at I,J; P2 at I,J; OFFST2 at I,J; P3 at I; P4 at J	Y	Y
SDIR	Axial direct stress	*1*	*1*
SBYT	Bending stress on the element +Y side of the beam	*1*	*1*
SBYB	Bending stress on the element -Y side of the beam	*1*	*1*
SMAX	Maximum stress (direct stress + bending stress)	*1*	*1*
SMIN	Minimum stress (direct stress - bending stress)	*1*	*1*
EPELDIR	Axial elastic strain at the end	*1*	*1*
EPELBYT	Bending elastic strain on the element +Y side of the beam	*1*	*1*
EPELBYB	Bending elastic strain on the element -Y side of the beam	*1*	*1*
EPTHDIR	Axial thermal strain at the end	*1*	*1*
EPTHBYT	Bending thermal strain on the element +Y side of the beam	*1*	*1*
EPTHBYB	Bending thermal strain on the element -Y side of the beam	*1*	*1*
EPINAXL	Initial axial strain in the element	*1*	*1*
SXY	Average shear (Y-direction)	*2*	*2*
MFOR(X, Y)	Member forces in the element coordinate system	*3*	Y
MMOMZ	Member moment in the element coordinate system	*3*	Y

1. The item repeats for end I, intermediate locations (see KEYOPT(9)), and end J
2. Output only if real constants AREAS1 and AREAS2 are input
3. If KEYOPT(6) = 1
4. Available only at centroid as a ***GET** item.

Table 3, "BEAM54 Item and Sequence Numbers (KEYOPT(9) = 0)" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "BEAM54 Item and Sequence Numbers (KEYOPT(9) = 0)"*:

Name
output quantity as defined in the *Table 2, "BEAM54 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

IL*n*
sequence number for data at Intermediate Location *n*

Table 3 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 0)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDIR	LS	-	1	4
SBYT	LS	-	2	5
SBYB	LS	-	3	6
EPELDIR	LEPEL	-	1	4
EPELBYT	LEPEL	-	2	5
EPELBYB	LEPEL	-	3	6
EPTHDIR	LEPTH	-	1	4
EPTHBYT	LEPTH	-	2	5
EPTHBYB	LEPTH	-	3	6
EPINAXL	LEPTH	7	-	-
MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MMOMZ	SMISC	-	6	12
SXY	SMISC	-	13	14
P1	SMISC	-	15	16
OFFST1	SMISC	-	17	18
P2	SMISC	-	19	20
OFFST2	SMISC	-	21	22
P3	SMISC	-	23	-
P4	SMISC	-	-	24
SMAX	NMISC	-	1	3
SMIN	NMISC	-	2	4

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 4 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 1)

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	IL1	J
SDIR	LS	-	1	4	7

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	IL1	J
SBYT	LS	-	2	5	8
SBYB	LS	-	3	6	9
EPELDIR	LEPEL	-	1	4	7
EPELBYT	LEPEL	-	2	5	8
EPELBYB	LEPEL	-	3	6	9
EPTHDIR	LEPTH	-	1	4	7
EPTHBYT	LEPTH	-	2	5	8
EPTHBYB	LEPTH	-	3	6	9
EPINAXL	LEPTH	10	-	-	-
MFORX	SMISC	-	1	7	13
MFORY	SMISC	-	2	8	14
MMOMZ	SMISC	-	6	12	18
SXY	SMISC	-	19	20	21
P1	SMISC	-	22	-	23
OFFST1	SMISC	-	24	-	25
P2	SMISC	-	26	-	27
OFFST2	SMISC	-	28	-	29
P3	SMISC	-	30	-	-
P4	SMISC	-	-	-	31
SMAX	NMISC	-	1	3	5
SMIN	NMISC	-	2	4	6

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 5 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 3)

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
SDIR	LS	-	1	4	7	10	13
SBYT	LS	-	2	5	8	11	14
SBYB	LS	-	3	6	9	12	15
EPELDIR	LEPEL	-	1	4	7	10	13
EPELBYT	LEPEL	-	2	5	8	11	14
EPELBYB	LEPEL	-	3	6	9	12	15
EPTHDIR	LEPTH	-	1	4	7	10	13
EPTHBYT	LEPTH	-	2	5	8	11	14
EPTHBYB	LEPTH	-	3	6	9	12	15
EPINAXL	LEPTH	16	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25
MFORY	SMISC	-	2	8	14	20	26

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	IL1	IL2	IL3	J
MMOMZ	SMISC	-	6	12	18	24	30
SXY	SMISC	-	31	32	33	34	35
P1	SMISC	-	36	-	-	-	37
OFFST1	SMISC	-	38	-	-	-	39
P2	SMISC	-	40	-	-	-	41
OFFST2	SMISC	-	42	-	-	-	43
P3	SMISC	-	44	-	-	-	-
P4	SMISC	-	-	-	-	-	45
SMAX	NMISC	-	1	3	5	7	9
SMIN	NMISC	-	2	4	6	8	10

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 6 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 5)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	I	IL1	IL2	IL3	IL4	IL5	J
SDIR	LS	-	1	4	7	10	13	16	19
SBYT	LS	-	2	5	8	11	14	17	20
SBYB	LS	-	3	6	9	12	15	18	21
EPELDIR	LEPEL	-	1	4	7	10	13	16	19
EPELBYT	LEPEL	-	2	5	8	11	14	17	20
EPELBYB	LEPEL	-	3	6	9	12	15	18	21
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21
EPINAXL	LEPTH	22	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37
MFORY	SMISC	-	2	8	14	20	26	32	38
MMOMZ	SMISC	-	6	12	18	24	30	36	42
SXY	SMISC	-	43	44	45	46	47	48	49
P1	SMISC	-	50	-	-	-	-	-	51
OFFST1	SMISC	-	52	-	-	-	-	-	53
P2	SMISC	-	54	-	-	-	-	-	55
OFFST2	SMISC	-	56	-	-	-	-	-	57
P3	SMISC	-	58	-	-	-	-	-	-
P4	SMISC	-	-	-	-	-	-	-	59
SMAX	NMISC	-	1	3	5	7	9	11	13
SMIN	NMISC	-	2	4	6	8	10	12	14

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 7 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 7)

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
SDIR	LS	-	1	4	7	10	13	16	19	22	25
SBYT	LS	-	2	5	8	11	14	17	20	23	26
SBYB	LS	-	3	6	9	12	15	18	21	24	27
EPELDIR	LEPEL	-	1	4	7	10	13	16	19	22	25
EPELBYT	LEPEL	-	2	5	8	11	14	17	20	23	26
EPELBYB	LEPEL	-	3	6	9	12	15	18	21	24	27
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19	22	25
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20	23	26
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21	24	27
EPINAXL	LEPTH	28	-	-	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54
SXY	SMISC	-	55	56	57	58	59	60	61	62	63
P1	SMISC	-	64	-	-	-	-	-	-	-	65
OFFST1	SMISC	-	66	-	-	-	-	-	-	-	67
P2	SMISC	-	68	-	-	-	-	-	-	-	69
OFFST2	SMISC	-	70	-	-	-	-	-	-	-	71
P3	SMISC	-	72	-	-	-	-	-	-	-	-
P4	SMISC	-	-	-	-	-	-	-	-	-	73
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 8 BEAM54 Item and Sequence Numbers (KEYOPT(9) = 9)

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SDIR	LS	-	1	4	7	10	13	16	19	22	25	28	31
SBYT	LS	-	2	5	8	11	14	17	20	23	26	29	32
SBYB	LS	-	3	6	9	12	15	18	21	24	27	30	33
EPELDIR	LEPEL	-	1	4	7	10	13	16	19	22	25	28	31
EPELBYT	LEPEL	-	2	5	8	11	14	17	20	23	26	29	32
EPELBYB	LEPEL	-	3	6	9	12	15	18	21	24	27	30	33

Output Quantity Name	ETABLE and ESOL Command Input												
	Item	E	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
EPTHDIR	LEPTH	-	1	4	7	10	13	16	19	22	25	28	31
EPTHBYT	LEPTH	-	2	5	8	11	14	17	20	23	26	29	32
EPTHBYB	LEPTH	-	3	6	9	12	15	18	21	24	27	30	33
EPINAXL	LEPTH	34	-	-	-	-	-	-	-	-	-	-	-
MFORX	SMISC	-	1	7	13	19	25	31	37	43	49	55	61
MFORY	SMISC	-	2	8	14	20	26	32	38	44	50	56	62
MMOMZ	SMISC	-	6	12	18	24	30	36	42	48	54	60	66
SXY	SMISC	-	67	68	69	70	71	72	73	74	75	76	77
P1	SMISC	-	78	-	-	-	-	-	-	-	-	-	79
OFFST1	SMISC	-	80	-	-	-	-	-	-	-	-	-	81
P2	SMISC	-	82	-	-	-	-	-	-	-	-	-	83
OFFST2	SMISC	-	84	-	-	-	-	-	-	-	-	-	85
P3	SMISC	-	86	-	-	-	-	-	-	-	-	-	-
P4	SMISC	-	-	-	-	-	-	-	-	-	-	-	87
SMAX	NMISC	-	1	3	5	7	9	11	13	15	17	19	21
SMIN	NMISC	-	2	4	6	8	10	12	14	16	18	20	22

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

BEAM54 Assumptions and Restrictions

- The beam must not have a zero length, area, or moment of inertia. The beam must lie in an X-Y plane.
- The element heights are used in locating the extreme fibers for the stress calculations and in computing the thermal gradient. Incorrect bending or thermal stresses may result if zero heights are input.
- Tapers within an element, if any, should be gradual. If AREA2/AREA1 or I_2/I_1 is not between 0.5 and 2.0, a warning message is output. If the ratio is outside of the range of 0.1 to 10.0, an error message is output. The element should not taper to a point at either end (zero thickness).
- The applied thermal gradient is assumed to be linear across the thickness and along the length of the element.
- The flexible length of the beam is adjusted to account for the effect of the offsets. The offset lengths may be regarded as rigid portions of the beam. Unequal lateral offsets, which rotate the beam, also cause a corresponding shortening of the beam's flexible length. The difference between the lateral offsets should not exceed the length of the element.
- The effect of offsets on the mass matrix is ignored if the lumped mass matrix formulation is specified [**LUMPM**,ON].
- Rotational body forces resulting from an angular velocity are based upon the node locations (as if zero offsets).
- The shear stress is calculated based on the shear force rather than the shear deflection.

BEAM54 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special features allowed are stress stiffening and large deflection.

PLANE55

2-D Thermal Solid

MP ME <> PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

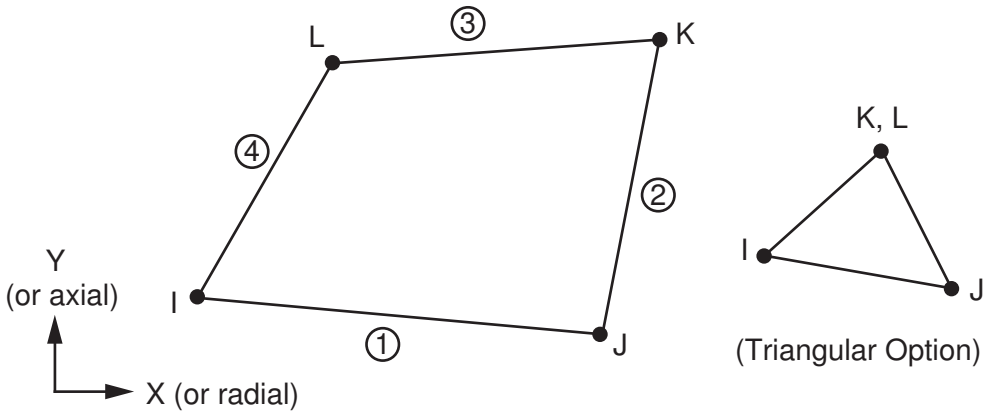
PLANE55 Element Description

PLANE55 can be used as a plane element or as an axisymmetric ring element with a 2-D thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node.

The element is applicable to a 2-D, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field. If the model containing the temperature element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE42). A similar element with midside node capability is PLANE77. A similar axisymmetric element which accepts nonaxisymmetric loading is PLANE75.

An option exists that allows the element to model nonlinear steady-state fluid flow through a porous medium. With this option the thermal parameters are interpreted as analogous fluid flow parameters. See PLANE55 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE55 Geometry



PLANE55 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE55 Geometry"*. The element is defined by four nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "PLANE55 Geometry"*.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

A mass transport option is available with KEYOPT(8). With this option the velocities VX and VY must be input as real constants (in the element coordinate system). Also, temperatures should be specified along the entire inlet

boundary to assure a stable solution. With mass transport, you should use specific heat (C) and density (DENS) material properties instead of enthalpy (ENTH).

The nonlinear porous flow option is selected with KEYOPT(9) = 1. For this option, temperature is interpreted as pressure and the absolute permeabilities of the medium are input as material properties KXX and KYY. Properties DENS and VISC are used for the mass density and viscosity of the fluid. See the *Theory Reference for ANSYS and ANSYS Workbench* for a description of the properties C and MU, which are used in calculating the coefficients of permeability, with reference to the Z terms ignored. Temperature boundary conditions input with the **D** command are interpreted as pressure boundary conditions, and heat flow boundary conditions input with the **F** command are interpreted as mass flow rate (mass/time).

This element can also have a Z-depth specified by KEYOPT(3) and real constant THK. Be careful when using this option with other physics, especially radiation. Radiation view factors will be based on a unit Z-depth (only).

A summary of the element input is given in *PLANE55 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE55 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP

Real Constants
THK, VX, VY

THK = Thickness (used only if KEYOPT(3) = 3)

VX = Mass transport velocity in X (used only if KEYOPT(8) > 0)

VY = Mass transport velocity in Y (used only if KEYOPT(8) > 0)

Material Properties
KXX, KYY, DENS, C, ENTH, VISC, MU (VISC and MU used only if KEYOPT (9) = 1. Do not use ENTH with KEYOPT(8) = 1 or 2).

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Special Features
Birth and death

KEYOPT(1)
How to evaluate film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS - TB|

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

3 --
Plane with Z-depth, specified via real constant THK.

KEYOPT(4)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side.

KEYOPT(8)
Mass transport effects:

0 --
No mass transport effects

1 --
Mass transport with VX and VY

2 --
Same as 1 but also print mass transport heat flow

KEYOPT(9)
Nonlinear fluid flow option:

0 --
Standard heat transfer element

1 --
Nonlinear steady-state fluid flow analogy element (temperature degree of freedom interpreted as pressure)

PLANE55 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE55 Element Output Definitions"*

For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. If KEYOPT(9) = 1, the standard thermal output should be interpreted as the analogous fluid flow output. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* and of postprocessing data in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE55 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*4*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	-
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	*1*
HFILM	Film coefficient at each node of face	*1*	-
TBULK	Bulk temperature at each node of face	*1*	-
TAVG	Average face temperature	*1*	*1*
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFLUX	Heat flux at each node of face	*1*	-
HEAT FLOW BY MASS TRANSPORT	Heat flow rate across face by mass transport	*2*	-
PRESSURE GRAD	Total pressure gradient and its X and Y components	*3*	-
MASS FLUX	Mass flow rate per unit cross-sectional area	*3*	-
FLUID VELOCITY	Total fluid velocity and its X and Y components	*3*	-

1. If a surface load is input
2. If KEYOPT(8) = 2
3. If KEYOPT(9) = 1
4. Available only at centroid as a ***GET** item.

Table 2, "PLANE55 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2:*

The Item and Sequence Number Table of this manual for more information. The following notation is used in *Table 2, "PLANE55 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE55 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 PLANE55 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE55 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- The element must lie in an X-Y plane as shown in *Figure 1, "PLANE55 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal element is to be replaced by a PLANE42 structural element with surface stresses requested, the thermal element should be oriented with face IJ or face KL as a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- If KEYOPT(8) > 0, unsymmetric matrices are produced.
- When mass flow is activated (KEYOPT(8)=1 or 2), the element Peclet number should be less than 1:

  $Pe = \rho*v*L*Cp/(2*k) < 1.0$

  Where L is an element length scale based on the flow direction and element geometry. See PLANE55 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

PLANE55 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element does not have the mass transport or fluid flow options. KEYOPT(8) and KEYOPT(9) can only be set to 0 (default).
- The VX and VY real constants are not applicable.
- The VISC and MU material properties are not applicable.
- The element does not have the birth and death feature.

SHELL57

Thermal Shell

MP ME <> PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

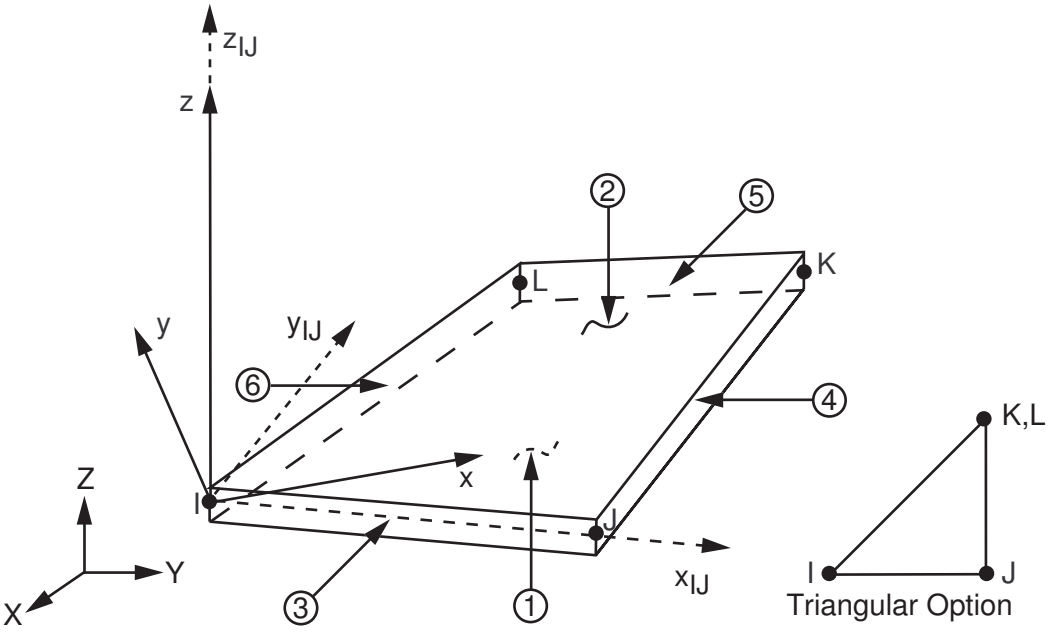
SHELL57 Element Description

SHELL57 is a 3-D element having in-plane thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis. See SHELL57 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

If the model containing the conducting shell element is to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SHELL63). If both in-plane and transverse conduction are needed, SHELL131 with KEYOPT(3) = 0 or 1 should be used.

SHELL57 is essentially the same as SHELL131 with KEYOPT(3) = 2.

Figure 1 SHELL57 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL57 Input Data

The geometry, node locations, and coordinate systems for this element are shown in *Figure 1, "SHELL57 Geometry"*. The element is defined by four nodes, four thicknesses, a material direction angle, and the material properties.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*. The element x-axis may be rotated by an angle THETA (in degrees).

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be specified as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SHELL57 Geometry"*. Edge convection and flux loads are input on a per unit length basis.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

A summary of the element input is given in *SHELL57 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL57 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J; defaults to TK(I)
TK(K) - Shell thickness at node K; defaults to TK(I)
TK(L) - Shell thickness at node L; defaults to TK(I)
THETA - Element X-axis rotation

Material Properties
KXX, KYY, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (I-J-K-L) (bottom, -Z side), face 2 (I-J-K-L) (top, +Z side),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Special Features
Birth and death

KEYOPT(2)
Where to evaluate film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS - TB|

SHELL57 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL57 Element Output Definitions"*

Heat flowing out of the element is considered to be positive. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL57 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Convection face area	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	*1*
HFILM	Film coefficient	*1*	*1*
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HEAT FLUX	Heat flux at each node of face	*1*	-

1. If a surface load is input

2. Available only at centroid as a ***GET** item.

Table 2, "SHELL57 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL57 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL57 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

Table 2 SHELL57 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FACE 1 (BOT)	FACE 2 (TOP)	FACE 3 (J-I)	FACE 4 (K-J)	FACE 5 (L-K)	FACE 6 (I-L)
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SHELL57 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the elements are not numbered properly.
- The element must not taper down to a zero thickness at any corner.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.

SHELL57 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

PIPE59

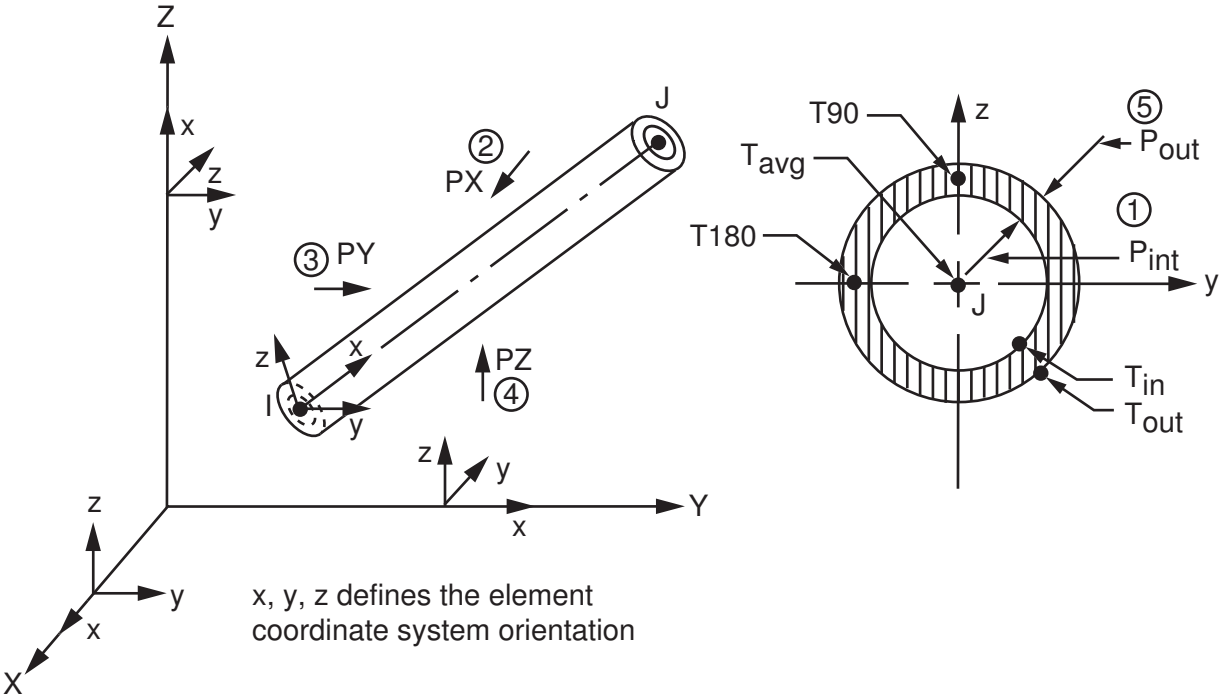
Immersed Pipe or Cable

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

PIPE59 Element Description

PIPE59 is a uniaxial element with tension-compression, torsion, and bending capabilities, and with member forces simulating ocean waves and current. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The element is similar to PIPE16 except that the element loads include the hydrodynamic and buoyant effects of the water and the element mass includes the added mass of the water and the pipe internals. A cable representation option (similar to LINK8) is also available with the element. The element has stress stiffening and large deflection capabilities. See PIPE59 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PIPE59 Geometry

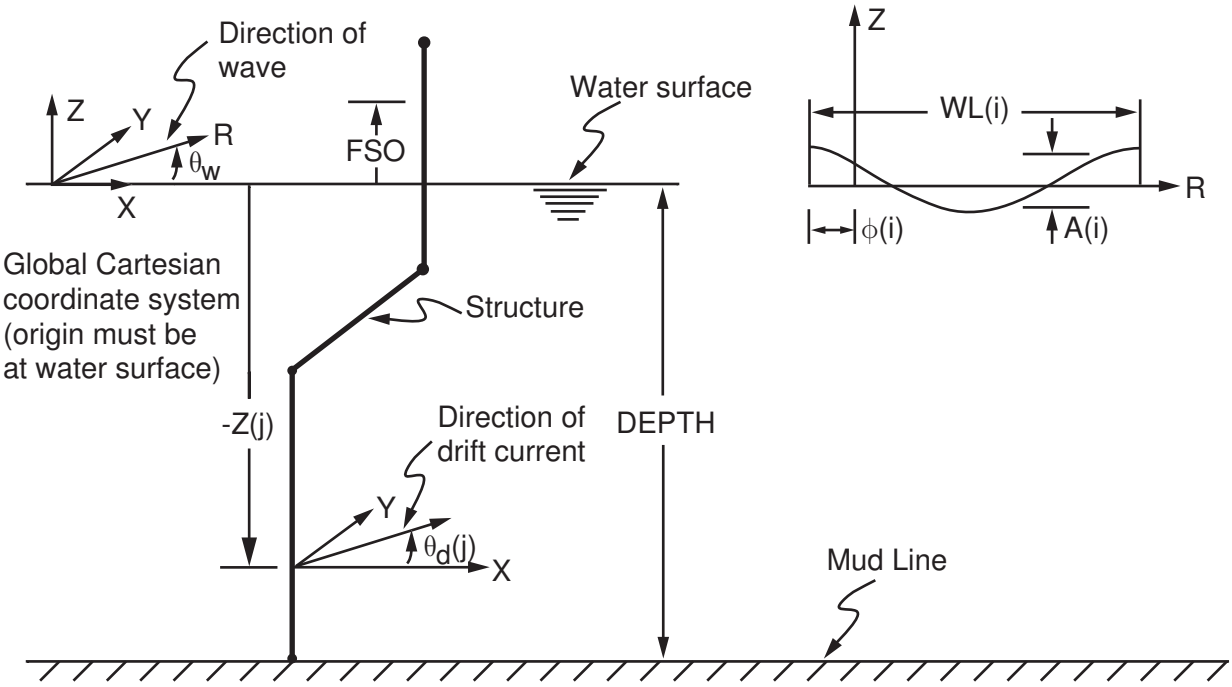


PIPE59 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE59 Geometry"*. The element input data (see *PIPE59 Input Summary*) includes two nodes, the pipe outer diameter and wall thickness, certain loading and inertial information (described in *Table 1, "PIPE59 Real Constants"* and *Figure 2, "PIPE59 Geometry"*), and the isotropic material properties. An external "insulation" may be defined to represent ice loads or biofouling. The material VISC is used only to determine Reynolds number of the fluid outside the pipe.

The element x-axis is oriented from node I toward node J. The element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in *Figure 1, "PIPE59 Geometry"*. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). Input and output locations around the pipe circumference identified as being at 0° are located along the element y-axis, and similarly 90° is along the element z-axis.

Figure 2 PIPE59 Geometry



KEYOPT(1) may be used to convert the element to the cable option by deleting the bending stiffnesses. If the element is not "torque balanced", the twist-tension option may be used (KEYOPT(1) = 2). This option accounts for the twisting induced when a helically wound or armored structure is stretched. The KEYOPT(2) key allows a reduced mass matrix and load vector formulation (with rotational degrees of freedom terms deleted as described in the *Theory Reference for ANSYS and ANSYS Workbench*). This formulation is useful for suppressing large deflections and improving bending stresses in long, slender members. It is also often used with the twist-tension pipe option for cable structures.

The description of the waves, the current, and the water density are input through the water motion table. The water motion table is associated with a material number and is explained in detail in *Table 2, "PIPE59 Water Motion Table"*. If the water motion table is not input, no water is assumed to surround the pipe. Note that even though the word "water" is used to describe various input quantities, the quantities may actually be characteristic of any fluid. Alternate drag coefficient and temperature data may also be input through this table.

A summary of the element input is given in *PIPE59 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PIPE59 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) ≠ 1, or
UX, UY, UZ if KEYOPT(1) = 1

Real Constants

DO, TWALL, CD, CM, DENSO, FSO,
CENMPL, CI, CB, CT, ISTR, DENSIN,
TKIN, TWISTTEN

See *Table 1, "PIPE59 Real Constants"* for details.

Material Properties

EX, ALPX (or CTEX *or* THSX), PRXY (or NUXY), DENS, GXY, DAMP, VISC

Surface Loads

Pressures --

1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --

TOUT(I), TIN(I), TOUT(J), TIN(J) if KEYOPT(3) = 0
TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J) if KEYOPT(3) = 1

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(1)

Element behavior:

0 --

Pipe option

1 --

Cable option

2 --

Pipe with twist-tension option

KEYOPT(2)

Load vector and mass matrix:

0 --

Consistent mass matrix and load vector

1 --

Reduced mass matrix and load vector

KEYOPT(3)

Temperatures represent:

0 --

The through-wall gradient

1 --

The diametral gradient

KEYOPT(5)

Wave force modifications:

0 --

Waves act on elements at their actual location

1 --

Elements are assumed to be at wave peak

2 --

Upward vertical wave velocity acts on element

3 --
Downward vertical wave velocity acts on element

4 --
Elements are assumed to be at wave trough

KEYOPT(6)
Member force and moment output:

0 --
No printout of member forces or moments

2 --
Print member forces and moments in the element coordinate system

KEYOPT(7)
Extra element output:

0 --
Basic element printout

1 --
Additional hydrodynamic integration point printout

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

KEYOPT(9)
PX, PY, and PZ transverse pressures:

0 --
Use only the normal component of pressure

1 --
Use the full pressure (normal and shear components)

Table 1 PIPE59 Real Constants

No.	Name	Description
1	DO	Outside diameter (D_o)
2	TWALL	Wall thickness of the pipe (defaults to D_o/2.0)
3	CD	Normal drag coefficient (C_D). May be overridden by Constants 43 through 54 of water motion table (see *Table 2, "PIPE59 Water Motion Table"*)
4	CM	Coefficient of inertia (C_M)
5	DENSO	Internal fluid density (used for pressure effect only) (Mass/Length^3)
6	FSO	Z coordinate location of the free surface of the fluid on the inside of the pipe (used for pressure effect only)
7	CENMPL	Mass per unit length of the internal fluid and additional hardware (used for mass matrix computation)
8	CI	Added-mass-used/added-mass for circular cross section (if blank or 0, defaults to 1; if CI should be 0.0, enter negative number)

No.	Name	Description
9	CB	Buoyancy force ratio (Buoyancy-force based on outside diameter and water density) (if blank or 0, defaults to 1; if CB should be 0.0, enter negative number)
10	CT	Coefficient of tangential drag (C_T). May be overridden by Constants 55 through 66 of water motion table (See *Table 2, "PIPE59 Water Motion Table"*).
11	ISTR	Initial strain in axial direction.
12	DENSIN	Density of external insulation[*1*].
13	TKIN	Thickness of external insulation (t_i).
14	TWISTTEN	Twist tension constant (used if KEYOPT(1) = 2) (See *Theory Reference for ANSYS and ANSYS Workbench* for more details).

1. Density of external insulation (ρ_i).

PIPE59 Water Motion Information

The data listed in *Table 2, "PIPE59 Water Motion Table"* is entered in the data table with the **TB** commands. If the table is not input, no water is assumed to surround the pipe. Constants not input are assumed to be zero. If the table is input, ACELZ must also have a positive value and remain constant for all load steps. The constant table is started by using the **TB** command (with *Lab* = WATER). Up to 196 constants may be defined with the **TBDATA** commands. The constants (C1-C196) entered on the **TBDATA** commands (6 per command) are:

where:

KWAVE = Wave selection key (see next section)
KCRC = Wave/current interaction key (see next section)
DEPTH = Depth of water to mud line (DEPTH > 0.0) (Length)
DENSW = Water density, ρ_w, (DENSW > 0.0) (Mass/Length^3)
θ_w = Wave direction (see *Figure 2, "PIPE59 Geometry"*)
Z(j) = Z coordinate of location j of drift current measurement (see *Figure 2, "PIPE59 Geometry"*) (location must be input starting at the ocean floor (Z(1) = -DEPTH) and ending at the water surface (Z(MAX) = 0.0). If the current does not change with height, only W(1) needs to be defined.)
W(j) = Velocity of drift current at location j (Length/Time)
θ_d(j) = Direction of drift current at location j (Degrees) (see *Figure 2, "PIPE59 Geometry"*)
Re(k) = Twelve Reynolds number values (if used, all 12 must be input in ascending order)
CD(k) = Twelve corresponding normal drag coefficients (if used, all 12 must be input)
CT(k) = Twelve corresponding tangential drag coefficients (if used, all 12 must be input)
T(j) = Temperature at Z(j) water depth (Degrees)
A(i) = Wave peak-to-trough height (0.0 $\leq$ A(i) < DEPTH) (Length) (if KWAVE = 2, A(1) is entire wave height and A(2) through A(5) are not used)
τ(i) = Wave period (τ(i) > 0.0) (Time/Cycle)
ϕ(i) = Adjustment for phase shift (Degrees)
WL(i) = Wave length (0.0 $\leq$ WL(i) < 1000.0*DEPTH) (Length)

(default $WL(i) = \frac{ACELZ(\tau(i))^2}{2\pi}\tanh\frac{2\pi DEPTH}{WL(i)}$)

Use 0.0 with Stokes theory (KWAVE = 2).

Table 2 PIPE59 Water Motion Table

Constant	Meaning					
1-5	KWAVE	KCRC	DEPTH	DENSW	θ_w	
7-12	Z(1)	W(1)	θ_d(1)	Z(2)	W(2)	θ_d(2)
13-18	Z(3)	W(3)	θ_d(3)	Z(4)	W(4)	θ_d(4)
19-24	Z(5)	W(5)	θ_d(5)	Z(6)	W(6)	θ_d(6)
25-30	Z(7)	W(7)	θ_d(7)	Z(8)	W(8)	θ_d(8)
31-36	Re(1)	Re(2)	Re(3)	Re(4)	Re(5)	Re(6)
37-42	Re(7)	Re(8)	Re(9)	Re(10)	Re(11)	Re(12)
43-48	CD(1)	CD(2)	CD(3)	CD(4)	CD(5)	CD(6)
49-54	CD(7)	CD(8)	CD(9)	CD(10)	CD(11)	CD(12)
55-60	CT(1)	CT(2)	CT(3)	CT(4)	CT(5)	CT(6)
61-66	CT(7)	CT(8)	CT(9)	CT(10)	CT(11)	CT(12)
67-72	T(1)	T(2)	T(3)	T(4)	T(5)	T(6)
73-74	T(7)	T(8)				
79-82	A(1)	τ(1)	ϕ(1)	WL(1)	For KWAVE = 0, 1, or 2	
85-88	A(2)	τ(2)	ϕ(2)	WL(2)	For KWAVE = 2, use only A(1), τ(1), ϕ(1)	
etc.	etc.					
193-196	A(20)	τ(20)	ϕ(20)	WL(20)		
79-81	X(1)/(H*T*G)	Not Used	ϕ(1)	For KWAVE = 3 (See *7.* for definitions other than ϕ(1))		
85-86	X(2)/(H*T*G)	DPT/LO				
91-92	X(3)/(H*T*G)	L/LO				
97-98	X(4)/(H*T*G)	H/DPT				
103-104	X(5)/(H*T*G)	Ψ/(G*H*T)				
109	X(6)/(H*T*G)					
etc.	etc.					
193	X(20)/(H*T*G)					

The distributed load applied to the pipe by the hydrodynamic effects is computed from a generalized Morison's equation. This equation includes the coefficient of normal drag (C_D) (perpendicular to the element axis) and the coefficient of tangential drag (C_T), both of which are a functions of Reynolds numbers (Re). These values are input as shown in *Table 1, "PIPE59 Real Constants"* and *Table 2, "PIPE59 Water Motion Table"*.

The Reynolds numbers are determined from the normal and tangential relative particle velocities, the pipe geometry, the water density, and the viscosity μ (input as VISC). The relative particle velocities include the effects of water motion due to waves and current, as well as motion of the pipe itself. If both Re(1) and CD(1) are positive, the value of C_D from the real constant table (*Table 1, "PIPE59 Real Constants"*) is ignored and a log-log table based on Constants 31 through 54 of the water motion table (*Table 2, "PIPE59 Water Motion Table"*) is used to determine C_D. If this capability is to be used, the viscosity, Re, and C_D constants must be input and none may be less than or equal to zero.

Similarly, if both Re(1) and CT(1) are positive, the value of C_T from the real constant table (*Table 1, "PIPE59 Real Constants"*) is ignored, and a log-log table based on Constants 31 through 42 and 55 through 66 of the water motion table (*Table 2, "PIPE59 Water Motion Table"*) is used to determine C_T. If this capability is to be used, the viscosity, Re, and C_T constants must be input and none may be less than or equal to zero.

Various wave theories may be selected with the KWAVE constant of the water motion table (*Table 2, "PIPE59 Water Motion Table"*). These are:

- Small Amplitude Wave Theory with empirical modification of depth decay function (KWAVE = 0)
- Small Amplitude Airy Wave Theory without modifications (KWAVE = 1)
- Stokes Fifth Order Wave Theory (KWAVE = 2)
- Stream Function Wave Theory (KWAVE = 3).

The wave loadings can be altered (KEYOPT(5)) so that horizontal position has no effect on the wave-induced forces.

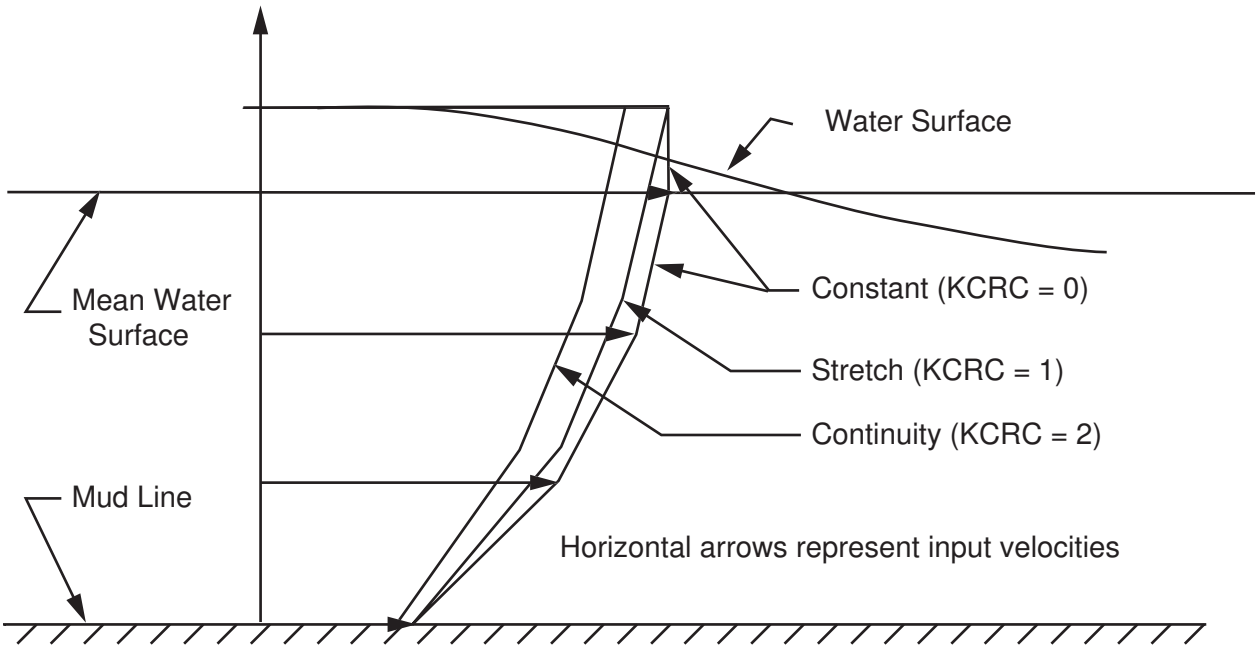
Wave loading depends on the acceleration due to gravity (ACELZ), and it *may not change* between substeps or load steps. Therefore, when performing an analysis using load steps with multiple substeps, the gravity may only be "stepped on" [**KBC**,1] and not ramped.

With the stream function wave theory (KWAVE = 3), the wave is described by alternate Constants 79 through 193 as shown in *Table 2, "PIPE59 Water Motion Table"*. The definitions of the constants correspond exactly to those given in the tables in *7.* for the forty cases of ratio of wave height and water depth to the deep water wave length. The other wave-related constants that the user inputs directly are the water density (DENSW), water depth (DEPTH), wave direction (Φ), and acceleration due to gravity (ACELZ). The wave height, length, and period are inferred from the tables. The user should verify the input by comparing the interpreted results (the columns headed DIMENSIONLESS under the STREAM FUNCTION INPUT VALUES printout) with the data presented in the *7.* tables. Note that this wave theory uses the current value defined for time [**TIME**] (which defaults to 1.0 for the first load step).

Several adjustments to the current profile are available with the KCRC constant of the water motion table as shown in *Figure 3, "PIPE59 Velocity Profiles for Wave-current Interactions"*. The adjustments are usually used only when the wave amplitude is large relative to the water depth, such that there is significant wave/current interaction. Options include

1. use the current profile (as input) for wave locations below the mean water level and the top current profile value for wave locations above the mean water level (KCRC = 0)
2. "stretch" (or compress) the current profile to the top of the wave (KCRC = 1)
3. same as (2) but also adjust the current profile horizontally such that total flow continuity is maintained with the input profile (KCRC = 2) (all current directions (θ(j)) must be the same for this option).

Figure 3 PIPE59 Velocity Profiles for Wave-current Interactions



Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE59 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. These pressures are in addition to the linearly varying pressure of the fluids on the inside and outside of the pipe. In handling the pressures, each element is assumed to be capped (that is, have closed ends). The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. The normal component or the projected full pressure may be used (KEYOPT(9)). See the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

Temperatures may be input as element body loads at the nodes. Temperatures may have wall gradients or diametral gradients (KEYOPT(3)). Diametral gradients are not valid for the cable option. The average wall temperature at $\theta = 0°$ is computed as 2 * TAVG - T(180) and the average wall temperature at $\theta = -90°$ is computed as 2 * TAVG - T(90). The element temperatures are assumed to be linear along the length. The first temperature at node I (TOUT(I) or TAVG(I)) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other pattern of input temperatures, unspecified temperatures default to TUNIF.

Eight temperatures (T(j)) are read as Constants 67-74 corresponding to the eight water depths (Z(j)) input as Constants 7-30. These temperatures override any other temperature input (except TREF) unless the element is entirely out of the water or if all eight temperatures are input as zero. The thermal load vector from these temperatures may not be scaled in a superelement use pass if an expansion pass is to follow. Constants 31 through 66 may have zero values if desired. The temperatures input as Constants 67-74 are used to compute a temperature-dependent viscosity based on linear interpolation (if previous constants are not all zero). In the case of a solid cross section (inside diameter = 0.0), they are also used to compute the material properties of the element.

For the mass matrix, the mass per unit length used for axial motion is the mass of the pipe wall (DENS), the external insulation (DENSIN), and the internal fluid together with the added mass of any additional hardware (CENMPL). The mass per unit length used for motion normal to the pipe is all of the above plus the added mass of the external fluid (DENSW).

CI should be 1.0 for a circular cross section. Values for other cross sections may be found in *8.*. The user should remember, however, that other properties of PIPE59 are based on a circular cross section.

PIPE59 Output Data

The solution output associated with the element is in two forms:

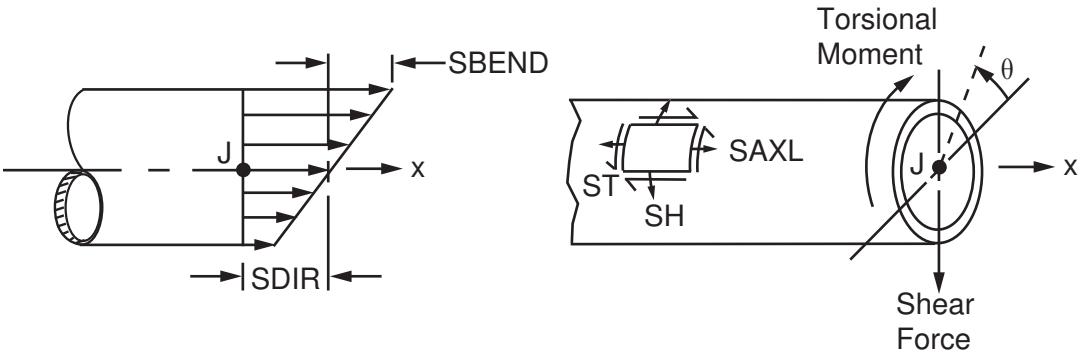
- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 3, "PIPE59 Element Output Definitions"*

Several items are illustrated in *Figure 4, "PIPE59 Stress Output"*. Note that the output is simplified and reduced if the cable option, KEYOPT(1) = 1, is used.

The principal stresses are computed at the two points around the circumference where the bending stresses are at a maximum. The principal stresses and the stress intensity include the shear force stress component. The principal stresses and the stress intensity are based on the stresses at two extreme points on opposite sides of the neutral axis. If KEYOPT(6) = 2, the 12-member forces and moments (6 at each end) are also printed (in the element coordinate system).

The axial force (FX) excludes the hydrostatic force component, as does the MFORX member force (printed if KEYOPT(6) = 2). If KWAVE = 2 or 3 (Stokes or Stream Function theory), additional wave information is also printed. If KEYOPT(7) = 1, detailed hydrodynamic information is printed at the immersed integration points. Angles listed in the output are measured (θ) as shown in *Figure 4, "PIPE59 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 4 PIPE59 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 PIPE59 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	*9*
LEN	Length	Y	-
PRES	Pressures PINTE (average effective internal pressure), PX, PY, PZ, POUTE (average effective external pressure)	Y	Y
STH	Stress due to maximum thermal gradient through the wall thickness	Y	Y
SPR2	Hoop pressure stress for code calculations	-	*1*
SMI, SMJ	Moment stress at nodes I and J for code calculations	-	*1*
SDIR	Direct (axial) stress	-	*1*
SBEND	Maximum bending stress at outer surface	-	*1*
ST	Shear stress at outer surface due to torsion	-	*1*
SSF	Shear stress due to shear force	-	*1*
S(1MX, 3MN, INTMX, EQVMX)	Maximum principal stress, minimum principal stress, maximum stress intensity, maximum equivalent stress (all at the outer surface)	*1*	*1*
TEMP	Temperatures TOUT(I), TIN(I), TOUT(J), TIN(J)	*2*	*2*
TEMP	Temperatures TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)	*3*	*3*
S(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*4*	*4*
S(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*4*	*4*
EPEL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*4*	*4*
EPTH(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*4*	*4*
MFOR(X, Y, Z)	Member forces for nodes I and J (in the element coordinate system)	*7*	*7*
MMOM(X, Y, Z)	Member moments for nodes I and J (in the element coordinate system)	*5*	*5*
NODE	Node I or J	*6*	*6*
FAXL	Axial force (excludes the hydrostatic force)	*6*	*6*
SAXL	Axial stress (includes the hydrostatic stress)	*6*	*6*
SRAD	Radial stress	*6*	*6*
SH	Hoop stress	*6*	*6*
SINT	Stress intensity	*6*	*6*
SEQV	Equivalent stress (SAXL minus the hydrostatic stress)	*6*	*6*
EPEL(AXL, RAD, H)	Axial, radial, and hoop elastic strains (excludes the thermal strain)	*6*	*6*
TEMP	TOUT(I), TOUT(J)	*6*	*6*
EPTHAXL	Axial thermal strains at nodes I and J	*6*	*6*
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	*8*	*8*

Name	Definition	O	R
AR, AZ	Radial and vertical fluid particle accelerations	*8*	*8*
PHDYN	Dynamic fluid pressure head	*8*	*8*
ETA	Wave amplitude over integration point	*8*	*8*
TFLUID	Fluid temperature (printed if VISC is nonzero)	*8*	*8*
VISC	Viscosity	*8*	*8*
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	*8*	*8*
CT, CD, CM	Input coefficients evaluated at Reynolds numbers	*8*	*8*
CTW, CDW	CT*DENSW*DO/2, CD*DENSW*DO/2	*8*	*8*
CMW	CM*DENSW*PI*DO**2/4	*8*	*8*
URT, URN	Tangential (parallel to element axis) and normal relative velocity	*8*	*8*
ABURN	Vector sum of normal (URN) velocities	*8*	*8*
AN	Accelerations normal to the element	*8*	*8*
FX, FY, FZ	Hydrodynamic forces tangential and normal to element axis	*8*	*8*
ARGU	Effective position of integration point (radians)	*8*	*8*

1. Output only for the pipe option (KEYOPT(1) = 0 or 2)
2. If KEYOPT(3) = 0 or if KEYOPT(1) = 1
3. If KEYOPT(3) = 1
4. Output only for the pipe option and the item repeats at 0, 45, 90, 135, 180, 225, 270, 315° at node I, then at node J (all at the outer surface)
5. Output only for the pipe option (KEYOPT(1) = 0 or 2) and if KEYOPT(6) = 2
6. Output only for the cable option (KEYOPT(1) = 1)
7. Output only if KEYOPT(6) = 2
8. Hydrodynamic solution (if KEYOPT(7) = 1 for immersed elements at integration points)
9. Available only at centroid as a ***GET** item.

Table 4, "PIPE59 Item and Sequence Numbers (Node I)" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 4, "PIPE59 Item and Sequence Numbers (Node I)"*:

Name
: output quantity as defined in the *Table 3, "PIPE59 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J
: sequence number for data at nodes I and J

Table 4 PIPE59 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	5	9	13	17	21	25	29
EPTHRAD	LEPTH	-	2	6	10	14	18	22	26	30
EPTHH	LEPTH	-	3	7	11	15	19	23	27	31
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	13	-	-	-	-	-	-	-	-
ST	SMISC	14	-	-	-	-	-	-	-	-
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	88	-	-	-	-	-	-	-	-
SSF	NMISC	89	-	-	-	-	-	-	-	-
TOUT	LBFE	-	4	-	1	-	2	-	3	-
TIN	LBFE	-	8	-	5	-	6	-	7	-

Table 5 PIPE59 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
EPTHAXL	LEPTH	-	33	37	41	45	49	53	57	61
EPTHRAD	LEPTH	-	34	38	42	46	50	54	58	62
EPTHH	LEPTH	-	35	39	43	47	51	55	59	63
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	15	-	-	-	-	-	-	-	-
ST	SMISC	16	-	-	-	-	-	-	-	-
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	90	-	-	-	-	-	-	-	-
SSF	NMISC	91	-	-	-	-	-	-	-	-
TOUT	LBFE	-	12	-	9	-	10	-	11	-
TIN	LBFE	-	16	-	13	-	14	-	15	-

Table 6 PIPE59 Item and Sequence Numbers (Pipe Options)

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
STH	SMISC	17
PINTE	SMISC	18
PX	SMISC	19
PY	SMISC	20
PZ	SMISC	21
POUTE	SMISC	22
SPR2	NMISC	81
SMI	NMISC	82
SMJ	NMISC	83
S1MX	NMISC	84
S3MN	NMISC	85
SINTMX	NMISC	86
SEQVMX	NMISC	87

Table 7 PIPE59 Item and Sequence Numbers (Cable Option)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	Node I	Node J
SAXL	LS		1	4
SRAD	LS		2	5
SH	LS		3	6
EPELAXL	LEPEL		1	4
EPELRAD	LEPEL		2	5
EPELH	LEPEL		3	6
EPTHAXL	LEPTH		1	4
TOUT	LBFE		1	9
TIN	LBFE		5	13
SINT	NMISC		4	9
SEQV	NMISC		5	10
FAXL	SMISC		1	6
STH	SMISC	13		
PINTE	SMISC	14		
PX	SMISC	15		
PY	SMISC	16		
PZ	SMISC	17		
POUTE	SMISC	18		

Table 8, "PIPE59 Item and Sequence Numbers (Additional Output)" lists additional print and post data file output available through the **ETABLE** command if KEYOPT(7) = 1.

Table 8 PIPE59 Item and Sequence Numbers (Additional Output)

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E- First Integration Point	E- Second Integration Point
GLOBAL COORD	NMISC	N + 1, N + 2, N + 3	N + 31, N + 32, N + 33
VR	NMISC	N + 4	N + 34
VZ	NMISC	N + 5	N + 35
AR	NMISC	N + 6	N + 36
AZ	NMISC	N + 7	N + 37
PHDY	NMISC	N + 8	N + 38
ETA	NMISC	N + 9	N + 39
TFLUID	NMISC	N + 10	N + 40
VISC	NMISC	N + 11	N + 41
REN	NMISC	N + 12	N + 42
RET	NMISC	N + 13	N + 43
CT	NMISC	N + 14	N + 44
CTW	NMISC	N + 15	N + 45
URT	NMISC	N + 16	N + 46

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E- First Integration Point	E- Second Integration Point
FX	NMISC	N + 17	N + 47
CD	NMISC	N + 18	N + 48
CDW	NMISC	N + 19	N + 49
URN	NMISC	N + 20, N + 21	N + 50, N + 51
ABURN	NMISC	N + 22	N + 52
FY	NMISC	N + 23	N + 53
CM	NMISC	N + 24	N + 54
CMW	NMISC	N + 25	N + 55
AN	NMISC	N + 26, N + 27	N + 56, N + 57
FZ	NMISC	N + 28	N + 58
ARGU	NMISC	N + 29	N + 59

Note

For the pipe option (KEYOPT(1) = 0 or 2): N = 99. For the cable option (KEYOPT(1) = 1): N = 10.

PIPE59 Assumptions and Restrictions

- The pipe must not have a zero length. In addition, the O.D. must not be less than or equal to zero and the I.D. must not be less than zero.
- Elements input at or near the water surface should be small in length relative to the wave length.
- Neither end of the element may be input below the mud line (ocean floor). Integration points that move below the mud line are presumed to have no hydrodynamic forces acting on them.
- If the element is used out of water, the water motion table (*Table 2, "PIPE59 Water Motion Table"*) need not be included.
- The element should also be used with caution in the reduced transient dynamic analysis since this analysis type ignores the element load vector. Fluid damping, if any, should be handled via the hydrodynamic load vector rather than α (mass matrix) damping.
- When performing a transient analysis, the solution may be unstable with small time steps due to the nature of Morrison's equation.
- The applied thermal gradient is assumed to vary linearly along the length of the element.
- The same water motion table (*Table 2, "PIPE59 Water Motion Table"*) should not be used for different wave theories in the same problem.
- The lumped mass matrix formulation [**LUMPM**,ON] is not allowed for PIPE59 when using "added mass" on the outside of the pipe (CI $\geq$ 0.0).

PIPE59 Product Restrictions

There are no product-specific restrictions for this element.

PIPE60

Plastic Curved Thin-Walled Pipe

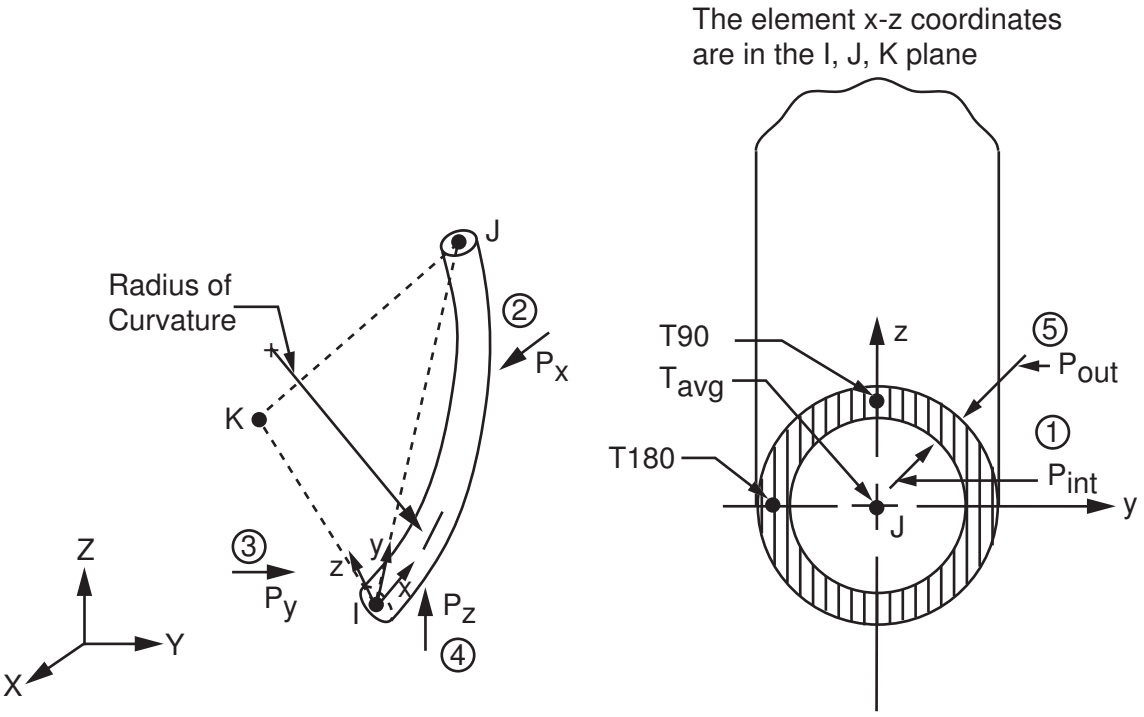
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

PIPE60 Element Description

PIPE60, also known as an elbow element, is a uniaxial element with tension-compression, bending, and torsion capabilities. The element has six degrees of freedom at each node: translations in the nodal, x, y, and z directions and rotations about the nodal x, y, and z-axes.

The element has plastic, creep and swelling capabilities. If these effects are not needed, the elastic curved pipe element, PIPE18, may be used. Options are available for including a flexibility factor and for printing the forces and moments acting on the element in the element coordinate system. See PIPE60 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PIPE20 for a plastic straight pipe element.

Figure 1 PIPE60 Geometry



PIPE60 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PIPE60 Geometry"*. The element input data include three nodes, the pipe outer diameter (OD) and wall thickness (TKWALL), radius of curvature (RADCUR), optional stress intensification (SIFI and SIFJ) and flexibility (FLXI and FLXO) factors, and the isotropic material properties.

Although the curved pipe element has only two end points (nodes I and J), the third node (K) is required to define the plane in which the element lies. This node must lie in the plane of the curved pipe and on the center of curvature side of line I-J. A node belonging to another element (such as the other node of a connecting straight pipe element) may be used.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PIPE60 Geometry"*. Internal pressure (PINT) and external pressure (POUT) are input as positive values. The internal and external pressure loads are designed for closed-loop static pressure environments and therefore include pressure loads on fictitious "end caps" so that the pressure loads induce an axial stress and/or reaction in the pipe system. If a dynamic situation needs to be represented, such as a pipe venting to a lower pressure area or the internal flow is past a constriction in the pipe, these end cap loads may need to be modified by applying a nodal force normal to the cross-section of the pipe with the magnitude representing the change in pressure. Alternatively, the precomputed end cap loads can be removed using KEYOPT(8) = 1 and the appropriate end cap loads added by the user. Note that when using KEYOPT(8) = 1, the pressure load will be acting on only the wall of the elbow element so that the total pressure load will not be self-equilibrating. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. Tapered pressures are not recognized. Only constant pressures are supported for this element. See PIPE60 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

Temperatures may be input as element body loads at the nodes. Temperatures may have wall gradients or diametral gradients (KEYOPT(3)). The average wall temperature at $\theta = 0°$ is computed as 2 * TAVG - T(180) and the average wall temperature at $\theta = -90°$ is computed as 2 * TAVG - T(90). The element temperatures are assumed to be linear along the length. The first temperature at node I (TOUT(I) or TAVG(I)) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures.

The KEYOPT(2) and KEYOPT(3) options control the flexibility and stress intensification factors as follows:

ANSYS Flexibility Factor = $1.65/(h(1 + PrX_k/tE))$ or 1.0, whichever is greater (used if KEYOPT(3) = 0 or 1)
Karman Flexibility Factor = $(10 + 12h^2)/(1 + 12h^2)$, used if KEYOPT(2) = 2
User Defined Flexibility Factors = FLXI (in-plane) and FLXO (out-of-plane), both must be positive (used if KEYOPT(3) = 3)
FLXO defaults to FLXI for all cases.
Reference Stress Intensification Factor (SIF) = $0.9/h^{2/3}$ or 1.0, whichever is greater. Used for SIFI or SIFJ if KEYOPT(2) = 0 or if user supplied SIF's are less than 2.0 (user supplied values must be positive).
User Defined Stress Intensification Factors = SIFI, SIFJ, must be positive (used if KEYOPT(2) = 4)

where:

$h = tR/r^2$
R = radius of curvature
t = thickness
r = average radius
E = modulus of elasticity
$X_k = 6\ (r/t)^{4/3}(R/r)1/3$ if KEYOPT(3) = 1 and $R/r \geq 1.7$, otherwise $X_k = 0$
$P = P_i - P_o$
P_i = internal pressure
P_o = external pressure

KEYOPT(3) = 1 should not be used if the included angle of the complete elbow is less than $360/(\pi(R/r))$ degrees.

A summary of the element input is given below. A general description of element input is given in *Section 2.1: Element Input*.

PIPE60 Input Summary

Nodes
I, J, K (K is a node in the plane of the elbow, on the center of curvature side of line I-J)

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

For KEYOPT(2) = 0 and KEYOPT(3) < 3
OD, TKWALL, RADCUR
For KEYOPT(2) = 4 and KEYOPT(3) < 3
OD, TKWALL, RADCUR, SIFI, SIFJ
For KEYOPT(2) = 4 and KEYOPT(3) = 3
OD, TKWALL, RADCUR, SIFI, SIFJ, FLXI,
(Blank), (Blank), (Blank), (Blank), (Blank), FLXO
See *Table 1, "PIPE60 Real Constants"* for details.

Material Properties
EX, ALPX (or CTEX or THSX), PRXY (or NUXY), DENS, GXY, DAMP

Surface Loads

Pressures --
1-PINT, 2-PX, 3-PY, 4-PZ, 5-POUT

Body Loads

Temperatures --
TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)

Fluences --
FLAVG(I), FL90(I), FL180(I), FLAVG(J), FL90(J), FL180(J)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Large deflection
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(2)
Stress intensification factor:

0 --
Include reference stress intensification factors (SIF)

4 --
Include stress intensification factors at nodes I and J as input with SIFI and SIFJ real constants

KEYOPT(3)
Flexibility factor:

0 --
Do not include pressure term in ANSYS flexibility factor

1 --
Include pressure term in ANSYS flexibility factor

2 --
Use Karman flexibility factor

3 --
Use input flexibility factors (FLXI, FLXO)

KEYOPT(6)
Member force and moment output:

0 --
No printout of member forces or moments

1 --
Print member forces and moments in the element coordinate system

KEYOPT(8)
End cap loads:

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

Table 1 PIPE60 Real Constants

No.	Name	Description
1	OD	Outer diameter
2	TKWALL	Wall thickness
3	RADCUR	Radius of curvature
4	SIFI	Stress intensity factor at node I
5	SIFJ	Stress intensity factor at node J
6	FLXI	User-supplied in-plane flexibility factor
7, ... 11	(blank)	unused
12	FLXO	User-supplied out-of-plane flexibility factor

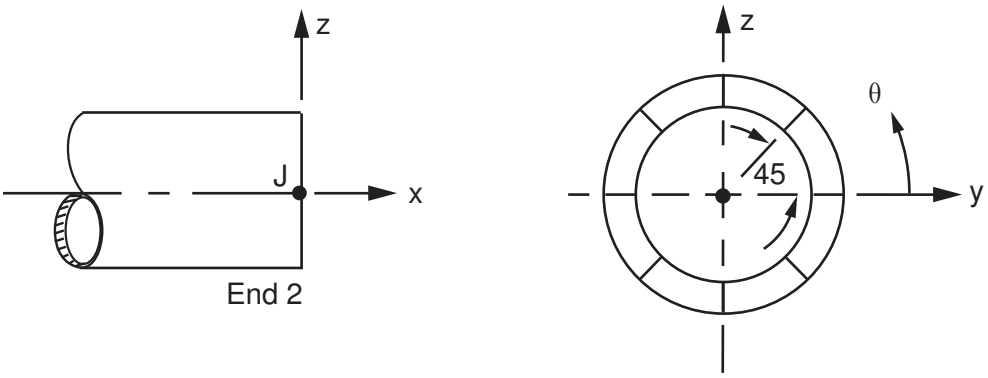
PIPE60 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "PIPE60 Element Output Definitions"*

The meaning of THETA is illustrated in *Figure 2, "PIPE60 Printout Locations"*. The nonlinear solution is given at eight circumferential locations at both ends of the elbow. The linear solution, similar to that for PIPE18, is printed as long as the element remains elastic. Only the bending stress (SBEND) is multiplied by the stress intensification factor (selected by KEYOPT(2)), provided the factor is greater than 1.0. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PIPE60 Printout Locations



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 PIPE60 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*4*
TEMP	Temperatures TAVG(I), T90(I), T180(I), TAVG(J), T90(J), T180(J)	Y	Y
FLUE	Fluences FLAVG(I), FL90(I), FL180(I), FLAVG(J), FL90(J), FL180(J)	Y	Y
PRES	Pressures PINT, PX, PY, PZ, POUT	Y	Y
FFACT	Element flexibility factor	-	Y
SFACTI, SFACTJ	Stress intensification factors at nodes I and J	-	Y
MFOR(X, Y, Z)	Member forces for nodes I and J (in the element coordinate system)	*1*	*1*
MMOM(X, Y, Z)	Member moments for nodes I and J (in the element coordinate system)	*1*	*1*
SDIR	Direct (axial) stress	-	*2*
SBEND	Maximum bending stress at outer surface	-	*2*
ST	Shear stress at outer surface due to torsion	-	*2*
SSF	Shear stress due to shear force	-	*2*
S1MX, S3MN, SINTMX, SEQVMX	Maximum principal stress, minimum principal stress, maximum stress intensity, maximum equivalent stress all at the outer surface (based on SDIR, SBEND, ST, SSF	*2*	*2*

Name	Definition	O	R
	but also accounting for the values of S1, S3, SINT, SEQV given below)		
S(AXL, RAD, H, XH)	Axial, radial, hoop, and shear stresses	*3*	*3*
S(1, 3, INT, EQV)	Maximum principal stress, minimum principal stress, stress intensity, equivalent stress	*3*	*3*
EPEL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear strains	*3*	*3*
EPTH(AXL, RAD, H)	Axial, radial, and hoop thermal strain	*3*	*3*
EPPL(AXL, RAD, H, XH)	Axial, radial, hoop, and shear plastic strains	*3*	*3*
EPCR(AXL, RAD, H, XH)	Axial, radial, hoop, and shear creep strains	*3*	*3*
SEPL	Equivalent stress from stress-strain curve	*3*	*3*
SRAT	Ratio of trial stress to stress on yield surface	*3*	*3*
HPRES	Hydrostatic pressure (postdata only)	-	*3*
EPEQ	Equivalent plastic strain	*3*	*3*
EPSWAXL	Axial swelling strain	*3*	*3*

1. If KEYOPT(6) = 1
2. Initial elastic solution output before yield
3. The item repeats for THETA = 0, 45, 90, 135, 180, 225, 270, 315° at node I, then at node J, all at the midthickness of the wall
4. Available only at centroid as a ***GET** item.

Table 3, "PIPE60 Item and Sequence Numbers (Node I)" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "PIPE60 Item and Sequence Numbers (Node I)"*:

Name
output quantity as defined in the *Table 2, "PIPE60 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 3 PIPE60 Item and Sequence Numbers (Node I)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	1	5	9	13	17	21	25	29
SRAD	LS	-	2	6	10	14	18	22	26	30
SH	LS	-	3	7	11	15	19	23	27	31
SXH	LS	-	4	8	12	16	20	24	28	32
EPELAXL	LEPEL	-	1	5	9	13	17	21	25	29
EPELRAD	LEPEL	-	2	6	10	14	18	22	26	30

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
EPELH	LEPEL	-	3	7	11	15	19	23	27	31
EPELXH	LEPEL	-	4	8	12	16	20	24	28	32
EPTHAXL	LEPTH	-	1	5	9	13	17	21	25	29
EPTHRAD	LEPTH	-	2	6	10	14	18	22	26	30
EPTHH	LEPTH	-	3	7	11	15	19	23	27	31
EPPLAXL	LEPPL	-	1	5	9	13	17	21	25	29
EPPLRAD	LEPPL	-	2	6	10	14	18	22	26	30
EPPLH	LEPPL	-	3	7	11	15	19	23	27	31
EPPLXH	LEPPL	-	4	8	12	16	20	24	28	32
EPCRAXL	LEPCR	-	1	5	9	13	17	21	25	29
EPCRRAD	LEPCR	-	2	6	10	14	18	22	26	30
EPCRH	LEPCR	-	3	7	11	15	19	23	27	31
EPCRXH	LEPCR	-	4	8	12	16	20	24	28	32
SEPL	NLIN	-	1	5	9	13	17	21	25	29
SRAT	NLIN	-	2	6	10	14	18	22	26	30
HPRES	NLIN	-	3	7	11	15	19	23	27	31
EPEQ	NLIN	-	4	8	12	16	20	24	28	32
S1	NMISC	-	1	6	11	16	21	26	31	36
S3	NMISC	-	3	8	13	18	23	28	33	38
SINT	NMISC	-	4	9	14	19	24	29	34	39
SEQV	NMISC	-	5	10	15	20	25	30	35	40
SBEND	NMISC	84	-	-	-	-	-	-	-	-
SSF	NMISC	85	-	-	-	-	-	-	-	-
S1MX	NMISC	104	-	-	-	-	-	-	-	-
S3MN	NMISC	105	-	-	-	-	-	-	-	-
SINTMX	NMISC	106	-	-	-	-	-	-	-	-
SEQVMX	NMISC	107	-	-	-	-	-	-	-	-
FOUT	NMISC	-	91	-	88	-	89	-	90	-
FIN	NMISC	-	95	-	92	-	93	-	94	-
MFORX	SMISC	1	-	-	-	-	-	-	-	-
MFORY	SMISC	2	-	-	-	-	-	-	-	-
MFORZ	SMISC	3	-	-	-	-	-	-	-	-
MMOMX	SMISC	4	-	-	-	-	-	-	-	-
MMOMY	SMISC	5	-	-	-	-	-	-	-	-
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-
SDIR	SMISC	13	-	-	-	-	-	-	-	-
ST	SMISC	14	-	-	-	-	-	-	-	-
TOUT	LBFE	-	4	-	1	-	2	-	3	-
TIN	LBFE	-	8	-	5	-	6	-	7	-

Table 4 PIPE60 Item and Sequence Numbers (Node J)

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
SAXL	LS	-	33	37	41	45	49	53	57	61
SRAD	LS	-	34	38	42	46	50	54	58	62
SH	LS	-	35	39	43	47	51	55	59	63
SXH	LS	-	36	40	44	48	52	56	60	64
EPELAXL	LEPEL	-	33	37	41	45	49	53	57	61
EPELRAD	LEPEL	-	34	38	42	46	50	54	58	62
EPELH	LEPEL	-	35	39	43	47	51	55	59	63
EPELXH	LEPEL	-	36	40	44	48	52	56	60	64
EPTHAXL	LEPTH	-	33	37	41	45	49	53	57	61
EPTHRAD	LEPTH	-	34	38	42	46	50	54	58	62
EPTHH	LEPTH	-	35	39	43	47	51	55	59	63
EPPLAXL	LEPPL	-	33	37	41	45	49	53	57	61
EPPLRAD	LEPPL	-	34	38	42	46	50	54	58	62
EPPLH	LEPPL	-	35	39	43	47	51	55	59	63
EPPLXH	LEPPL	-	36	40	44	48	52	56	60	64
EPCRAXL	LEPCR	-	33	37	41	45	49	53	57	61
EPCRRAD	LEPCR	-	34	38	42	46	50	54	58	62
EPCRH	LEPCR	-	35	39	43	47	51	55	59	63
EPCRXH	LEPCR	-	36	40	44	48	52	56	60	64
SEPL	NLIN	-	33	37	41	45	49	53	57	61
SRAT	NLIN	-	34	38	42	46	50	54	58	62
HPRES	NLIN	-	35	39	43	47	51	55	59	63
EPEQ	NLIN	-	36	40	44	48	52	56	60	64
S1	NMISC	-	41	46	51	56	61	66	71	76
S3	NMISC	-	43	48	53	58	63	68	73	78
SINT	NMISC	-	44	49	54	59	64	69	74	79
SEQV	NMISC	-	45	50	55	60	65	70	75	80
SBEND	NMISC	86	-	-	-	-	-	-	-	-
SSF	NMISC	87	-	-	-	-	-	-	-	-
S1MX	NMISC	108	-	-	-	-	-	-	-	-
S3MN	NMISC	109	-	-	-	-	-	-	-	-
SINTMX	NMISC	110	-	-	-	-	-	-	-	-
SEQVMX	NMISC	111	-	-	-	-	-	-	-	-
FOUT	NMISC	-	99	-	96	-	97	-	98	-
FIN	NMISC	-	103	-	100	-	101	-	102	-
MFORX	SMISC	7	-	-	-	-	-	-	-	-
MFORY	SMISC	8	-	-	-	-	-	-	-	-
MFORZ	SMISC	9	-	-	-	-	-	-	-	-
MMOMX	SMISC	10	-	-	-	-	-	-	-	-
MMOMY	SMISC	11	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	E	Circumferential Location							
			0°	45°	90°	135°	180°	225°	270°	315°
MMOMZ	SMISC	12	-	-	-	-	-	-	-	-
SDIR	SMISC	15	-	-	-	-	-	-	-	-
ST	SMISC	16	-	-	-	-	-	-	-	-
TOUT	LBFE	-	12	-	9	-	10	-	11	-
TIN	LBFE	-	16	-	13	-	14	-	15	-

Table 5 PIPE60 Item and Sequence Numbers (Additional Output)

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SFACTI	NMISC	81
SFACTJ	NMISC	82
FFACT	NMISC	83
PINT	SMISC	17
PX	SMISC	18
PY	SMISC	19
PZ	SMISC	20
POUT	SMISC	21

PIPE60 Assumptions and Restrictions

- The curved pipe must not have a zero length or wall thickness. In addition, the O.D. must not be less than or equal to zero and the I.D. must not be less than zero. The three nodes must not be colinear.
- The element is limited to having an axis with a single curvature and a subtended angle of $0° < \theta \leq 90°$ since there are integration points only at each end of the element.
- When loaded with an in-plane strain gradient (thermal, plastic, creep, or swelling) a very fine mesh of elements is recommended.
- If there are effects other than internal pressure and in-plane bending, the elements should have a subtended angle no larger than 45°.
- The pipe element is assumed to have "closed ends" so that the axial pressure effect is included.
- Shear deflection capability is also included in the element formulation.
- If this element is used in a large deflection analysis, it should be noted that the location of the third node (K) is used only to initially orient the element.
- The element formulation is based upon thin-walled theory. The elbow should have a large radius-to-thickness ratio since the integration points are assumed to be located at the midthickness of the wall. If the ratio is less than 5.0 (OD/TKWALL = 10.0), an error message will be generated. If the ratio is less than 10.0 (OD/TKWALL = 20.0), a warning message will be generated.
- The elastic stiffness matrix is used in plasticity analyses (no tangent matrix is formed) and plasticity convergence may be slow.

PIPE60 Product Restrictions

There are no product-specific restrictions for this element.

SHELL61

Axisymmetric-Harmonic Structural Shell

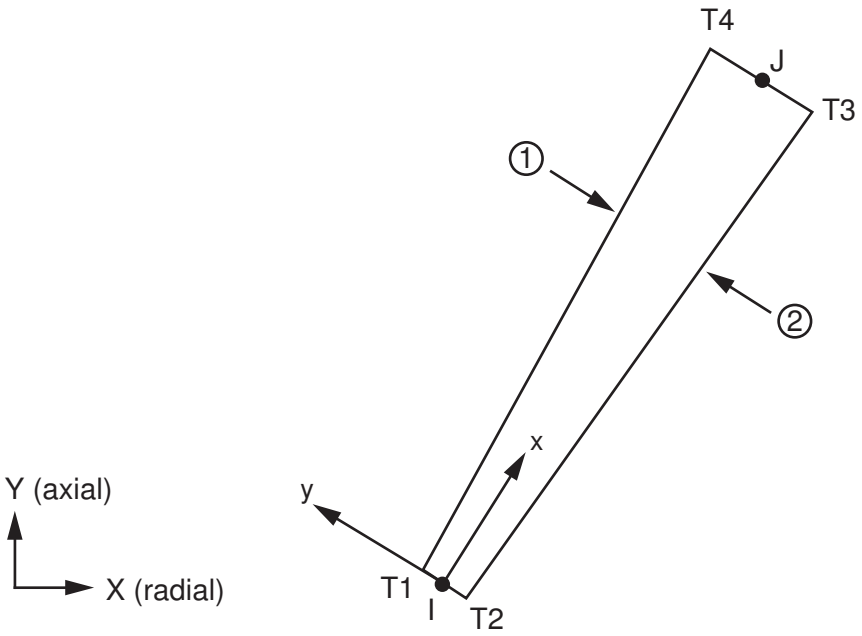
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SHELL61 Element Description

SHELL61 has four degrees of freedom at each node: translations in the nodal x, y, and z directions and a rotation about the nodal z-axis. The loading may be axisymmetric or nonaxisymmetric. Various loading cases are described in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

Extreme orientations of the conical shell element result in a cylindrical shell element or an annular disc element. The shell element may have a linearly varying thickness. See SHELL61 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL61 Geometry



SHELL61 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL61 Geometry"*. The element is defined by two nodes, two end thicknesses, the number of harmonic waves (*MODE* on the **MODE** command), a symmetry condition (*ISYM* on the **MODE** command), and the orthotropic material properties. The element coordinate system is shown in *Figure 2, "SHELL61 Stress Output"*. θ is in the tangential (hoop) direction. The *MODE* or *ISYM* parameters are discussed in detail in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

The material may be orthotropic, with nine elastic constants required for its description. The element loading may be input as any combination of harmonically varying temperatures and pressures. Harmonically varying nodal forces, if any, should be input on a full 360° basis.

The element may have variable thickness. The thickness is assumed to vary linearly between the nodes. If the element has a constant thickness, only TK(I) is required. Real constant ADMSUA is used to define an added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL61 Geometry"*. Positive pressures act into the element. The pressures are applied at the surface of the element rather than at the centroidal plane so that some thickness effects can be considered. These include the increase or decrease in size of surface area the load is acting on and (in the case of a nonzero Poisson's ratio) an interaction effect causing the element to grow longer or shorter under equal pressures on both surfaces. Material properties EY, PRXY, and PRYZ (or EY, NUXY, and NUYZ) are required for this effect.

Harmonically varying temperatures may be input as element body loads at the four corner locations shown in *Figure 1, "SHELL61 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(1) is used for temperature loading with *MODE* greater than zero and temperature dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If *MODE* equals zero, the material properties are always evaluated at the average element temperature. KEYOPT(3) is used to include or suppress the extra displacement shapes.

A summary of the element input is given in *SHELL61 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL61 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ, ROTZ

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J (TK(J) defaults to TK(I))
ADMSUA - Added mass/unit area

Material Properties
EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXZ, DAMP. (X is meridional, Y is through-the-thickness, and Z is circumferential.)

Surface Loads

Pressures --

face 1 (I-J) (top, in -Y direction)
face 2 (I-J) (bottom, in +Y direction)

Body Loads

Temperatures --
T1, T2, T3, T4

Mode Number --
Input mode number on **MODE** command

Special Features
Stress stiffening

Loading Condition --
Input for *ISYM* on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

KEYOPT(1)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(3)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(4)
Member force and moment output:

0 --
No printout of member forces and moments

1 --
Print out member forces and moments in the element coordinate system

KEYOPT(6)
Location of element solution output:

0 --
Output solution at mid-length only

N --
Output solution at *N* equally spaced interior points and at end points (where *N* = 1, 3, 5, 7 or 9)

SHELL61 Output Data

The solution output associated with the element is in two forms:

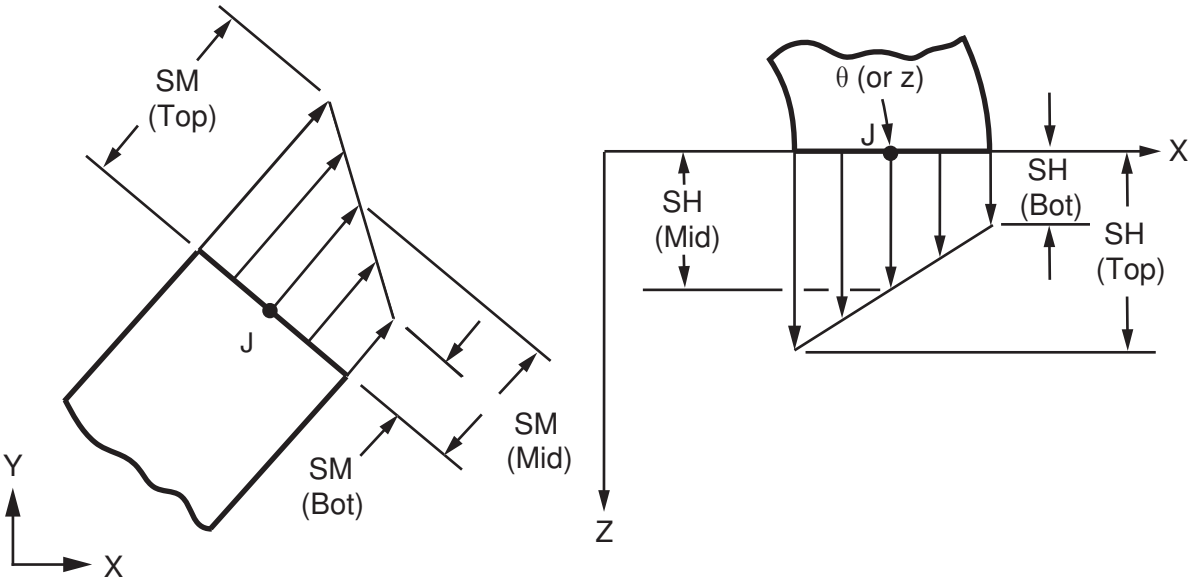
- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL61 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL61 Stress Output"*. The printout may be displayed at the centroid, at the end points and at *N* equally spaced interior points, where *N* is the KEYOPT(6) value. For example, if *N* = 3, printout will be produced at end I, 1/4 length, mid-length (centroid), 3/4 length, and at end J. Printout location number 1 is always at end I. Stress components which are inherently zero are printed for clarity.

In the displacement printout, the UZ components are out-of-phase with the UX and UY components. For example, in the *MODE* = 1, *ISYM* = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL61 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL61 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
LENGTH	Distance between node I and node J	Y	Y
XC, YC	Location where results are reported	Y	*2*
TEMP	Temperatures T1, T2, T3, T4	Y	Y
PRES	Pressures P1 (top) at nodes I,J; P2 (bottom) at nodes I,J	Y	Y
MODE	Number of waves in loading	Y	Y
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	Y
T(X, Z, XZ)	In-plane element X, Z, and XZ forces at KEYOPT(6) location(s)	Y	Y
M(X, Z, XZ)	Out-of-plane element X, Z, and XZ moments at KEYOPT(6) location(s)	Y	Y
MFOR(X, Y, Z), MMOMZ	Member forces and member moment for each node in the element coordinate system	*1*	Y
PK ANG	Angle where stresses have peak values: 0 and 90/MODE°. Blank if *MODE* = 0.	Y	Y

Name	Definition	O	R
S(M, THK, H, MH)	Stresses (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y
EPEL(M, THK, H, MH)	Elastic strains (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y
EPTH(M, THK, H, MH)	Thermal strains (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y

1. These items are printed only if KEYOPT(4) = 1.
2. Available only at centroid as a ***GET** item.

Table 2, "SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1)" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1)"*:

Name
output quantity as defined in the *Table 1, "SHELL61 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J
sequence number for data at nodes I and J

IL*n*
sequence number for data at Intermediate Location *n*

Table 2 SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	I	IL1	J
Top				
SM	LS	1	13	25
STHK	LS	2	14	26
SH	LS	3	15	27
SMH	LS	4	16	28
EPELM	LEPEL	1	13	25
EPELTHK	LEPEL	2	14	26
EPELH	LEPEL	3	15	27
EPELMH	LEPEL	4	16	28
EPTHM	LEPTH	1	13	25
EPTHTHK	LEPTH	2	14	26
EPTHH	LEPTH	3	15	27
EPTHMH	LEPTH	4	16	28
Mid				
SM	LS	5	17	29

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	I	IL1	J
STHK	LS	6	18	30
SH	LS	7	19	31
SMH	LS	8	20	32
EPELM	LEPEL	5	17	29
EPELTHK	LEPEL	6	18	30
EPELH	LEPEL	7	19	31
EPELMH	LEPEL	8	20	32
EPTHM	LEPTH	5	17	29
EPTHTHK	LEPTH	6	18	30
EPTHH	LEPTH	7	19	31
EPTHMH	LEPTH	8	20	32
Bot				
SM	LS	9	21	33
STHK	LS	10	22	34
SH	LS	11	23	35
SMH	LS	12	24	36
EPELM	LEPEL	9	21	33
EPELTHK	LEPEL	10	22	34
EPELH	LEPEL	11	23	35
EPELMH	LEPEL	12	24	36
EPTHM	LEPTH	9	21	33
EPTHTHK	LEPTH	10	22	34
EPTHH	LEPTH	11	23	35
EPTHMH	LEPTH	12	24	36
Element				
MFORX	SMISC	1	-	7
MFORY	SMISC	2	-	8
MFORZ	SMISC	3	-	9
MMOMZ	SMISC	6	-	12
TX	SMISC	13	19	25
TZ	SMISC	14	20	26
TXZ	SMISC	15	21	27
MX	SMISC	16	22	28
MZ	SMISC	17	23	29
MXZ	SMISC	18	24	30
P1	SMISC	31	-	32
P2	SMISC	35	-	36

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 3 SHELL61 Item and Sequence Numbers (KEYOPT(6) = 3)

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	IL1	IL2	IL3	J
Top						
SM	LS	1	13	25	37	49
STHK	LS	2	14	26	38	50
SH	LS	3	15	27	39	51
SMH	LS	4	16	28	40	52
EPELM	LEPEL	1	13	25	37	49
EPELTHK	LEPEL	2	14	26	38	50
EPELH	LEPEL	3	15	27	39	51
EPELMH	LEPEL	4	16	28	40	52
EPTHM	LEPTH	1	13	25	37	49
EPTHTHK	LEPTH	2	14	26	38	50
EPTHH	LEPTH	3	15	27	39	51
EPTHMH	LEPTH	4	16	28	40	52
Mid						
SM	LS	5	17	29	41	53
STHK	LS	6	18	30	42	54
SH	LS	7	19	31	43	55
SMH	LS	8	20	32	44	56
EPELM	LEPEL	5	17	29	41	53
EPELTHK	LEPEL	6	18	30	42	54
EPELH	LEPEL	7	19	31	43	55
EPELMH	LEPEL	8	20	32	44	56
EPTHM	LEPTH	5	17	29	41	53
EPTHTHK	LEPTH	6	18	30	42	54
EPTHH	LEPTH	7	19	31	43	55
EPTHMH	LEPTH	8	20	32	44	56
Bot						
SM	LS	9	21	33	45	57
STHK	LS	10	22	34	46	58
SH	LS	11	23	35	47	59
SMH	LS	12	24	36	48	60
EPELM	LEPEL	9	21	33	45	57
EPELTHK	LEPEL	10	22	34	46	58
EPELH	LEPEL	11	23	35	47	59
EPELMH	LEPEL	12	24	36	48	60
EPTHM	LEPTH	9	21	33	45	57
EPTHTHK	LEPTH	10	22	34	46	58
EPTHH	LEPTH	11	23	35	47	59
EPTHMH	LEPTH	12	24	36	48	60
Element						

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	IL1	IL2	IL3	J
MFORX	SMISC	1	-	-	-	7
MFORY	SMISC	2	-	-	-	8
MFORZ	SMISC	3	-	-	-	9
MMOMZ	SMISC	6	-	-	-	12
TX	SMISC	13	19	25	31	37
TZ	SMISC	14	20	26	32	38
TXZ	SMISC	15	21	27	33	39
MX	SMISC	16	22	28	34	40
MZ	SMISC	17	23	29	35	41
MXZ	SMISC	18	24	30	36	42
P1	SMISC	43	-	-	-	44
P2	SMISC	47	-	-	-	48

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 4 SHELL61 Item and Sequence Numbers (KEYOPT(6) = 5)

Output Quantity Name	ETABLE and ESOL Command Input							
	Item	I	IL1	IL2	IL3	IL4	IL5	J
Top								
SM	LS	1	13	25	37	49	61	73
STHK	LS	2	14	26	38	50	62	74
SH	LS	3	15	27	39	51	63	75
SMH	LS	4	16	28	40	52	64	76
EPELM	LEPEL	1	13	25	37	49	61	73
EPELTHK	LEPEL	2	14	26	38	50	62	74
EPELH	LEPEL	3	15	27	39	51	63	75
EPELMH	LEPEL	4	16	28	40	52	64	76
EPTHM	LEPTH	1	13	25	37	49	61	73
EPTHTHK	LEPTH	2	14	26	38	50	62	74
EPTHH	LEPTH	3	15	27	39	51	63	75
EPTHMH	LEPTH	4	16	28	40	52	64	76
Mid								
SM	LS	5	17	29	41	53	65	77
STHK	LS	6	18	30	42	54	66	78
SH	LS	7	19	31	43	55	67	79
SMH	LS	8	20	32	44	56	68	80
EPELM	LEPEL	5	17	29	41	53	65	77
EPELTHK	LEPEL	6	18	30	42	54	66	78
EPELH	LEPEL	7	19	31	43	55	67	79

Output Quantity Name	ETABLE and ESOL Command Input							
	Item	I	IL1	IL2	IL3	IL4	IL5	J
EPELMH	LEPEL	8	20	32	44	56	68	80
EPTHM	LEPTH	5	17	29	41	53	65	77
EPTHTHK	LEPTH	6	18	30	42	54	66	78
EPTHH	LEPTH	7	19	31	43	55	67	79
EPTHMH	LEPTH	8	20	32	44	56	68	80
Bot								
SM	LS	9	21	33	45	57	69	81
STHK	LS	10	22	34	46	58	70	82
SH	LS	11	23	35	47	59	71	83
SMH	LS	12	24	36	48	60	72	84
EPELM	LEPEL	9	21	33	45	57	69	81
EPELTHK	LEPEL	10	22	34	46	58	70	82
EPELH	LEPEL	11	23	35	47	59	71	83
EPELMH	LEPEL	12	24	36	48	60	72	84
EPTHM	LEPTH	9	21	33	45	57	69	81
EPTHTHK	LEPTH	10	22	34	46	58	70	82
EPTHH	LEPTH	11	23	35	47	59	71	83
EPTHMH	LEPTH	12	24	36	48	60	72	84
Element								
MFORX	SMISC	1	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	12
TX	SMISC	13	19	25	31	37	43	49
TZ	SMISC	14	20	26	32	38	44	50
TXZ	SMISC	15	21	27	33	39	45	51
MX	SMISC	16	22	28	34	40	46	52
MZ	SMISC	17	23	29	35	41	47	53
MXZ	SMISC	18	24	30	36	42	48	54
P1	SMISC	55	-	-	-	-	-	56
P2	SMISC	59	-	-	-	-	-	60

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 5 SHELL61 Item and Sequence Numbers (KEYOPT(6) = 7)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
Top										
SM	LS	1	13	25	37	49	61	73	85	97

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
STHK	LS	2	14	26	38	50	62	74	86	98
SH	LS	3	15	27	39	51	63	75	87	99
SMH	LS	4	16	28	40	52	64	76	88	100
EPELM	LEPEL	1	13	25	37	49	61	73	85	97
EPELTHK	LEPEL	2	14	26	38	50	62	74	86	98
EPELH	LEPEL	3	15	27	39	51	63	75	87	99
EPELMH	LEPEL	4	16	28	40	52	64	76	88	100
EPTHM	LEPTH	1	13	25	37	49	61	73	85	97
EPTHTHK	LEPTH	2	14	26	38	50	62	74	86	98
EPTHH	LEPTH	3	15	27	39	51	63	75	87	99
EPTHMH	LEPTH	4	16	28	40	52	64	76	88	100
Mid										
SM	LS	5	17	29	41	53	65	77	89	101
STHK	LS	6	18	30	42	54	66	78	90	102
SH	LS	7	19	31	43	55	67	79	91	103
SMH	LS	8	20	32	44	56	68	80	92	104
EPELM	LEPEL	5	17	29	41	53	65	77	89	101
EPELTHK	LEPEL	6	18	30	42	54	66	78	90	102
EPELH	LEPEL	7	19	31	43	55	67	79	91	103
EPELMH	LEPEL	8	20	32	44	56	68	80	92	104
EPTHM	LEPTH	5	17	29	41	53	65	77	89	101
EPTHTHK	LEPTH	6	18	30	42	54	66	78	90	102
EPTHH	LEPTH	7	19	31	43	55	67	79	91	103
EPTHMH	LEPTH	8	20	32	44	56	68	80	92	104
Bot										
SM	LS	9	21	33	45	57	69	81	93	105
STHK	LS	10	22	34	46	58	70	82	94	106
SH	LS	11	23	35	47	59	71	83	95	107
SMH	LS	12	24	36	48	60	72	84	96	108
EPELM	LEPEL	9	21	33	45	57	69	81	93	105
EPELTHK	LEPEL	10	22	34	46	58	70	82	94	106
EPELH	LEPEL	11	23	35	47	59	71	83	95	107
EPELMH	LEPEL	12	24	36	48	60	72	84	96	108
EPTHM	LEPTH	9	21	33	45	57	69	81	93	105
EPTHTHK	LEPTH	10	22	34	46	58	70	82	94	106
EPTHH	LEPTH	11	23	35	47	59	71	83	95	107
EPTHMH	LEPTH	12	24	36	48	60	72	84	96	108
Element										
MFORX	SMISC	1	-	-	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	-	-	12

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
TX	SMISC	13	19	25	31	37	43	49	55	61
TZ	SMISC	14	20	26	32	38	44	50	56	62
TXZ	SMISC	15	21	27	33	39	45	51	57	63
MX	SMISC	16	22	28	34	40	46	52	58	64
MZ	SMISC	17	23	29	35	41	47	53	59	65
MXZ	SMISC	18	24	30	36	42	48	54	60	66
P1	SMISC	67	-	-	-	-	-	-	-	68
P2	SMISC	71	-	-	-	-	-	-	-	72

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 6 SHELL61 Item and Sequence Numbers (KEYOPT(6) = 9)

Output Quantity Label	ETABLE and ESOL Command Input											
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
Top												
SM	LS	1	13	25	37	49	61	73	85	97	109	121
STHK	LS	2	14	26	38	50	62	74	86	98	110	122
SH	LS	3	15	27	39	51	63	75	87	99	111	123
SMH	LS	4	16	28	40	52	64	76	88	100	112	124
EPELM	LEPEL	1	13	25	37	49	61	73	85	97	109	121
EPELTHK	LEPEL	2	14	26	38	50	62	74	86	98	110	122
EPELH	LEPEL	3	15	27	39	51	63	75	87	99	111	123
EPELMH	LEPEL	4	16	28	40	52	64	76	88	100	112	124
EPTHM	LEPTH	1	13	25	37	49	61	73	85	97	109	121
EPTHTHK	LEPTH	2	14	26	38	50	62	74	86	98	110	122
EPTHH	LEPTH	3	15	27	39	51	63	75	87	99	111	123
EPTHMH	LEPTH	4	16	28	40	52	64	76	88	100	112	124
Mid												
SM	LS	5	17	29	41	53	65	77	89	101	113	125
STHK	LS	6	18	30	42	54	66	78	90	102	114	126
SH	LS	7	19	31	43	55	67	79	91	103	115	127
SMH	LS	8	20	32	44	56	68	80	92	104	116	128
EPELM	LEPEL	5	17	29	41	53	65	77	89	101	113	125
EPELTHK	LEPEL	6	18	30	42	54	66	78	90	102	114	126
EPELH	LEPEL	7	19	31	43	55	67	79	91	103	115	127
EPELMH	LEPEL	8	20	32	44	56	68	80	92	104	116	128
EPTHM	LEPTH	5	17	29	41	53	65	77	89	101	113	125
EPTHTHK	LEPTH	6	18	30	42	54	66	78	90	102	114	126
EPTHH	LEPTH	7	19	31	43	55	67	79	91	103	115	127

Output Quantity Label	ETABLE and ESOL Command Input											
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
EPTHMH	LEPTH	8	20	32	44	56	68	80	92	104	116	128
Bot												
SM	LS	9	21	33	45	57	69	81	93	105	117	129
STHK	LS	10	22	34	46	58	70	82	94	106	118	130
SH	LS	11	23	35	47	59	71	83	95	107	119	131
SMH	LS	12	24	36	48	60	72	84	96	108	120	132
EPELM	LEPEL	9	21	33	45	57	69	81	93	105	117	129
EPELTHK	LEPEL	10	22	34	46	58	70	82	94	106	118	130
EPELH	LEPEL	11	23	35	47	59	71	83	95	107	119	131
EPELMH	LEPEL	12	24	36	48	60	72	84	96	108	120	132
EPTHM	LEPTH	9	21	33	45	57	69	81	93	105	117	129
EPTHTHK	LEPTH	10	22	34	46	58	70	82	94	106	118	130
EPTHH	LEPTH	11	23	35	47	59	71	83	95	107	119	131
EPTHMH	LEPTH	12	24	36	48	60	72	84	96	108	120	132
Element												
MFORX	SMISC	1	-	-	-	-	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-	-	12
TX	SMISC	13	19	25	31	37	43	49	55	61	67	73
TZ	SMISC	14	20	26	32	38	44	50	56	62	68	74
TXZ	SMISC	15	21	27	33	39	45	51	57	63	69	75
MX	SMISC	16	22	28	34	40	46	52	58	64	70	76
MZ	SMISC	17	23	29	35	41	47	53	59	65	71	77
MXZ	SMISC	18	24	30	36	42	48	54	60	66	72	78
P1	SMISC	79	-	-	-	-	-	-	-	-	-	80
P2	SMISC	83	-	-	-	-	-	-	-	-	-	84

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

SHELL61 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane and must not have a zero length. Both ends must have nonnegative X coordinate values and the element must not lie along the global Y-axis.
- If the element has a constant thickness, only TK(I) need be defined. TK(I) must not be zero.
- The element thickness is assumed to vary linearly from node I to node J. Some thick shell effects have been included in the formulation of SHELL61 but it cannot be properly considered to be a thick shell element. If these effects are important, it is recommended to use PLANE25.
- The element assumes a linear elastic material.
- Post analysis superposition of results is valid only with other linear elastic solutions.

- Strain energy does not consider thermal effects.
- The element should not be used with the large deflection option.
- The element may not be deactivated with the **EKILL** command.
- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.

SHELL61 Product Restrictions

There are no product restrictions for this element.

SOLID62

3-D Magneto-Structural Solid

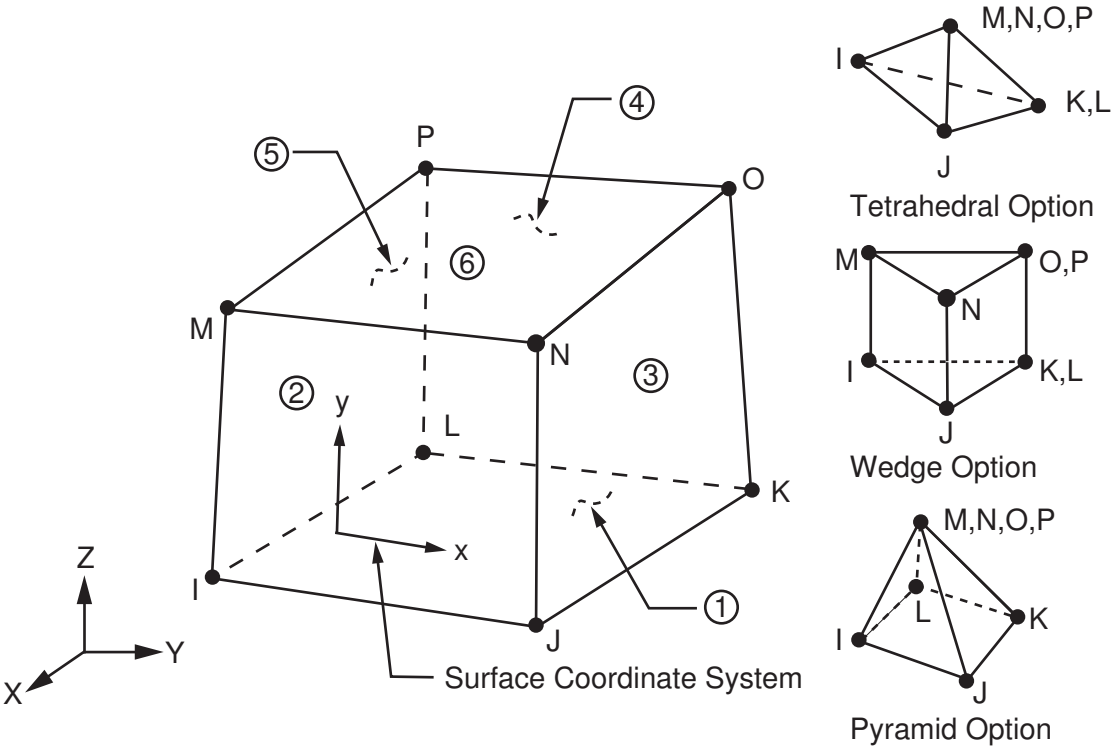
MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SOLID62 Element Description

SOLID62 has the capability of modeling 3-D coupled magneto-structural fields. The magnetic formulation uses a vector potential (AX, AY, AZ) in static analysis and a vector potential combined with a time-integrated scalar potential (VOLT) for harmonic and transient analysis. The structural formulation is similar to that in the SOLID45 element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Other elements with magneto-structural capability are PLANE13, SOLID5 and SOLID98. Magneto-structural coupling is not available for harmonic analysis. See SOLID62 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

The element has nonlinear magnetic harmonic capability for modeling B-H curves or permanent magnet demagnetization curves.

Figure 1 SOLID62 Geometry



SOLID62 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID62 Geometry"*. The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID62 Geometry"*. The type of units (MKS or user defined) for magnetic field analysis is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4 \pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is available and is specified through the MURX, MURY,

and MURZ material property labels. Orthotropic resistivity is specified through the RSVX, RSVY, and RSVZ material labels.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Nonlinear magnetic properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (UX, UY, UZ, AX, AY, AZ, VOLT) and *VALUE* corresponds to the value (displacement, magnetic vector potential, and time-integrated electric scalar potential). With the **F** command, the *Lab* variable corresponds to the force (FX, FY, FZ, CSGX, CSGY, CSGZ, AMPS) and *VALUE* corresponds to the value (force, magnetic current segments, and current).

Element loads are described in *Section 2.8: Node and Element Loads*. The surface loads; pressure and Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID62 Geometry"* using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required). A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

The body loads; temperature (structural), magnetic virtual displacement, fluence, and source current density may be input based on their value at the element's nodes or as a single element value (**BF** and **BFE** commands.) In general, unspecified nodal values of temperatures and fluence default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. The vector components of the current density are with respect to the element coordinate system. Calculated Joule heating (JHEAT) may be made available for a subsequent thermal analysis with companion elements (**LDREAD** command).

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI (Magnetic Virtual Displacements) label (**BF** command). See the *Low-Frequency Electromagnetic Analysis Guide* for details.

A summary of the element input is given in *SOLID62 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID62 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ, AX, AY, AZ, VOLT

Real Constants
None

Material Properties

MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ,
MGXX, MGYY, MGZZ plus BH data table (see *Section 2.5: Data Tables - Implicit Analysis*),

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
(PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
DENS, GXY, GXZ, GYZ, DAMP

Surface Loads

Maxwell Force Flags --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Pressures --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

Fluences --
FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)

Source Current Density --
JSX(I), JSY(I), JSZ(I), PHASE(I), JSX(J), JSY(J), JSZ(J), PHASE(J), JSX(K), JSY(K), JSZ(K), PHASE(K), JSX(L), JSY(L), JSZ(L), PHASE(L), JSX(M), JSY(M), JSZ(M), PHASE(M), JSX(N), JSY (N), JSZ(N), PHASE(N), JSX(O), JSY(O), JSZ(O), PHASE(O), JSX(P), JSY(P), JSZ(P), PHASE(P)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Magneto-structural coupling (requires an iterative solution for field coupling)
Birth and death
Adaptive descent

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field and stress printout

KEYOPT(6)
Extra surface output:

0 --
Basic element solution

1 --
Structural surface solution for face I-J-N-M also

2 --
Structural surface solution for face I-J-N-M and face K-L-P-O (Surface solution available for linear materials only)

3 --
Structural nonlinear solution at each integration point also

4 --
Structural surface solution for faces with nonzero pressure

SOLID62 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID62 Structural Element Output Definitions"*

The element output directions are parallel to the element coordinate system.

The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate systems and are available for any face (KEYOPT(6)). The coordinate systems for faces IJNM and KLPO are shown in *Figure 1, "SOLID62 Geometry"*. The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID62 Structural Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y

Name	Definition	O	R
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Input temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
FLUEN	Input fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)	Y	-
S:X, Y, Z, XY, YZ, XZ	Stresses (X, Y, Z, XY, YZ, XZ)	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [*4*]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strain [*4*]	*1*	*1*
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strain	*1*	*1*
EPPL:EQV	Equivalent plastic strain [*4*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strain	*1*	*1*
EPCR:EQV	Equivalent creep strain [*4*]	*1*	*1*
EPSW:	Average swelling strain	*1*	*1*
NL:EPEQ	Average equivalent plastic strain	*1*	*1*
NL:SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
NL:SEPL	Average equivalent stress from stress-strain curve	*1*	*1*
NL:HPRES	Hydrostatic pressure	-	*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
TEMP	Surface average temperature	*2*	*2*
EPEL(X, Y, XY)	Surface elastic strains	*2*	*2*
PRESS	Surface pressure	*2*	*2*
S(X, Y, XY)	Surface stresses (X-axis parallel to line defined by first two nodes which define the face)	*2*	*2*
S(1, 2, 3)	Surface principal stresses	*2*	*2*
SINT	Surface stress intensity	*2*	*2*
SEQV	Surface equivalent stress	*2*	*2*

1. Nonlinear solution (if the element has a nonlinear material)
2. Face printout (if KEYOPT(6) is 1, 2, or 4)
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2 SOLID62 Miscellaneous Structural Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Pt. Solution	EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR, EPSW	*1*	-
Integration Point Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV, EPEL	*2*	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV, EPEL	*3*	-

1. Output at each of eight integration points, if the element has a nonlinear material and KEYOPT(6) = 3
2. Output at each integration point, if KEYOPT(5) = 1
3. Output at each node, if KEYOPT(5) = 2

Table 3 SOLID62 Magnetic Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
CENT: X, Y, Z	Global location XC, YC, ZC	Y	Y
TEMP	Input temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
MU(X, Y, Z)	Magnetic permeability	*1*	*1*
H:X, Y, Z	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y, Z	Magnetic flux density components	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
JS:X, Y, Z	Source current density, valid for static analysis only	*1*	*1*
JT(X, Y, Z)	Total current density components	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
FJB(X, Y, Z)	Lorentz magnetic force components	*1*	-
FMX(X, Y, Z)	Maxwell magnetic force components	*1*	-
FVW(X, Y, Z)	Virtual work force components	*1*	*1*
Combined (FJB or FMX) force components	Combined (FJB or FMX) force components	-	*1*

1. The solution value is output only if calculated (based on input data). The element solution is at the centroid.

Table 4 SOLID62 Miscellaneous Magnetic Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	*1*	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1

2. Output at each corner node, if KEYOPT(5) = 2

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

Table 5, "SOLID62 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 5, "SOLID62 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID62 Structural Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 5 SOLID62 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JSX	SMISC	1
JSY	SMISC	2
JSZ	SMISC	3
JS(SUM)	SMISC	4
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
FVWX	NMISC	4
FVWY	NMISC	5
FVWZ	NMISC	6
FVW(SUM)	NMISC	7
JTX	NMISC	12
JTY	NMISC	13
JTZ	NMISC	14
JT(SUM)	NMISC	15

SOLID62 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID62 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The PCG solver does not support SOLID62 elements.
- For models containing materials with different permeabilities, the 3-D nodal-based vector potential formulation (either static or time-dependent) is not recommended. The solution has been found to be incorrect when the normal component of the vector potential is significant at the interface between elements of

different permeability. To obtain the normal component of the vector potential in postprocessing, issue **PLVECT**,A or **PRVECT**,A in a rotated coordinate system [**RSYS**] that orients one of the vector potential components normal to the material interface.

- For static analysis, the VOLT degree of freedom is not used.
- For transient analyses, the following restrictions apply: The VOLT degree of freedom is required in all regions with a specified nonzero resistivity. The VOLT degree of freedom should be set to zero in nonconducting regions where it is not required. For conducting regions (RSVX $\neq$ 0), current loading should be applied as nodal loads (AMPS); current density loading (JS) is not allowed.
- No coupling is introduced for harmonic analysis. The magneto-structural coupling is invoked only for static and transient analyses. No reduced transient analysis capability is available. Structural coupling is introduced automatically in current carrying conductors (either those with an applied current density, JS, or induced current density, JT). Structural coupling is also introduced by specifying a Maxwell surface on the "air" elements adjacent to the structure.

Note

Applying MVDI does not introduce magneto-structural coupling. The coupling is highly nonlinear if large deflection is involved. Ramp load slowly and converge at intermediate time substeps.

- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the stress gradients and field gradients.
- Pyramid elements are best used as filler elements or in meshing transition zones.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

SOLID62 Product Restrictions

There are no product-specific restrictions for this element.

SHELL63

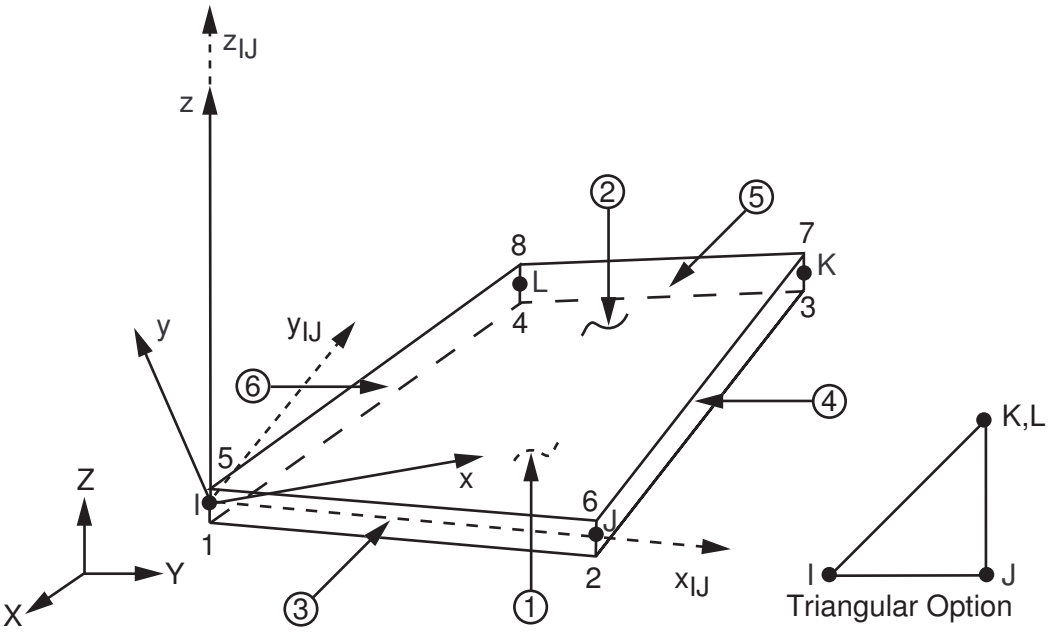
Elastic Shell

MP ME ST PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

SHELL63 Element Description

SHELL63 has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses. See SHELL63 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Similar elements are SHELL43 and SHELL181 (plastic capability), and SHELL281 (midside node capability). The **ETCHG** command converts SHELL57 and SHELL157 elements to SHELL63.

Figure 1 SHELL63 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL63 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL63 Geometry"*. The element is defined by four nodes, four thicknesses, an elastic foundation stiffness, and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element x-axis may be rotated by an angle THETA (in degrees).

The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

The elastic foundation stiffness (EFS) is defined as the pressure required to produce a unit normal deflection of the foundation. The elastic foundation capability is bypassed if EFS is less than, or equal to, zero.

For certain nonhomogeneous or sandwich shell applications, the following real constants are provided: RMI is the ratio of the bending moment of inertia to be used to that calculated from the input thicknesses. RMI defaults to 1.0. CTOP and CBOT are the distances from the middle surface to the extreme fibers to be used for stress evaluations. Both CTOP and CBOT are positive, assuming that the middle surface is between the fibers used for stress evaluation. If not input, stresses are based on the input thicknesses. ADMSUA is the added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL63 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length. The lateral pressure loading may be an equivalent (lumped) element load applied at the nodes (KEYOPT(6) = 0) or distributed over the face of the element (KEYOPT(6) = 2). The equivalent element load produces more accurate stress results with flat elements representing a curved surface or elements supported on an elastic foundation since certain fictitious bending stresses are eliminated.

Temperatures may be input as element body loads at the "corner" locations (1-8) shown in *Figure 1, "SHELL63 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T1 is used for T1, T2, T3, and T4, while T2 (as input) is used for T5, T6, T7, and T8. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(1) is available for neglecting the membrane stiffness or the bending stiffness, if desired. A reduced out-of-plane mass matrix is also used when the bending stiffness is neglected.

KEYOPT(2) is used to activate the consistent tangent stiffness matrix (that is, a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) in large deflection analyses [**NLGEOM**,ON]. You can often obtain more rapid convergence in a geometrically nonlinear analysis, such as a nonlinear buckling or postbuckling analysis, by activating this option. However, you should not use this option if you are using the element to simulate a rigid link or a group of coupled nodes. The resulting abrupt changes in stiffness within the structure make the consistent tangent stiffness matrix unsuitable for such applications.

KEYOPT(3) allows you to include (KEYOPT(3) = 0 or 2) or suppress (KEYOPT(3) = 1) extra displacement shapes. It also allows you to choose the type of in-plane rotational stiffness used:

- KEYOPT(3) = 0 or 1 activates a spring-type in-plane rotational stiffness about the element z-axis
- KEYOPT(3) = 2 activates a more realistic in-plane rotational stiffness (Allman rotational stiffness - the program uses default penalty parameter values of d_1 = 1.0E-6 and d_2 = 1.0E-3).

Using the Allman stiffness will often enhance convergence behavior in large deflection (finite rotation) analyses of planar shell structures (that is, flat shells or flat regions of shells).

KEYOPT(7) allows a reduced mass matrix formulation (rotational degrees of freedom terms deleted). This option is useful for improved bending stresses in thin members under mass loading.

KEYOPT(8) allows a reduced stress stiffness matrix (rotational degrees of freedom deleted). This option can be useful for calculating improved mode shapes and a more accurate load factor in linear buckling analyses of certain curved shell structures.

KEYOPT(11) = 2 is used to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains

with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

A summary of the element input is given in *SHELL63 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL63 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

TK(I), TK(J), TK(K), TK(L), EFS, THETA,
RMI, CTOP, CBOT, (Blank), (Blank), (Blank),
(Blank), (Blank), (Blank), (Blank), (Blank), (Blank),
ADMSUA
See *Table 1, "SHELL63 Real Constants"* for a description of the real constants

Material Properties
EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, DAMP

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --
T1, T2, T3, T4, T5, T6, T7, T8

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness

1 --
Membrane stiffness only

2 --
Bending stiffness only

KEYOPT(2)
Stress stiffening option:

0 --

Use only the main tangent stiffness matrix when **NLGEOM** is ON. (Stress stiffening effects used in linear buckling or other linear prestressed analyses must be activated separately with **PSTRES**,ON.)

1 --

Use the consistent tangent stiffness matrix (that is, a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) when **NLGEOM** is ON and when KEYOPT(1) = 0. (**SSTIF**,ON will be ignored for this element when KEYOPT(2) = 1 is activated.) Note that if **SOLCONTROL** is ON and **NLGEOM** is ON, KEYOPT(2) is automatically set to 1; that is, the consistent tangent will be used.

2 --

Use to turn off consistent tangent stiffness matrix (i.e., a matrix composed of the main tangent stiffness matrix plus the consistent stress stiffness matrix) when **SOLCONTROL** is ON. Sometimes it is necessary to turn off the consistent tangent stiffness matrix if the element is used to simulate rigid bodies by using a very large real constant number . KEYOPT(2) = 2 is the same as KEYOPT(2) = 0, however, KEYOPT(2) = 0 is controlled by **SOLCONTROL**, ON or OFF, while KEYOPT(2) = 2 is independent of **SOLCONTROL**.

KEYOPT(3)

Extra displacement shapes:

0 --

Include extra displacement shapes, and use spring-type in-plane rotational stiffness about the element z-axis (the program automatically adds a small stiffness to prevent numerical instability for non-warped elements if KEYOPT(1) = 0).

Note

For models with large rotation about the in-plane direction, KEYOPT(3) = 0 results in some transfer of moment directly to ground.

1 --

Suppress extra displacement shapes, and use spring-type in-plane rotational stiffness about the element z-axis (the program automatically adds a small stiffness to prevent numerical instability for non-warped elements if KEYOPT(1) = 0).

2 --

Include extra displacement shapes, and use the Allman in-plane rotational stiffness about the element z-axis). See the *Theory Reference for ANSYS and ANSYS Workbench*.

KEYOPT(5)

Extra stress output:

0 --

Basic element printout

2 --

Nodal stress printout

KEYOPT(6)

Pressure loading:

0 --

Reduced pressure loading (must be used if KEYOPT(1) = 1)

2 --

Consistent pressure loading

KEYOPT(7)

Mass matrix:

0 --
Consistent mass matrix

1 --
Reduced mass matrix

KEYOPT(8)
Stress stiffness matrix:

0 --
"Nearly" consistent stress stiffness matrix (default)

1 --
Reduced stress stiffness matrix

KEYOPT(9)
Element coordinate system defined:

0 --
No user subroutine to define element coordinate system

4 --
Element x-axis located by user subroutine USERAN

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(11)
Specify data storage:

0 --
Store data for TOP and BOTTOM surfaces only

2 --
Store data for TOP, BOTTOM, and MID surfaces

Table 1 SHELL63 Real Constants

No.	Name	Description
1	TK(I)	Shell thickness at node I
2	TK(J)	Shell thickness at node J
3	TK(K)	Shell thickness at node K
4	TK(L)	Shell thickness at node L
5	EFS	Elastic foundation stiffness
6	THETA	Element X-axis rotation
7	RMI	Bending moment of inertia ratio
8	CTOP	Distance from mid surface to top
9	CBOT	Distance from mid surface to bottom
10, ..., 18	(Blank)	- -
19	ADMSUA	Added mass/unit area

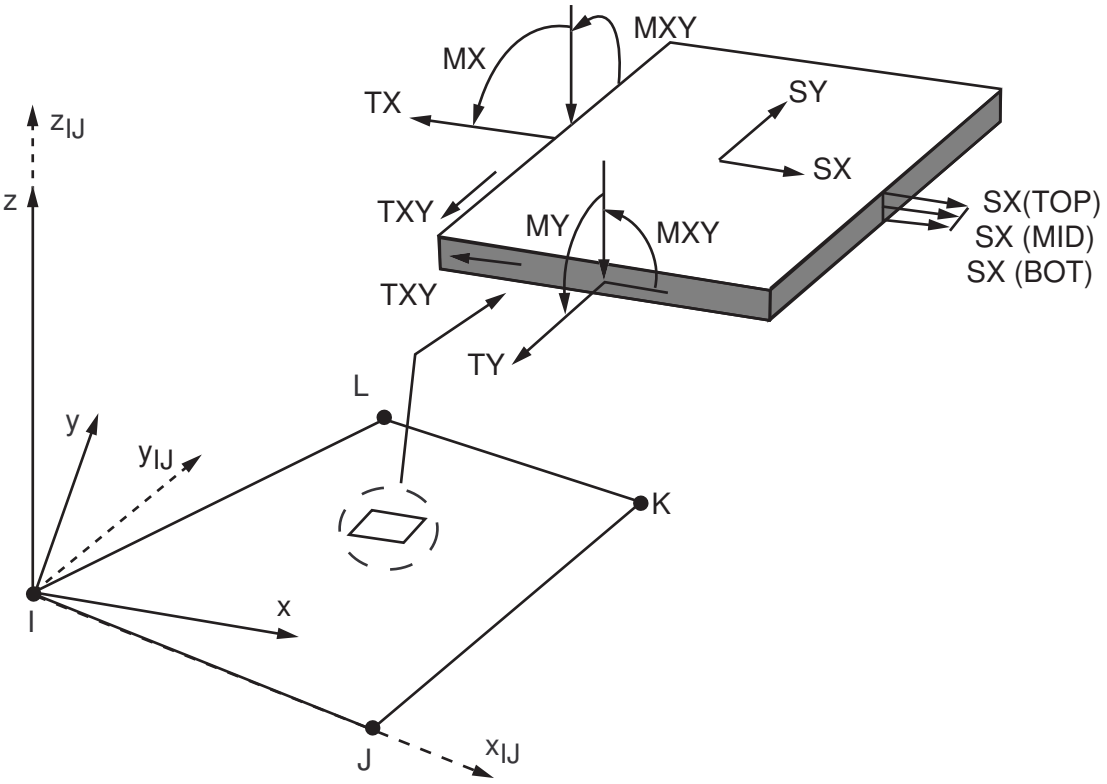
SHELL63 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL63 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL63 Stress Output"*. Printout includes the moments about the x face (MX), the moments about the y face (MY), and the twisting moment (MXY). The moments are calculated per unit length in the element coordinate system. The element stress directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL63 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL63 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y

Name	Definition	O	R
MAT	Material number	Y	Y
AREA	AREA	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T1, T2, T3, T4, T5, T6, T7, T8	Y	Y
T(X, Y, XY)	In-plane element X, Y, and XY forces	Y	Y
M(X, Y, XY)	Element X, Y, and XY moments	Y	Y
FOUND.PRESS	Foundation pressure (if nonzero)	Y	-
LOC	Top, middle, or bottom	Y	Y
S:X, Y, Z, XY	Combined membrane and bending stresses	Y	Y
S:1, 2, 3	Principal stress	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Average elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain [*2*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain [*2*]	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 3 SHELL63 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Stress Solution	TEMP, S(X, Y, Z, XY), SINT, SEQV	*1*	-

1. Output at each node, if KEYOPT(5) = 2, repeats each location

Table 4, "SHELL63 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "SHELL63 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SHELL63 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,K,L
: sequence number for data at nodes I,J,K,L

Table 4 SHELL63 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
TX	SMISC	1	-	-	-	-
TY	SMISC	2	-	-	-	-
TXY	SMISC	3	-	-	-	-
MX	SMISC	4	-	-	-	-
MY	SMISC	5	-	-	-	-
MXY	SMISC	6	-	-	-	-
P1	SMISC	-	9	10	11	12
P2	SMISC	-	13	14	15	16
P3	SMISC	-	18	17	-	-
P4	SMISC	-	-	20	19	-
P5	SMISC	-	-	-	22	21
P6	SMISC	-	23	-	-	24
Top						
S:1	NMISC	-	1	6	11	16
S:2	NMISC	-	2	7	12	17
S:3	NMISC	-	3	8	13	18
S:INT	NMISC	-	4	9	14	19
S:EQV	NMISC	-	5	10	15	20
Bot						
S:1	NMISC	-	21	26	31	36
S:2	NMISC	-	22	27	32	37
S:3	NMISC	-	23	28	33	38
S:INT	NMISC	-	24	29	34	39
S:EQV	NMISC	-	25	30	35	40

SHELL63 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness and vary bilinearly over the shell surface.
- An assemblage of flat shell elements can produce a good approximation of a curved shell surface provided that each flat element does not extend over more than a 15° arc. If an elastic foundation stiffness is input, one-fourth of the total is applied at each node. Shear deflection is not included in this thin-shell element.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. The extra shapes are automatically deleted for triangular elements so that the membrane stiffness reduces to a constant strain formulation. For large deflection analyses, if KEYOPT(1) = 1 (membrane stiffness only), the element *must* be triangular.
- For KEYOPT(1) = 0 or 2, the four nodes defining the element should lie as close as possible to a flat plane (for maximum accuracy), but a moderate amount of warping is permitted. For KEYOPT(1) = 1, the warping

limit is very restrictive. In either case, an excessively warped element may produce a warning or error message. In the case of warping errors, triangular elements should be used (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). Shell element warping tests are described in detail in tables of Applicability of Warping Tests and Warping Factor Limits in the *Theory Reference for ANSYS and ANSYS Workbench*.

- If the lumped mass matrix formulation is specified [**LUMPM**,ON], the effect of the implied offsets on the mass matrix is ignored for warped SHELL63 elements.

SHELL63 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special features allowed are stress stiffening and large deflection.
- KEYOPT(2) can only be set to 0 (default).
- KEYOPT(9) can only be set to 0 (default).

SOLID65

3-D Reinforced Concrete Solid

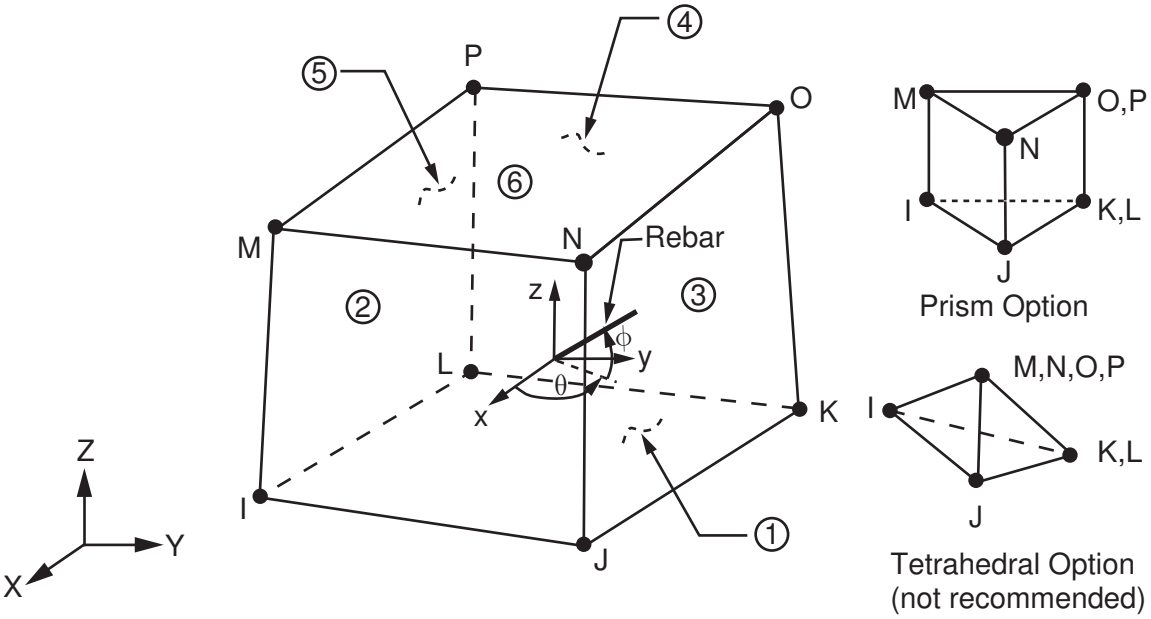
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SOLID65 Element Description

SOLID65 is used for the 3-D modeling of solids with or without reinforcing bars (rebar). The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. Other cases for which the element is also applicable would be reinforced composites (such as fiberglass), and geological materials (such as rock). The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. Up to three different rebar specifications may be defined.

The concrete element is similar to the SOLID45 (3-D Structural Solid) element with the addition of special cracking and crushing capabilities. The most important aspect of this element is the treatment of nonlinear material properties. The concrete is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep. The rebar are capable of tension and compression, but not shear. They are also capable of plastic deformation and creep. See SOLID65 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID65 Geometry



SOLID65 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID65 Geometry"*. The element is defined by eight nodes and the isotropic material properties. The element has one solid material and up to three rebar materials. Use the **MAT** command to input the concrete material properties. Rebar specifications, which are input as real constants, include the material number (MAT), the volume ratio (VR), and the orientation angles (THETA, PHI). The rebar orientations can be graphically verified with the **/ESHAPE** command.

The volume ratio is defined as the rebar volume divided by the total element volume. The orientation is defined by two angles (in degrees) from the element coordinate system. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. A rebar material number of zero or equal to the element material number removes that rebar capability.

Additional concrete material data, such as the shear transfer coefficients, tensile stresses, and compressive stresses are input in the data table, for convenience, as described in *Table 1, "SOLID65 Concrete Material Data"*. Typical shear transfer coefficients range from 0.0 to 1.0, with 0.0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer). This specification may be made for both the closed and open crack. When the element is cracked or crushed, a small amount of stiffness is added to the element for numerical stability. The stiffness multiplier CSTIF is used across a cracked face or for a crushed element, and defaults to 1.0E-6.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID65 Geometry"*. Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

Use the **BETAD** command to supply the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to supply the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the rebar, it is used instead of either the global or element value.

KEYOPT(1) is used to include or suppress the extra displacement shapes. KEYOPT(5) and KEYOPT(6) provide various element printout options (see *Section 2.2.2: Element Solution*).

The stress relaxation associated with KEYOPT(7) = 1 is used only to help accelerate convergence of the calculations when cracking is imminent. (A multiplier for the amount of tensile stress relaxation can be input as constant C9 in the data table; see *Table 1, "SOLID65 Concrete Material Data"*) The relaxation does not represent a revised stress-strain relationship for post-cracking behavior. After the solution converges to the cracked state, the modulus normal to the crack face is set to zero. Thus, the stiffness is zero normal to the crack face. See the *Theory Reference for ANSYS and ANSYS Workbench* for details.

The program warns when each unreinforced element crushes at all integration points. If this warning is unwanted, it can be suppressed with KEYOPT(8) = 1.

If solution convergence is a problem, it is recommended to set KEYOPT(3) = 2 and apply the load in very small load increments.

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID65 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID65 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants

MAT1, VR1, THETA1, PHI1, MAT2, VR2,
THETA2, PHI2, MAT3, VR3, THETA3, PHI3, CSTIF

(where MAT*n* is material number, VR*n* is volume ratio, and THETA*n* and PHI*n* are orientation angles for up to 3 rebar materials)

Material Properties

EX, ALPX (or CTEX *or* THSX), PRXY or NUXY, DENS (for concrete)
EX, ALPX (or CTEX *or* THSX), DENS (for each rebar)
Supply DAMP only once for the element (use **MAT** command to assign material property set). REFT may be supplied once for the element, or may be assigned on a per rebar basis. See the discussion in *SOLID65 Input Data* for more details.

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Fluences --
FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Concrete (CONC)
Cracking
Crushing
Large deflection
Large strain
Stress stiffening
Birth and death
Adaptive descent

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(3)
Behavior of totally crushed unreinforced elements:

0 --
Base

1 --
Suppress mass and applied loads, and warning message (see KEYOPT(8))

2 --
Features of 1 and apply consistent Newton-Raphson load vector.

KEYOPT(5)
Concrete linear solution output:

0 --
Print concrete linear solution only at centroid

1 --
Repeat solution at each integration point

2 --
Nodal stress printout

KEYOPT(6)
Concrete nonlinear solution output:

0 --
Print concrete nonlinear solution only at centroid

3 --
Print solution also at each integration point

KEYOPT(7)
Stress relaxation after cracking:

0 --
No tensile stress relaxation after cracking

1 --
Include tensile stress relaxation after cracking to help convergence

KEYOPT(8)
Warning message for totally crushed unreinforced element:

0 --
Print the warning

1 --
Suppress the warning

SOLID65 Concrete Information

The data listed in *Table 1, "SOLID65 Concrete Material Data"* is entered in the data table with the **TB** commands. Data not input are assumed to be zero, except for defaults described below. The constant table is started by using the **TB**command (with *Lab* = CONCR). Up to eight constants may be defined with the **TBDATA** commands following a temperature definition on the **TBTEMP** command. Up to six temperatures (*NTEMP* = 6 maximum on the **TB** command) may be defined with the **TBTEMP** commands. The constants (C1-C9) entered on the **TBDATA** commands (6 per command), after each **TBTEMP** command, are:

Table 1 SOLID65 Concrete Material Data

Constant	Meaning
1	Shear transfer coefficients for an open crack.
2	Shear transfer coefficients for a closed crack.
3	Uniaxial tensile cracking stress.
4	Uniaxial crushing stress (positive).
5	Biaxial crushing stress (positive).
6	Ambient hydrostatic stress state for use with constants 7 and 8.
7	Biaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
8	Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
9	Stiffness multiplier for cracked tensile condition, used if KEYOPT(7) = 1 (defaults to 0.6).

Absence of the data table removes the cracking and crushing capability. A value of -1 for constant 3 or 4 also removes the cracking or crushing capability, respectively. If constants 1-4 are input and constants 5-8 are omitted, the latter constants default as discussed in the *Theory Reference for ANSYS and ANSYS Workbench*. If any one of Constants 5-8 are input, there are no defaults and all 8 constants must be input.

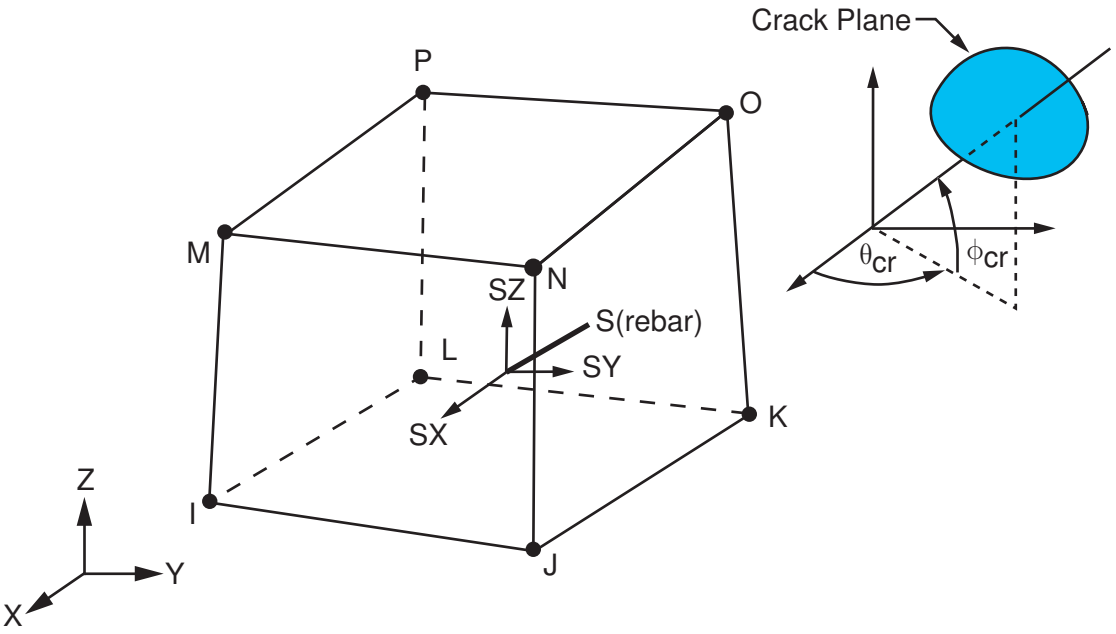
SOLID65 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SOLID65 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID65 Stress Output"*. The element stress directions are parallel to the element coordinate system. Nonlinear material printout appears only if nonlinear properties are specified. Rebar printout appears only for the rebar defined. If cracking or crushing is possible, printout for the concrete is also at the integration points, since cracking or crushing may occur at any integration point. The **PLCRACK** command can be used in POST1 to display the status of the integration points. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID65 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SOLID65 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
NREINF	Number of rebar	Y	-
VOLU:	Volume	Y	Y
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)	Y	Y
XC, YC, ZC	Location where results are reported	Y	*6*
S:X, Y, Z, XY, YZ, XZ	Stresses	*1*	*1*
S:1, 2, 3	Principal stresses	*1*	*1*
S:INT	Stress intensity	*1*	*1*
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal elastic strains	*1*	-
EPEL:EQV	Equivalent elastic strains [*7*]	*1*	*1*

Name	Definition	O	R
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strains [*7*]	*1*	*1*
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	*4*	*4*
EPPL:EQV	Equivalent plastic strains [*7*]	*4*	*4*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains	*4*	*4*
EPCR:EQV	Equivalent creep strains [*7*]	*4*	*4*
NL:EPEQ	Average equivalent plastic strain	*4*	*4*
NL:SRAT	Ratio of trial stress to stress on yield surface	*4*	*4*
NL:SEPL	Average equivalent stress from stress-strain curve	*4*	*4*
NL:HPRES	Hydrostatic pressure	-	*4*
THETCR, PHICR	THETA and PHI angle orientations of the normal to the crack plane	*1*	*1*
STATUS	Element status	*2*	*2*
IRF	Rebar number	*3*	-
MAT	Material number	*3*	-
VR	Volume ratio	*3*	-
THETA	Angle of orientation in X-Y plane	*3*	-
PHI	Angle of orientation out of X-Y plane	*3*	-
EPEL	Uniaxial elastic strain	*3*	-
S	Uniaxial stress	*3*	-
EPEL	Average uniaxial elastic strain	*5*	*5*
EPPL	Average uniaxial plastic strain	*5*	*5*
SEPL	Average equivalent stress from stress-strain curve	*5*	*5*
EPCR	Average uniaxial creep strain	*5*	*5*

1. Concrete solution item (output for each integration point (if KEYOPT(5) = 1) and the centroid)
2. The element status table (*Table 4, "SOLID65 Element Status Table"*) uses the following terms:
   - Crushed - solid is crushed.
   - Open - solid is cracked and the crack is open.
   - Closed - solid is cracked but the crack is closed.
   - Neither - solid is neither crushed nor cracked.
3. Rebar solution item repeats for each rebar
4. Concrete nonlinear integration point solution (if KEYOPT(6) = 3 and the element has a nonlinear material)
5. Rebar nonlinear integration point solution (if KEYOPT(6) = 3 and the rebar has a nonlinear material)
6. Available only at centroid as a ***GET** item.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 3 SOLID65 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV	*1*	-

1. Output at each node, if KEYOPT(5) = 2

Table 4 SOLID65 Element Status Table

Status	Status in Direction 1	Status in Direction 2	Status in Direction 3
1	Crushed	Crushed	Crushed
2	Open	Neither	Neither
3	Closed	Neither	Neither
4	Open	Open	Neither
5	Open	Open	Open
6	Closed	Open	Open
7	Closed	Open	Neither
8	Open	Closed	Open
9	Closed	Closed	Open
10	Open	Closed	Neither
11	Open	Open	Closed
12	Closed	Open	Closed
13	Closed	Closed	Neither
14	Open	Closed	Closed
15	Closed	Closed	Closed
16	Neither	Neither	Neither

Table 5, "SOLID65 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 5, "SOLID65 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SOLID65 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,...,P
: sequence number for data at nodes I,J,...,P

IP
: sequence number for Integration Point solution items

Table 5 SOLID65 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	Rebar 1	Rebar 2	Rebar 3
EPEL	SMISC	1	3	5
SIG	SMISC	2	4	6
EPPL	NMISC	41	45	49
EPCR	NMISC	42	46	50
SEPL	NMISC	43	47	51
SRAT	NMISC	44	48	52

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	8	7	10	9	-	-	-	-
P2	SMISC	11	12	-	-	14	13	-	-
P3	SMISC	-	15	16	-	-	18	17	-
P4	SMISC	-	-	19	20	-	-	22	21
P5	SMISC	24	-	-	23	25	-	-	26
P6	SMISC	-	-	-	-	27	28	29	30
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40
FLUEN	NMISC	109	110	111	112	113	114	115	116

	Output Quantity Name	ETABLE and ESOL Command Input								
		Item	Integration Point							
			1	2	3	4	5	6	7	8
	STATUS	NMISC	53	60	67	74	81	88	95	102
Dir 1	THETCR	NMISC	54	61	68	75	82	89	96	103
	PHICR	NMISC	55	62	69	76	83	90	97	104
Dir 2	THETCR	NMISC	56	63	70	77	84	91	98	105
	PHICR	NMISC	57	64	71	78	85	92	99	106
Dir 3	THETCR	NMISC	58	65	72	79	86	93	100	107
	PHICR	NMISC	59	66	73	80	87	94	101	108

SOLID65 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1, "SOLID65 Geometry"* or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.
- Whenever the rebar capability of the element is used, the rebar are assumed to be "smeared" throughout the element. The sum of the volume ratios for all rebar must not be greater than 1.0.
- The element is nonlinear and requires an iterative solution.
- When both cracking and crushing are used together, care must be taken to apply the load slowly to prevent possible fictitious crushing of the concrete before proper load transfer can occur through a closed crack. This usually happens when excessive cracking strains are coupled to the orthogonal uncracked directions through Poisson's effect. Also, at those integration points where crushing has occurred, the output plastic and creep strains are from the previous converged substep. Furthermore, when cracking has occurred,

the elastic strain output includes the cracking strain. The lost shear resistance of cracked and/or crushed elements cannot be transferred to the rebar, which have no shear stiffness.

- The following two options are not recommended if cracking or crushing nonlinearities are present:
  - Stress-stiffening effects.
  - Large strain and large deflection. Results may not converge or may be incorrect, especially if significantly large rotation is involved.

SOLID65 Product Restrictions

There are no product-specific restrictions for this element.

PLANE67

2-D Coupled Thermal-Electric Solid

MP ME <> PR PRN <> <> <> EM <> <> PP <>
Product Restrictions

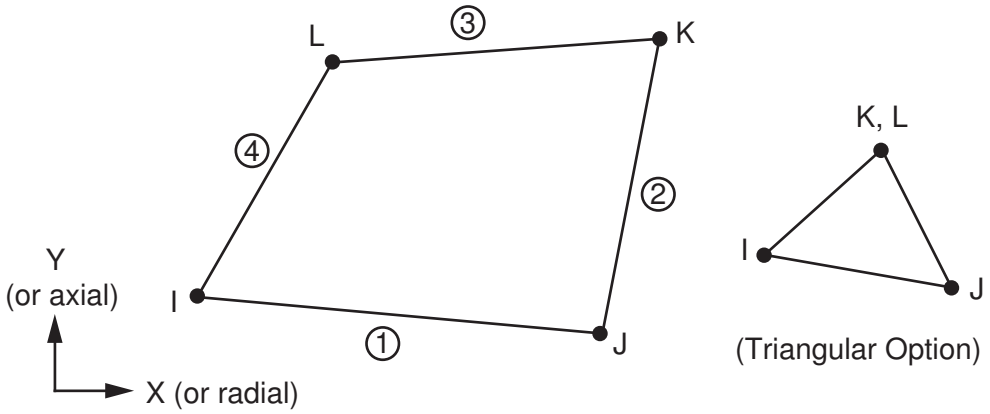
PLANE67 Element Description

PLANE67 has thermal and electrical conduction capability. Joule heat generated by the current flow is also included in the heat balance. The element has four nodes with two degrees of freedom, temperature and voltage, at each node.

The element is applicable to a 2-D (plane or axisymmetric), steady-state or transient thermal analysis, although no transient electrical capacitance or inductance effects are included in the element. The element requires an iterative solution to include the Joule heating effect in the thermal solution. See PLANE67 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If no electrical effects are present, the 2-D thermal solid (PLANE55) may be used.

If the model containing the thermal-electrical element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE42). You also can use the thermal-electric shell element, SHELL157, in conjunction with PLANE67.

Figure 1 PLANE67 Geometry



PLANE67 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE67 Geometry"*. The element is defined by four nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The specific heat and density may be assigned any values for steady-state solutions. The electrical material property, RSV_, is the resistivity of the material. The resistivity, like any other material property, may be input as a function of temperature. Properties not input default as described in *Section 2.4: Linear Material Properties*.

The word VOLT should be input for the *Lab* variable on the **D** command and the voltage input for the value. The word AMPS should be input for the *Lab* variable on the **F** command and the current into the node input for the value.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1,"PLANE67 Geometry"*.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). This rate is in addition to the Joule heat generated by the current flow.

A summary of the element input is given in *PLANE67 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE67 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP, VOLT

Real Constants
None

Material Properties
KXX, KYY, DENS, C, ENTH, RSVX, RSVY

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Special Features

Requires an iterative solution for electrical-thermal coupling
Birth and death

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(4)
Evaluation of film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS - TB|

PLANE67 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE67 Element Output Definitions"*

Heat flow out of the element is considered to be positive. The element output directions are parallel to the element coordinate system. The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used in the temperature distribution calculated for the next substep. The volume printout, like other quantities, is on a full 360° basis for axisymmetric elements. *Section 2.2: Solution Output* gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE67 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	Y
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
EF:X, Y, SUM	Component electric fields and vector sum	Y	Y
JS:X, Y	Component current densities	Y	Y
JSSUM	Component current vector sum	Y	-
JHEAT:	Joule heat generation per unit volume	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	*1*
HFILM	Film coefficient at each node of face	*1*	-
TBULK	Bulk temperature at each node of face	*1*	-
TAVG	Average face temperature	*1*	*1*
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-

Name	Definition	O	R
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HFLUX	Heat flux at each node of face	*1*	-

1. If a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "PLANE67 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE67 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE67 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 PLANE67 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE67 Assumptions and Restrictions

- The element must not have a negative or zero area.
- The element must lie in an X-Y plane as shown in *Figure 1, "PLANE67 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as for melting) within a coarse grid.
- If this thermal-electric element is to be replaced by a PLANE42 structural element with surface stresses requested, the thermal-electric element should be oriented so that face IJ and/or face KL is a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

- Current flow and heat flow must be in the same plane. If a current is specified at the same node that a voltage is specified, the current is ignored.
- The electrical and the thermal solutions are coupled through an iterative procedure.
- No conversion is included between electrical heat units and mechanical heat units.
- The resistivity may be divided by a conversion factor, such as 3.415 Btu/Hr per Watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- There is no conversion required when consistent units are used.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

PLANE67 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS Emag

- This element has only electric field capability, and does not have thermal capability.
- The element may only be used in a steady-state electric analysis.
- The only active degree of freedom is VOLT.
- The only allowable material properties are RSVX and RSVY.
- No surface loads or body loads are applicable.
- The birth and death special feature is not allowed.

LINK68

Coupled Thermal-Electric Line

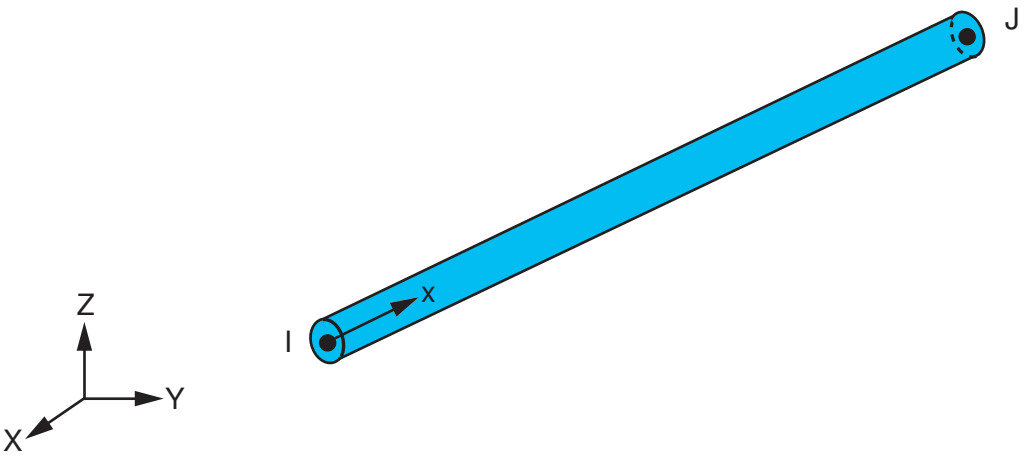
MP ME <> PR PRN <> <> <> EM <> <> PP <>
Product Restrictions

LINK68 Element Description

LINK68 is a uniaxial element in 3-D space with the ability to conduct heat and electrical current between its nodes. Joule heat generated by the current flow is also included in the heat balance. The element has two degrees of freedom, temperature and voltage, at each node. The thermal-electrical line element may be used in a steady-state or transient thermal analysis, although no transient electrical capacitance or inductance effects are included in the element.

The element is linear but requires an iterative solution to include the Joule heating effect in the thermal solution. If no electrical effects are present, the conducting bar element (LINK33) may be used. If the model containing the thermal-electrical element is also to be analyzed structurally, the element should be replaced by an equivalent structural element. See LINK68 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 LINK68 Geometry



LINK68 Input Data

The geometry, node locations, and the coordinate system for this thermal-electrical line element are shown in *Figure 1, "LINK68 Geometry"*. The element is defined by two nodes, the cross-sectional area, and the material properties. In an axisymmetric analysis the area should be input on a full 360° basis. The thermal conductivity and electrical resistivity are in the element longitudinal direction. The specific heat and density may be assigned any values for steady-state solutions.

The electrical material property, RSVX, is the resistivity of the material. The resistance of the element is calculated from RSVX*length/AREA. The resistivity, like any other material property, may be input as a function of temperature. Properties not input default as described in *Section 2.4: Linear Material Properties*.

The word VOLT should be input for the *Lab* variable on the **D** command and the voltage input for the value. The word AMPS should be input for the *Lab* variable on the **F** command and the current into the node input for the value.

Element loads are described in *Section 2.8: Node and Element Loads*. Element body loads may be input as heat generation rates at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I). This rate is in addition to the Joule heat generated by the current flow.

The current being calculated via this element can be directly coupled into a 3-D magnetostatic analysis [**BIOT**].

A summary of the element input is given in *LINK68 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK68 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP, VOLT

Real Constants
AREA - Cross-sectional area

Material Properties
KXX, DENS, C, ENTH, RSVX

Surface Loads
None

Body Loads

Heat Generations --
HG(I), HG(J)

Special Features

Requires an iterative solution for electrical-thermal coupling
Birth and death

KEYOPTS
None

LINK68 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK68 Element Output Definitions"*

The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used to determine the temperature distribution calculated for the next substep. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK68 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
HGEN	Heat generations HG(I), HG(J)	Y	-
TG	Thermal gradient at centroid	Y	Y
TF	Thermal flux at centroid (heat flow/cross-sectional area)	Y	Y
EF	Electric field (voltage gradient)	Y	Y
JS	Current density (voltage flux)	Y	Y
CUR	Current	Y	Y
JHEAT:	Joule heat generation per unit volume	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "LINK68 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "LINK68 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "LINK68 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 LINK68 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TG	NMISC	1
TF	NMISC	2
EF	NMISC	3
JS	NMISC	4
CUR	NMISC	5

LINK68 Assumptions and Restrictions

- Heat and current are assumed to flow only in the element longitudinal direction.
- The element must not have a zero length, that is, nodes I and J may not be coincident.

- A free end of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to adiabatic.
- No conversion is included between electrical heat units and mechanical heat units.
- The resistivity may be divided by a conversion factor, such as 3.415 Btu/Hr per Watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- If a current is specified at the same node that a voltage is specified, the current is ignored.
- The electrical and the thermal solutions are coupled through an iterative procedure.
- There is no conversion required when consistent units are used.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

LINK68 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS Emag 3-D

- This element has only electric field capability, and does not have thermal capability.
- The element may only be used in a steady-state electric analysis.
- The only active degree of freedom is VOLT.
- The only allowable material property is RSVX.
- No body loads are applicable.
- The birth and death special feature is not allowed.

SOLID69

3-D Coupled Thermal-Electric Solid

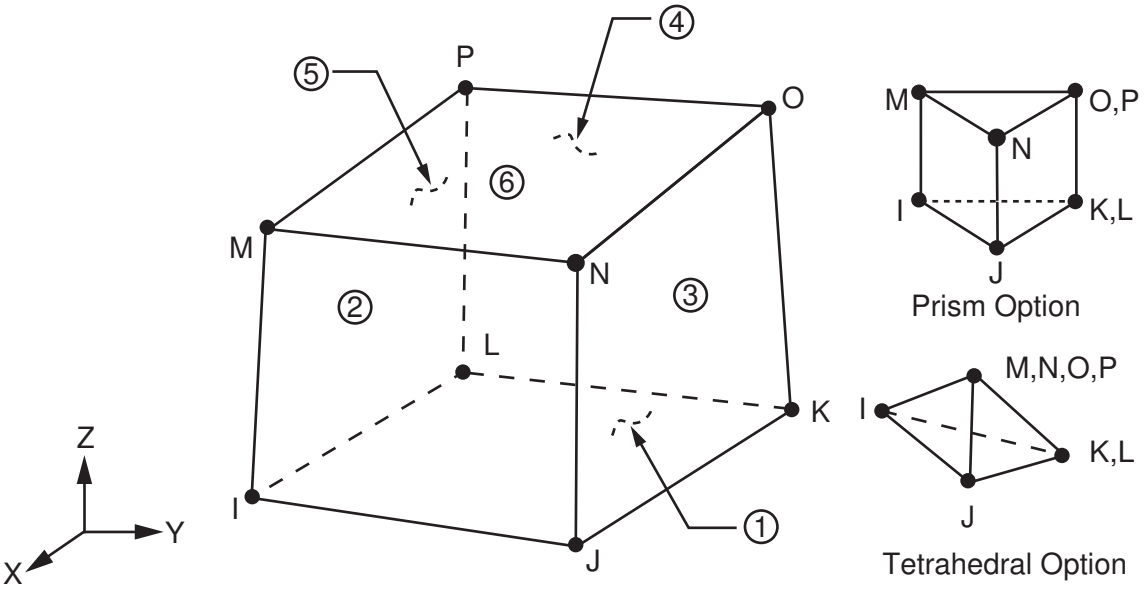
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SOLID69 Element Description

SOLID69 has a 3-D thermal and electrical conduction capability. Joule heat generated by the current flow is also included in the heat balance. The element has eight nodes with two degrees of freedom, temperature and voltage, at each node. The thermal-electric solid element is applicable to a 3-D, steady-state or transient thermal analysis, although no transient electrical capacitance or inductance effects are included in the element. The element requires an iterative solution to include the Joule heating effect in the thermal solution. See SOLID69 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If no electrical effects are present, the 3-D thermal solid (SOLID70) may be used.

If the model containing the thermal-electrical solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SOLID45). Another element related to SOLID69 is SHELL157.

Figure 1 SOLID69 Geometry



SOLID69 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID69 Geometry"*. The element is defined by eight nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The specific heat and density may be assigned any values for steady-state solution. The electrical material property is the resistivity (RSVX, RSVY, RSVZ) of the material. The resistivity, like any other material property, may be input as a function of temperature. Properties not input default as described in *Section 2.4: Linear Material Properties*.

The word VOLT should be input for the *Lab* variable on the **D** command and the voltage input for the value. The word AMPS should be input for the *Lab* variable on the **F** command and the current into the node input for the value.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID69 Geometry"*.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). This rate is in addition to the Joule heat generated by the current flow.

A summary of the element input is given in *SOLID69 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID69 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP, VOLT

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH, RSVX, RSVY, RSVZ

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features

Requires an iterative solution for electrical-thermal coupling
Birth and death

KEYOPT(2)
Evaluation of film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS - TB|

SOLID69 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID69 Element Output Definitions"*

Heat flow out of the element is considered to be positive. The element output directions are parallel to the element coordinate system. The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used in the temperature distribution calculated for the next substep. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID69 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
EF:X, Y, Z, SUM	Component electric fields and vector sum	Y	Y
JS:X, Y, Z	Component current densities	Y	Y
JSSUM	Component current vector sum	Y	-
JHEAT:	Joule heat generation per unit volume	Y	Y
FACE	Face label	*1*	-
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	-
HFILM	Film coefficient at each node of face	*1*	-
TBULK	Bulk temperature at each node of face	*1*	-
TAVG	Average face temperature	*1*	*1*
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HFLUX	Heat flux at each node of face	*1*	-

1. If a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "SOLID69 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID69 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID69 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 SOLID69 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID69 Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "SOLID69 Geometry"*, or may have the upper and lower planes interchanged (for example, plane IJKL may be interchanged with plane MNOP).
- A prism or tetrahedron-shaped element may be formed by defining duplicate node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal-electric element is to be replaced by a SOLID45 structural element with surface stresses requested, the thermal element should be oriented so that face I-J-N-M and/or face K-L-P-O is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- If a current is specified at the same node that a voltage is specified, the current is ignored.
- No conversion is included between electrical heat units and mechanical heat units.
- The resistivity may be divided by a conversion factor, such as 3.415 Btu/Hr per watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- The electrical and the thermal solutions are coupled through an iterative procedure. There is no conversion required when consistent units are used.

- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

SOLID69 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

SOLID70

3-D Thermal Solid

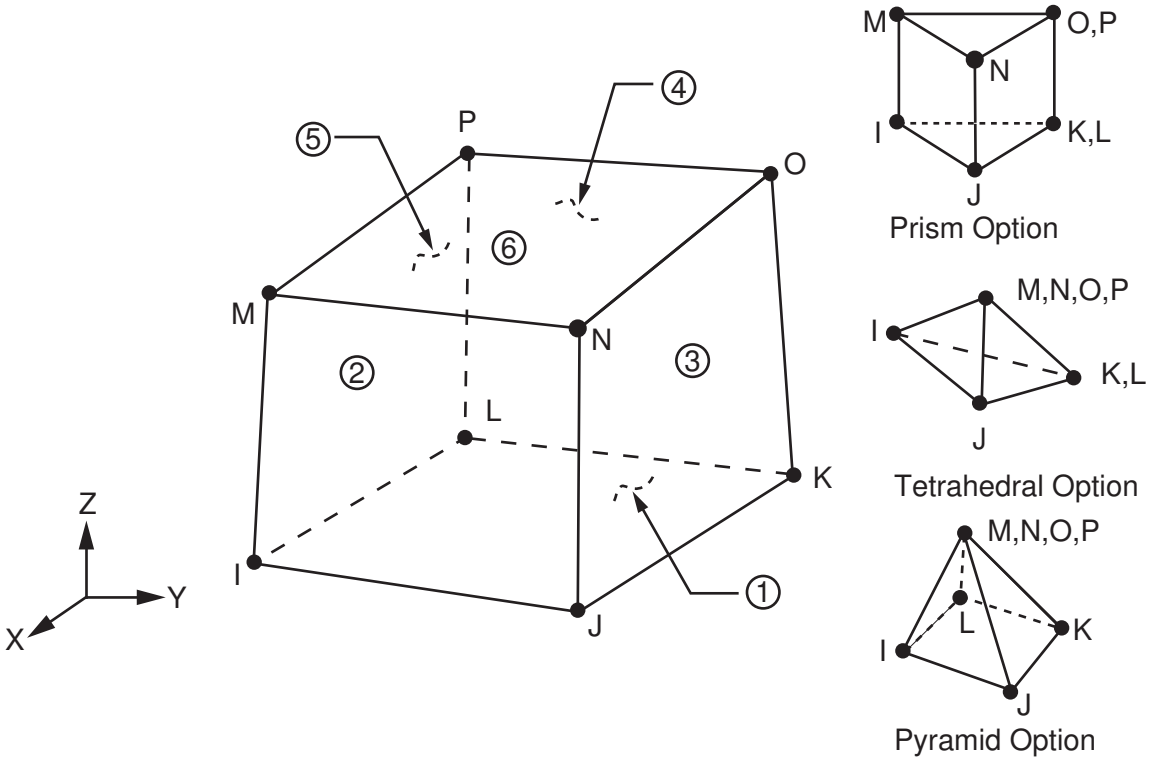
MP ME <> PR PRN DS <> <> <> <> <> PP VT
Product Restrictions

SOLID70 Element Description

SOLID70 has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. The element also can compensate for mass transport heat flow from a constant velocity field. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SOLID45). See SOLID90 for a similar thermal element, with mid-edge node capability.

An option exists that allows the element to model nonlinear steady-state fluid flow through a porous medium. With this option, the thermal parameters are interpreted as analogous fluid flow parameters. For example, the temperature degree of freedom becomes equivalent to a pressure degree of freedom. See SOLID70 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID70 Geometry



SOLID70 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID70 Geometry"*. The element is defined by eight nodes and the orthotropic material properties. A prism-shaped element, a tetrahedral-shaped element, and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID70 Geometry"*. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID70 Geometry"*.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

The nonlinear porous flow option is selected with KEYOPT(7) = 1. For this option, temperature is interpreted as pressure and the absolute permeability of the medium are input as material properties KXX, KYY, and KZZ. Properties DENS and VISC are used for the mass density and viscosity of the fluid. Properties C and MU are used in calculating the coefficients of permeability as described in the *Theory Reference for ANSYS and ANSYS Workbench*. Temperature boundary conditions input with the **D** command are interpreted as pressure boundary conditions, and heat flow boundary conditions input with the **F** command are interpreted as mass flow rate (mass/time).

A mass transport option is available with KEYOPT(8). With this option the velocities VX, VY, and VZ must be input as real constants (in the element coordinate system). Also, temperatures should be specified along the entire inlet boundary to assure a stable solution. With mass transport, you should use specific heat (C) and density (DENS) material properties instead of enthalpy (ENTH).

A summary of the element input is given in *SOLID70 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID70 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Real Constants

Mass transport effects (KEYOPT(8) = 1):
VX - X direction of mass transport velocity
VY - Y direction of mass transport velocity
VZ - Z direction of mass transport velocity

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH, VISC, MU (VISC and MU used only if KEYOPT(7) = 1. Do not use ENTH with KEYOPT(8) = 1).

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(2)
Evaluation of film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature |TS-TB|

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(7)
Nonlinear fluid flow option:

0 --
Standard heat transfer element

1 --
Nonlinear steady-state fluid flow analogy element

Note

Temperature degree of freedom interpreted as pressure.

KEYOPT(8)
Mass transport effects:

0 --
No mass transport effects

1 --
Mass transport with VX, VY, VZ

SOLID70 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID70 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. If KEYOPT(7) = 1, the standard thermal output should be interpreted as the analogous fluid flow output. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID70 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	-
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	-
HFILM	Film coefficient at each node of face	*1*	-
TBULK	Bulk temperature at each node of face	*1*	-
TAVG	Average face temperature	*1*	*1*
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HFLUX	Heat flux at each node of face	*1*	-
PRESSURE GRAD	Total pressure gradient and its X, Y, and Z components	*2*	-
MASS FLUX	Mass flow rate per unit cross-sectional area	*2*	-
FLUID VELOCITY	Total fluid velocity and its X, Y, and Z components	*2*	-

1. Output if a surface load is input
2. Output if KEYOPT(7) = 1
3. Available only at centroid as a ***GET** item.

Table 2, "SOLID70 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID70 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID70 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 SOLID70 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID70 Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "SOLID70 Geometry"* or may have the planes IJKL and MNOP interchanged.
- A prism or tetrahedron shaped element may be formed by defining duplicate node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as for melting) within a coarse grid.
- If the thermal element is to be replaced by a SOLID45 structural element with surface stresses requested, the thermal element should be oriented such that face I-J-N-M and/or face K-L-P-O is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- If KEYOPT(8) > 0, unsymmetric matrices are produced.
- When mass flow is activated (KEYOPT(8)=1), the element Peclet number should be less than 1:

  $Pe = \rho*v*L*Cp/(2*k) < 1.0$

  Where L is an element length scale based on the element geometry. See SOLID70 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

SOLID70 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element does not have the mass transport or fluid flow options. KEYOPT(7) and KEYOPT(8) can only be set to 0 (default).

- The VX, VY, and VZ real constants are not applicable.
- The VISC and MU material properties are not applicable.
- The element does not have the birth and death feature.

MASS71

Thermal Mass

MP ME <> PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

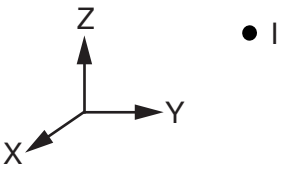
MASS71 Element Description

MASS71 is a point element having one degree of freedom, temperature, at the node. The element may be used in a transient thermal analysis to represent a body having thermal capacitance capability but negligible internal thermal resistance, that is, no significant temperature gradients within the body. The element also has a temperature-dependent heat generation rate capability. The lumped thermal mass element is applicable to a 1-D, 2-D, or 3-D steady-state or transient thermal analysis. See MASS71 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

In a steady-state solution the element acts only as a temperature-dependent heat source or sink. Other elements having special thermal applications are the COMBIN14 and COMBIN40 elements. These elements, which are normally used in structural models, may be used for thermally analogous situations.

If the model containing the thermal mass element is also to be analyzed structurally, the thermal element should be replaced by an equivalent structural element (such as MASS21)

Figure 1 MASS71 Geometry



MASS71 Input Data

The lumped thermal mass element is defined by one node (as shown in *Figure 1, "MASS71 Geometry"*) and a thermal capacitance (Heat/Degree). When used with axisymmetric elements, the thermal capacitance should be input on a full 360° basis. The thermal capacitance (CON1) may be input as a real constant or calculated (KEYOPT(3)) from the real constant volume (CON1) and either the DENS and C or ENTH material properties. KEYOPT(3) determines whether CON1 is interpreted as volume or thermal capacitance.

The heat generation is applied directly as a nodal load and is not first multiplied by the volume. Thus, if KEYOPT(3) = 0 (that is, when using the specific heat matrix), the heat generation rate must be adjusted to account for the volume. For an axisymmetric analysis the heat generation rate should be input on a full 360° basis. A temperature-dependent heat generation rate of the following polynomial form may be input:

$$\dddot{q}(T) = A_1 + A_2T + A_3T^{A_4} + A_5T^{A_6}$$

where T is the absolute temperature from the previous substep. The constants, A_1 through A_6, should be entered as real constants. If any of the constants A_2 through A_6 are nonzero, KEYOPT(4) must be set to 1. Also, if temperatures are not absolute, the offset conversion [**TOFFST**] must be specified.

Alternately, the heat generation expression may be defined as a temperature-dependent material property (QRATE) with the **MP** commands. QRATE can be input as numerical values or as tabular inputs evaluated as a function of temperature, time, and location.

A summary of the element input is given in *MASS71 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. See *Section 2.12: Axisymmetric Elements* for more details.

MASS71 Input Summary

Nodes
I

Degrees of Freedom
TEMP

Real Constants

CON1, A1, A2, A3, A4, A5
A6
See *Table 1, "MASS71 Real Constants"* for a description of the real constants

Material Properties
QRATE, DENS, C, ENTH if KEYOPT(3) = 0, or QRATE if KEYOPT(3) = 1

Surface Loads
None

Body Loads
None (heat generation rates may be defined as a function of temperature by using real constants A1, A2, ... or by the QRATE material property definition.)

Special Features

Nonlinear if heat generation is defined as a function of temperature
Birth and death

KEYOPT(3)
Interpretation of real constant CON1:

0 --
Interpret CON1 as volume (with either DENS and C or ENTH supplied as material properties)

1 --
Interpret CON1 as thermal capacitance (DENS*C*volume)

KEYOPT(4)
Temperature dependent heat generation:

0 --
No temperature-dependent heat generation (required if all real constants A2-A6 are zero)

1 --
Include temperature-dependent heat generation (required if any real constants A2-A6 are nonzero)

Table 1 MASS71 Real Constants

No.	Name	Description
1	CON1	Volume or thermal capacitance (see KEYOPT(3))
2	A1	Constant for temperature function
3	A2	Constant for temperature function
4	A3	Constant for temperature function
5	A4	Constant for temperature function
6	A5	Constant for temperature function

No.	Name	Description
7	A6	Constant for temperature function

MASS71 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 2, "MASS71 Element Output Definitions"*

The heat generation rate is in units of Heat/Time and is positive into the node. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 MASS71 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODE	Node I	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
TEMP	Element (node) temperature	Y	Y
HEAT RATE	Heat generation rate into node	Y	Y

1. Available only at centroid as a ***GET** item.

Table 3, "MASS71 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "MASS71 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "MASS71 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 MASS71 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TEMP	SMISC	2

MASS71 Assumptions and Restrictions

- When using the element with a temperature-dependent heat generation rate in a steady-state solution, an iterative solution is required.
- The heat generation is calculated at the uniform temperature for the first substep.

MASS71 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

PLANE75

Axisymmetric-Harmonic 4-Node Thermal Solid

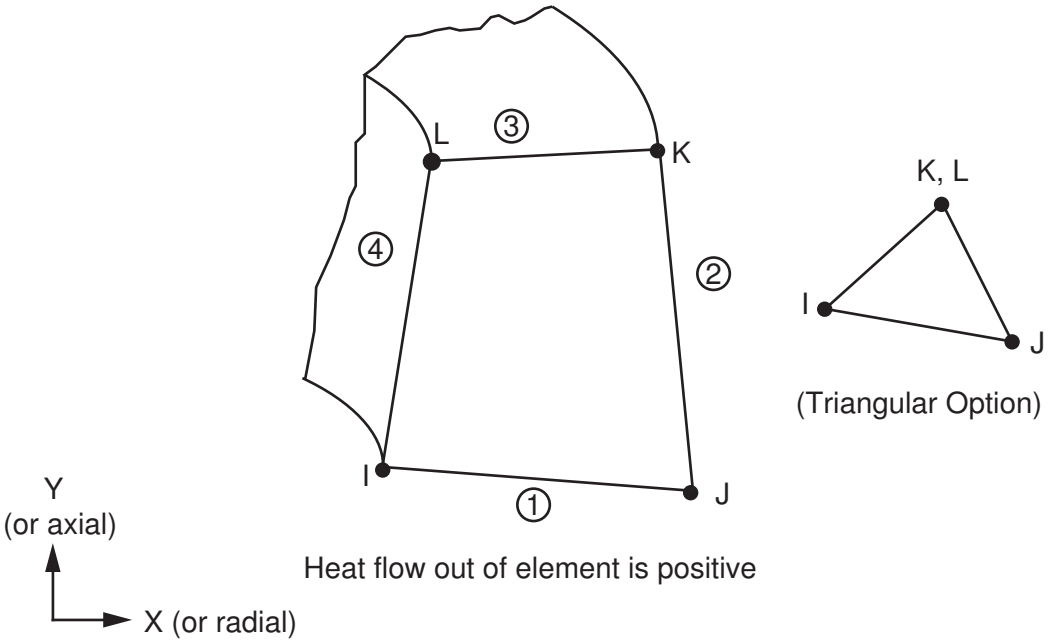
MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

PLANE75 Element Description

PLANE75 is used as an axisymmetric ring element with a 3-D thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node. The element is a generalization of the axisymmetric version of PLANE55 in that it allows nonaxisymmetric loading. Various loading cases are described in *Section 2.14: Shear Deflection*.

The element is applicable to a 2-D, axisymmetric, steady-state or transient thermal analysis. See PLANE75 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing the element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as PLANE25). A similar thermal element, with midside node capability is PLANE78.

Figure 1 PLANE75 Geometry



PLANE75 Input Data

The geometry, node locations, and the coordinate system for this axisymmetric thermal solid element are shown in *Figure 1, "PLANE75 Geometry"*. The data input is essentially the same as for PLANE55 and is described in *PLANE55 Input Data*. The element input data also includes the number of harmonic waves (MODE) and the symmetry condition (ISYM) on the **MODE** command. If MODE = 0 and ISYM = 1, the element behaves similar to the axisymmetric case of PLANE55. The MODE and ISYM parameters describe the type of temperature distribution and are discussed in *Section 2.14: Shear Deflection*.

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying convections or heat fluxes (but not both) may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE75 Geometry"*. Harmonically varying heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input and all others are unspecified, they default to HG(I).

A summary of the element input is given in *PLANE75 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PLANE75 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Heat Fluxes --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Mode Number
Input mode number on **MODE** command

Loading Condition
Input for *ISYM* in **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

Special Features
Birth and death

KEYOPTS
None

PLANE75 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE75 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. The face area and the heat flow rate are on a full 360° basis. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE75 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*3*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
MODE	Number of waves in loading	Y	-
TG:X, Y, SUM, Z	Thermal gradient components and vector sum (X and Y) at centroid	*1*	*1*
TF:X, Y, SUM, Z	Thermal flux (heat flow rate/cross-sectional area) components and vector sum (X and Y) at centroid	*1*	*1*
FACE	Face label	*2*	-
NODES	Face nodes	*2*	-
AREA	Face area	*2*	*2*
TAVG, TBULK	Average of the two end nodal temperatures evaluated at peak value, fluid bulk temperature evaluated at peak value	*2*	*2*
HEAT RATE	Heat flow rate across face by convection	*2*	*2*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*2*	-
HFAVG	Average film coefficient of the face	-	*2*
TBAVG	Average face bulk temperature	-	*2*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*2*
HFLUX	Heat flux at each node of face	*2*	-

1. Gradient and flux peak at THETA = 0 and THETA = 90 ÷ MODE degrees
2. Output if a surface load is input
3. Available only at centroid as a ***GET** item.

Table 2, "PLANE75 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE75 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "PLANE75 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 PLANE75 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE75 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- The element must lie in the global X-Y plane as shown in *Figure 1, "PLANE75 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- If the thermal element is to be replaced by the analogous structural element (PLANE25) with surface stresses requested, the thermal element should be oriented so that face I-J (and also face K-L, if applicable) is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- Temperature dependent material properties (including the film coefficient) are assumed to be axisymmetric even if the temperature varies harmonically.
- If MODE = 0, properties are evaluated at the temperatures calculated in the previous substep (or at TUNIF if for the first substep).
- If MODE > 0, properties are evaluated at temperatures calculated from the previous MODE = 0 substep; if no MODE = 0 substep exists, then evaluation is done at 0.0 degrees.

PLANE75 Product Restrictions

There are no product restrictions for this element.

PLANE77

2-D 8-Node Thermal Solid

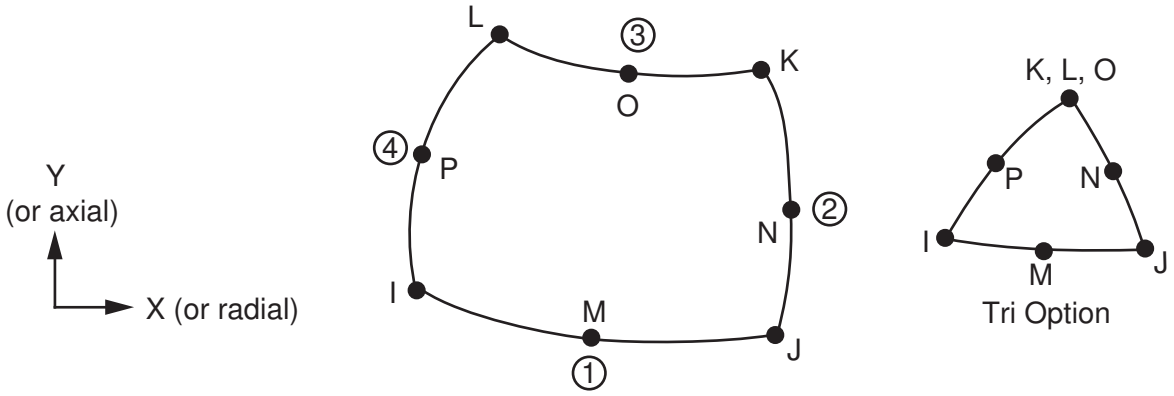
MP ME <> PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

PLANE77 Element Description

PLANE77 is a higher order version of the 2-D, 4-node thermal element (PLANE55). The element has one degree of freedom, temperature, at each node. The 8-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The 8-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis. See PLANE77 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE82). A similar axisymmetric thermal element which accepts nonaxisymmetric loading is PLANE78.

Figure 1 PLANE77 Geometry



PLANE77 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE77 Geometry"*. The element is defined by eight nodes and orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "PLANE77 Geometry"*. Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

A summary of the element input is given in *PLANE77 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE77 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

PLANE77 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE77 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE77 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	-
NODES	Face nodes	*1*	-
AREA	Face area	*1*	*1*
HFILM	Film coefficient	*1*	-
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HFLUX	Heat flux at each node of face	*1*	-

1. Output only if a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "PLANE77 Item and Component Labels" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE77 Item and Component Labels"*:

Name
output quantity as defined in the *Table 1, "PLANE77 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 PLANE77 Item and Component Labels

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE77 Assumptions and Restrictions

- The area of the element must be positive.
- The 2-D element must lie in an X-Y plane as shown in *Figure 1, "PLANE77 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid.
- If the thermal element is to be replaced by a PLANE82 structural element with surface stresses requested, the thermal element may be oriented such that face IJ and/or face KL is a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will require a fine mesh at the surface.

PLANE77 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

PLANE78

Axisymmetric-Harmonic 8-Node Thermal Solid

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

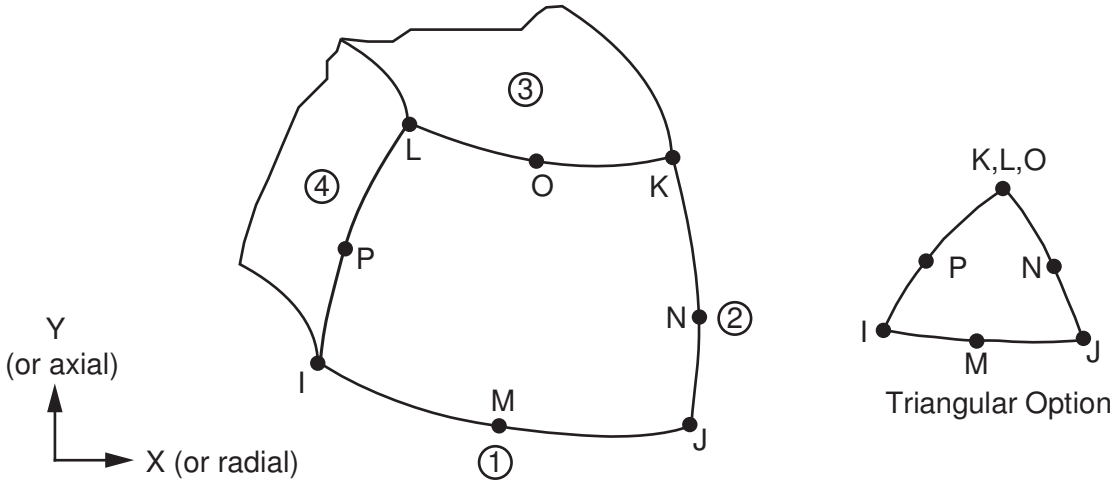
PLANE78 Element Description

PLANE78 is used as an axisymmetric ring element with a 3-D thermal conduction capability. The element has one degree of freedom, temperature, at each node. PLANE78 is a generalization of PLANE77 in that it allows a nonaxisymmetric loading. Various loading cases are described in *Section 2.13: Axisymmetric Elements with Non-axisymmetric Loads*.

The 8-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The element is applicable to a 2-D, axisymmetric, steady-state or transient thermal analysis. See PLANE78 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing the element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as PLANE83).

Figure 1 PLANE78 Geometry



PLANE78 Input Data

The geometry, node locations, and the coordinate system for this axisymmetric thermal solid element are shown in *Figure 1, "PLANE78 Geometry"*. The data input is essentially the same as for PLANE77 and is described in *PLANE77 Input Data*. The element input data also includes the number of harmonic waves (*MODE* on the **MODE** command) and the symmetry condition (*ISYM* on the **MODE** command). If *MODE* = 0 and *ISYM* = 1, the element behaves similar to the axisymmetric case of PLANE77. If *MODE* equals 1, the temperature is assumed to be 0° along an entire diameter. The *MODE* and *ISYM* parameters describe the type of temperature distribution and are discussed in detail in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying convections or heat fluxes (but not both) may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE78 Geometry"*. Harmonically varying heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

A summary of the element input is given in *PLANE78 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PLANE78 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Heat Fluxes --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Mode Number --
Input mode number on **MODE** command

Special Features
Birth and death

Loading Conditions
Input for *ISYM* on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

PLANE78 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE78 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. The face area and the heat flow rate are on a full 360° basis. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE78 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
MODE	Number of waves in loading	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*3*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, SUM, Z	Thermal gradient components and vector sum (X and Y) at centroid	*1*	*1*
TF:X, Y, SUM, Z	Thermal flux (heat flow rate/cross-sectional area) components and vector sum (X and Y) at centroid	*1*	*1*
FACE	Face label	*2*	*2*
NODES	Face nodes	*2*	*2*
AREA	Face area	*2*	*2*
HFILM	Film coefficient	*2*	*2*
TAVG, TBULK	Average of the two end nodal temperatures evaluated at peak value, fluid bulk temperature at peak value	*2*	*2*
HEAT RATE	Heat flow rate across face by convection	*2*	*2*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*2*	*2*
HFAVG	Average film coefficient of the face	-	*2*
TBAVG	Average face bulk temperature	-	*2*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*2*
HFLUX	Heat flux at each node of face	*2*	*2*

1. Gradient and flux peak at THETA = 0 and THETA = 90 ÷ Mode degrees
2. Output only if a surface load is input
3. Available only at centroid as a ***GET** item.

Table 2, "PLANE78 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE78 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE78 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 PLANE78 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE78 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- The element must lie in the global X-Y plane as shown in *Figure 1, "PLANE78 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- If the thermal element is to be replaced by the analogous structural element (PLANE83) with surface stresses requested, the thermal element should be oriented so that face IJ (and also face KL, if applicable) is a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- Temperature dependent material properties (including the film coefficient) are assumed to be axisymmetric even if the temperature varies harmonically.
- If *MODE* = 0, properties are evaluated at the temperatures calculated in the previous substep (or at TUNIF if for the first substep).
- If *MODE* > 0, properties are evaluated at temperatures calculated from the previous *MODE* = 0 substep; if no *MODE* = 0 substep exists, then evaluation is done at 0.0 degrees.

PLANE78 Product Restrictions

There are no product-specific restrictions for this element.

FLUID79

2-D Contained Fluid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID79 Element Description

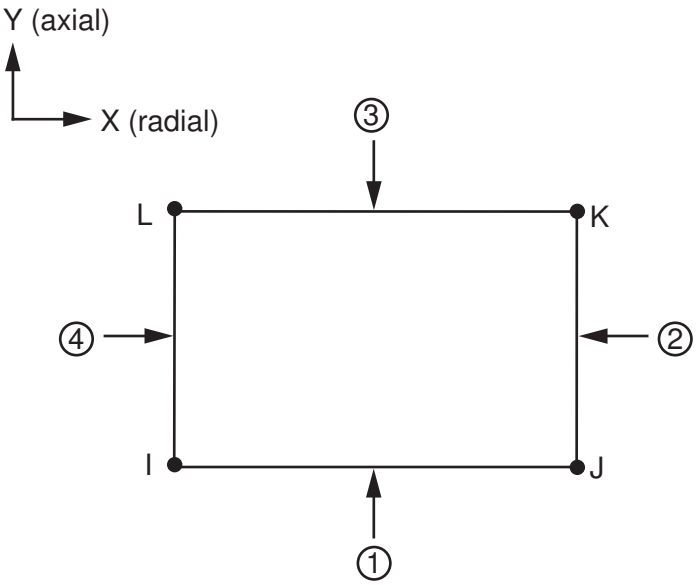
FLUID79 is a modification of the 2-D structural solid element (PLANE42). The fluid element is used to model fluids contained within vessels having no net flow rate. Another fluid element (FLUID116) is available to model fluids flowing in pipes and channels. The fluid element is particularly well suited for calculating hydrostatic pressures and fluid/solid interactions. Acceleration effects, such as in sloshing problems, as well as temperature effects, may be included.

The fluid element is defined by four nodes having two degrees of freedom at each node: translation in the nodal x and y directions. The element may be used in a structural analysis as a plane element or as an axisymmetric ring element. See FLUID79 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See FLUID80 for a 3-D version of this element.

Note

The reduced method is the only acceptable method for modal analyses using the ANSYS fluid elements.

Figure 1 FLUID79 Geometry



FLUID79 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID79 Geometry"*. The element input data includes four nodes and the isotropic material properties. EX, which is interpreted as the "fluid elastic modulus", should be the bulk modulus of the fluid (approximately 300,000 psi for water). The viscosity property (VISC) is used to compute a damping matrix for dynamic analyses (typical viscosity value for water is 1.639×10^{-7} lb-sec/in^{2}). The use of KEYOPT(2) for gravity springs is discussed in *FLUID80 Input Data*. Vertical acceleration (*ACELY* on the **ACEL** command) is needed for the gravity springs.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "FLUID79 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *FLUID79 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

FLUID79 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants
None

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, VISC, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features
None

KEYOPT(2)
Location of gravity springs:

0 --
Place gravity springs on all sides of all elements

1 --
Place gravity springs only on face of elements located on Y = 0.0 plane (elements must not have positive Y coordinates)

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

FLUID79 Output Data

The solution output associated with the element is in two forms:

- Degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID79 Element Output Definitions"*

The pressure and temperature are evaluated at the element centroid. Nodal forces and reaction forces are on a full 360° basis for axisymmetric models. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID79 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*1*
PRES	Pressures P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
TAVG	Average temperature	Y	-
PAVG	Average pressure	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "FLUID79 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID79 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "FLUID79 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,...,L
: sequence number for data at nodes I,J,...,L

Table 2 FLUID79 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	1	-	-	-	-
P1	SMISC	-	3	2	-	-

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P2	SMISC	-	-	5	4	-
P3	SMISC	-	-	-	7	6
P4	SMISC	-	8	-	-	9

FLUID79 Assumptions and Restrictions

- The area of the element must be positive.
- The fluid element must lie in an X-Y plane as shown in *Figure 1, "FLUID79 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- Radial motion should be constrained at the centerline.
- Usually the Y-axis is oriented in the vertical direction with the top surface at Y = 0.0.
- The element temperature is taken to be the average of the nodal temperatures.
- Elements should be rectangular whenever possible, as results are known to be of lower quality for some cases using nonrectangular shapes.
- Axisymmetric elements should always be rectangular.
- The nonlinear transient dynamic analysis should be used instead of the linear transient dynamic analysis for this element.
- A very small stiffness (EX x 1.0E-9) is associated with the shear and rotational strains to ensure static stability. See FLUID80 for more assumptions and restrictions.
- Only the lumped mass matrix is available.

FLUID79 Product Restrictions

There are no product-specific restrictions for this element.

FLUID80

3-D Contained Fluid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID80 Element Description

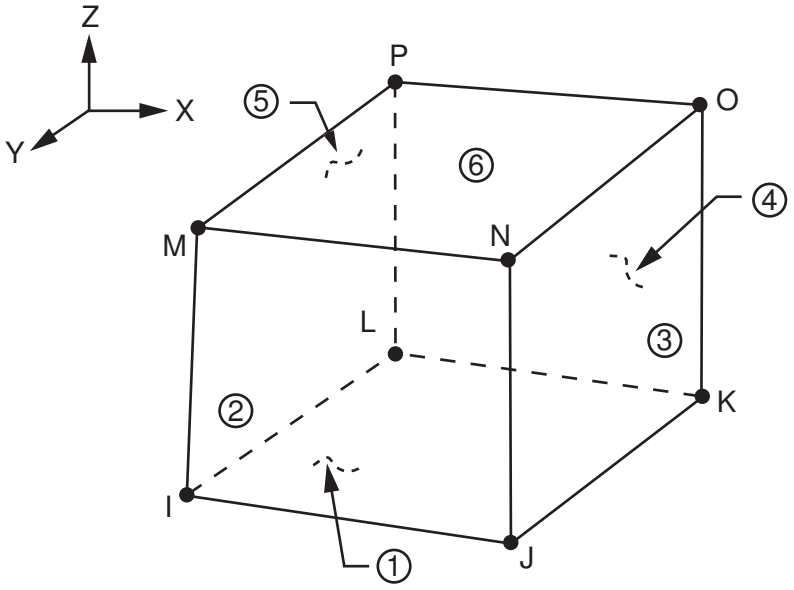
FLUID80 is a modification of the 3-D structural solid element (SOLID45). The fluid element is used to model fluids contained within vessels having no net flow rate. Another fluid element (FLUID116) is available to model fluids flowing in pipes and channels. The fluid element is particularly well suited for calculating hydrostatic pressures and fluid/solid interactions. Acceleration effects, such as in sloshing problems, as well as temperature effects, may be included.

The fluid element is defined by eight nodes having three degrees of freedom at each node: translation in the nodal x, y, and z directions. See FLUID80 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See FLUID79 for a 2-D version of this element.

Note

The reduced method is the only acceptable method for modal analyses using the ANSYS fluid elements.

Figure 1 FLUID80 Geometry



FLUID80 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID80 Geometry"*. The element input data includes eight nodes and the isotropic material properties. EX, which is interpreted as the "fluid elastic modulus", should be the bulk modulus of the fluid (approximately 300,000 psi for water). The viscosity property (VISC) is used to compute a damping matrix for dynamic analyses. A typical viscosity value for water is 1.639×10^{-7} lb-sec/in^2.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "FLUID80 Geometry"*. Positive pressures act into

the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The element also includes special surface effects, which may be thought of as gravity springs used to hold the surface in place. This is performed by adding springs to each node, with the spring constants being positive on the top of the element, and negative on the bottom. Gravity effects [**ACEL**] must be included if a free surface exists. For an interior node, the positive and negative effects cancel out, and at the bottom, where the fluid must be contained to keep the fluid from leaking out, the negative spring has no effect (as long as all degrees of freedom on the bottom are fixed). If the bottom consists of a flexible container, or if the degrees of freedom tangential to a curved surface are released, these negative springs may cause erroneous results and "negative pivot" messages. In this case, use of KEYOPT(2) = 1 is recommended.

These surface springs, while necessary to keep the free surface in place, artificially reduce the hydrostatic motion of the free surface. The error for a tank with vertical walls, expressed as a ratio of the computed answer over the correct answer is 1.0/(1.0 + (bottom pressure/bulk modulus)), which is normally very close to 1.0. Hydrodynamic results are not affected by this overstiffness.

A summary of the element input is given in *FLUID80 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

FLUID80 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, VISC, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features
None

KEYOPT(2)
Location of gravity springs:

0 --
Place gravity springs on all sides of all elements

1 --
Place gravity springs only on face of elements located on Z = 0.0 plane (elements must not have positive Z coordinates)

FLUID80 Output Data

The solution output associated with the element is in two forms:

- Degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID80 Element Output Definitions"*

The pressure and temperature are evaluated at the element centroid. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID80 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
TAVG	Average temperature	Y	-
PAVG	Average pressure	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "FLUID80 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID80 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "FLUID80 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,...,P
: sequence number for data at nodes I,J,...,P

Table 2 FLUID80 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
PRES	SMISC	1	-	-	-	-	-	-	-	-
P1	SMISC	-	3	2	5	4	-	-	-	-
P2	SMISC	-	6	7	-	-	9	8	-	-
P3	SMISC	-	-	10	11	-	-	13	12	-
P4	SMISC	-	-	-	14	15	-	-	17	16
P5	SMISC	-	19	-	-	18	20	-	-	21
P6	SMISC	-	-	-	-	-	22	23	24	25

FLUID80 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1, "FLUID80 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- Structures are usually modeled with the Z-axis oriented in the vertical direction and the top surface at Z = 0.0.
- The element temperature is taken to be the average of the nodal temperatures.
- Elements should be rectangular (brick shaped) whenever possible, as results are known to be of lower quality for some cases using nonrectangular shapes.
- The nonlinear transient dynamic analysis should be used instead of the linear transient dynamic analysis for this element.
- For the case of a modal analysis with irregular meshes, one can expect one or more low frequency eigenvectors, representing internal fluid motions, without significantly affecting the vertical motion of the free surface.
- The amount of flow permitted is limited to that which will not cause gross distortions in the element.
- The large deflection option should not be used with this element.
- In a reduced analysis, master degrees of freedom should be selected at all nodes on the free fluid surface in the direction normal to the free surface. Other master degrees of freedom, if any, should only be selected normal to one or more flat planes within the fluid, with all nodes on these planes being included. Other selections may produce large internal rotations.
- When used for a static application, the free surface must be input flat. Gravity must be input if there is a free surface. The element gives valid nodal forces representing hydrostatic pressure and also valid vertical displacements at the free surface. Other nodal displacements, which may be large, represent energy-free internal motions of the fluid.
- Fluid element at a boundary should not be attached directly to structural elements but should have separate, coincident nodes that are coupled only in the direction normal to the interface.
- Arbitrarily small numbers are included to give the element some shear and rotational stability.
- Only the lumped mass matrix is available.

FLUID80 Product Restrictions

There are no product-specific restrictions for this element.

FLUID81

Axisymmetric-Harmonic Contained Fluid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID81 Element Description

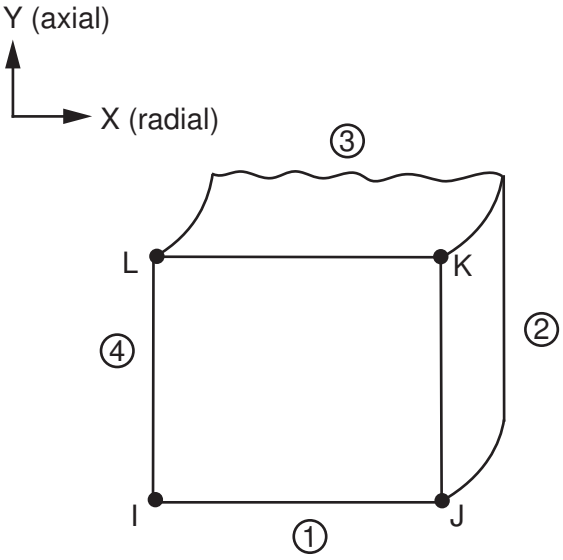
FLUID81 is a modification of the axisymmetric structural solid element (PLANE25). The element is used to model fluids contained within vessels having no net flow rate. It is defined by four nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element is used in a structural analysis as an axisymmetric ring element.

The element is a generalization of the axisymmetric version of FLUID79, the 2-D fluid element, in that the loading need not be axisymmetric. Various loading cases are described in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*. The fluid element is particularly well suited for calculating hydrostatic pressures and fluid/solid interactions. Acceleration effects, such as in sloshing problems, as well as temperature effects, may be included. See FLUID81 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Another fluid element (FLUID116) is available to model fluids flowing in pipes and channels.

Note

The reduced method is the only acceptable method for modal analyses using the ANSYS fluid elements.

Figure 1 FLUID81 Geometry



FLUID81 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID81 Geometry"*. The element input data includes four nodes, the number of harmonic waves (*MODE* on the **MODE** command), the symmetry condition (*ISYM* on the **MODE** command), and the isotropic material properties. If *MODE* = 0 and *ISYM* = 1, the element behaves similar to the axisymmetric case of FLUID79. The *MODE* and *ISYM* parameters are discussed in detail in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*. EX, which is interpreted as the "fluid elastic modulus," should be the bulk modulus of the fluid (approximately 300,000 psi

for water). The viscosity property (VISC) is used to compute a damping matrix for dynamic analyses. A typical viscosity value for water is 1.639 x 10^{-7} lb-sec/in^2. Density (DENS) must be input as a positive number.

The use of KEYOPT(2) for gravity springs is discussed in *FLUID80 Input Data*. Vertical acceleration (*ACELY* on the **ACEL** command) is needed for the gravity springs regardless of the value of *MODE*, even for a modal analysis. Harmonically varying nodal forces, if any, should be input on a full 360° basis.

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "FLUID81 Geometry"*. Positive pressures act into the element.

Harmonically varying temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *FLUID81 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

FLUID81 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
EX, ALPX (or CTEX *or* THSX), DENS, VISC, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Mode Number
Input mode number on **MODE** command

Loading Condition
Input for *ISYM* on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

Special Features
None

KEYOPT(2)
Location of gravity springs:

0 --
Place gravity springs on all sides of all elements

1 --
Place gravity springs only on face of elements located on Y = 0.0 plane (element must not have positive Y coordinates)

FLUID81 Output Data

The solution output associated with the element is in two forms:

- Degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID81 Element Output Definitions"*

The pressure and temperature are evaluated at the element centroid. Nodal forces and reaction forces are on a full 360° basis.

In the displacement printout, the UZ component is out-of-phase with the UX and UY components. For example, in the *MODE* = 1, *ISYM* = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. Printout for combined loading cases may be obtained from the POST1 routine. We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The following notation is used in *Table 1, "FLUID81 Element Output Definitions"*:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID81 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
ISYM	Loading Key	*1*	*1*
MODE	Number of waves in loading	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
TAVG	Average temperature	Y	-
PAVG	Average pressure	Y	Y

1. If *ISYM* is:

   1 - Symmetric loading

   -1 - Antisymmetric loading

2. Available only at centroid as a ***GET** item.

Table 2, "FLUID81 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID81 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID81 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,L
sequence number for data at nodes I,J,...,L

Table 2 FLUID81 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	1	-	-	-	-
P1	SMISC	-	3	2	-	-
P2	SMISC	-	-	5	4	-
P3	SMISC	-	-	-	7	6
P4	SMISC	-	8	-	-	9

FLUID81 Assumptions and Restrictions

- The area of the element must be positive.
- The fluid element must lie in an X-Y plane as shown in *Figure 1, "FLUID81 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- The Y-axis should be oriented in the vertical direction and the top surface is usually at Y = 0.0.
- The element temperature is taken to be the average of the nodal temperatures.
- Temperature dependent material properties, if any, are evaluated at the reference temperature [**TREF**].
- Elements should be rectangular since results are known to be of lower quality for nonrectangular shapes.
- The nonlinear transient dynamic analysis should be used instead of the linear transient dynamic analysis for this element.
- A lumped mass matrix may be obtained for this element with the **LUMPM** command.
- See FLUID80 for more assumptions and restrictions.

FLUID81 Product Restrictions

There are no product-specific restrictions for this element.

PLANE82

2-D 8-Node Structural Solid

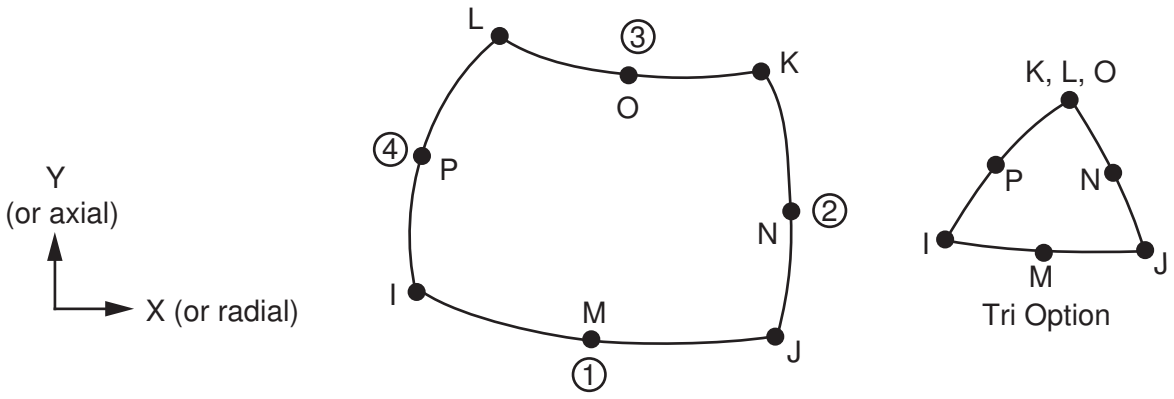
MP ME ST PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

PLANE82 Element Description

PLANE82 is a higher order version of the 2-D, four-node element (PLANE42). It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries.

The 8-node element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element or as an axisymmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Various printout options are also available. See PLANE82 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See PLANE83 for a description of an axisymmetric element which accepts nonaxisymmetric loading.

Figure 1 PLANE82 Geometry



PLANE82 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE82 Geometry"*.

A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. Besides the nodes, the element input data includes a thickness (TK) (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE82 Geometry"*. Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis. KEYOPT(5) and KEYOPT(6) parameters provide various element printout options (see *Section 2.2.2: Element Solution*).

Initial state conditions previously handled via the **ISTRESS** command will be discontinued for this element. The **INISTATE** command will provide increased functionality, but only via the Current Technology elements (180,181, etc.). To continue using Initial State conditions in future versions of ANSYS, consider switching to the appropriate Current Technology element. For more information on setting Initial State values see the **INISTATE** command and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. For more information on current -vs- legacy element technologies see *Section 2.17: Legacy vs. Current Element Technologies* in the *Elements Reference*

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *PLANE82 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE82 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT (3) = 0, 1, or 2
THK - Thickness, if KEYOPT (3) = 3

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (I-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Fluences --
FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO, CHABOCHE, HILL)
Creep (CREEP, RATE)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain

Birth and death
Adaptive descent
Initial stress import

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (TK) real constant input

KEYOPT(5)
Extra element output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points

2 --
Nodal Stress Solution

KEYOPT(6)
Extra surface output:

0 --
Basic element solution

1 --
Surface solution for face I-J also

2 --
Surface solution for both faces I-J and K-L also (surface solution valid for linear materials only)

3 --
Nonlinear solution at each integration point also

4 --
Surface solution for faces with nonzero pressure

KEYOPT(9)
Initial stress subroutine option (available only through direct input of the KEYOPT command):

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS. See the *Guide to ANSYS User Programmable Features* for user written subroutines

PLANE82 Output Data

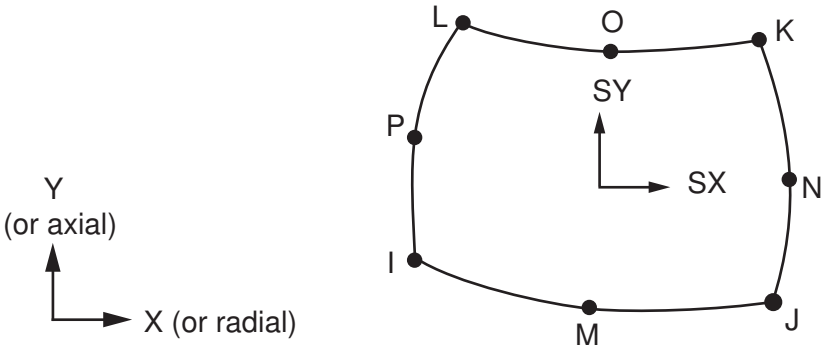
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE82 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE82 Stress Output"*.

The element stress directions are parallel to the element coordinate system. Surface stresses are available on any face. Surface stresses on face IJ, for example, are defined parallel and perpendicular to the IJ line and along the Z axis for a plane analysis or in the hoop direction for an axisymmetric analysis. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE82 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE82 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
THICK	Average thickness	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)	Y	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	Y	-
S:INT	Stress intensity	Y	-

Name	Definition	O	R
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [*4*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	Y	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
EPPL:X, Y, XY, Z	Average plastic strains	*2*	*2*
EPPL:EQV	Equivalent plastic strain [*4*]	-	*2*
EPCR:X, Y, XY, Z	Average creep strains	*2*	*2*
EPCR:EQV	Equivalent creep strain [*4*]	-	*2*
EPSW:	Swelling strain	*2*	*2*
NL:EPEQ	Equivalent plastic strain	*2*	*2*
NL:SRAT	Ratio of trial stress to stress on yield surface	*2*	*2*
NL:SEPL	Equivalent stress on stress-strain curve	*2*	*2*
NL:HPRES	Hydrostatic pressure	-	*2*
FACE	Face label	*1*	*1*
EPEL(PAR, PER, Z)	Surface elastic strains (parallel, perpendicular, Z or hoop)	*1*	*1*
TEMP	Surface average temperature	*1*	*1*
S(PAR, PER, Z)	Surface stresses (parallel, perpendicular, Z or hoop)	*1*	*1*
SINT	Surface stress intensity	*1*	*1*
SEQV	Surface equivalent stress	*1*	*1*
LOCI:X, Y, Z	Integration point locations	-	Y

1. Surface output (if KEYOPT(6) is 1, 2 or 4)
2. Nonlinear solution (if the element has a nonlinear material)
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2 PLANE82 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Pt. Solution	EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR, EPSW	*1*	-
Integration Point Stress Solution	TEMP, SINT, SEQV, EPEL, S	*2*	-
Nodal Stress Solution	TEMP, S, SINT, SEQV	*3*	-

1. Output at each integration point, if the element has a nonlinear material and KEYOPT(6) = 3
2. Output at each integration point, if KEYOPT(5) = 1
3. Output at each vertex node, if KEYOPT(5) = 2

Note

For axisymmetric solutions, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains.

Table 3, "PLANE82 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "PLANE82 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE82 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I,J,...,P

Table 3 PLANE82 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	-	-	-	-	-	-
P2	SMISC	-	-	4	3	-	-	-	-	-
P3	SMISC	-	-	-	6	5	-	-	-	-
P4	SMISC	-	7	-	-	8	-	-	-	-
S:1	NMISC	-	1	6	11	16	-	-	-	-
S:2	NMISC	-	2	7	12	17	-	-	-	-
S:3	NMISC	-	3	8	13	18	-	-	-	-
S:INT	NMISC	-	4	9	14	19	-	-	-	-
S:EQV	NMISC	-	5	10	15	20	-	-	-	-
FLUEN	NMISC	-	21	22	23	24	25	26	27	28
THICK	NMISC	29	-	-	-	-	-	-	-	-

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE82 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE82 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

PLANE82 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads are not applicable.
- The only special feature allowed is stress stiffening.
- KEYOPT(6) = 3 is not applicable.

PLANE83

Axisymmetric-Harmonic 8-Node Structural Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

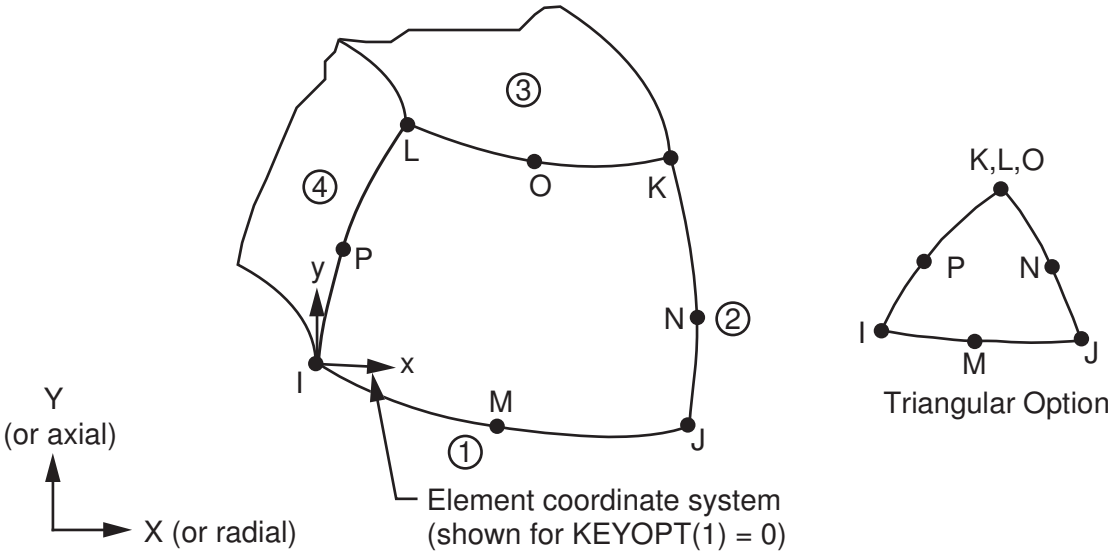
PLANE83 Element Description

PLANE83 is used for 2-D modeling of axisymmetric structures with nonaxisymmetric loading. Examples of such loading are bending, shear, or torsion. The element has three degrees of freedom per node: translations in the nodal x, y, and z directions. For unrotated nodal coordinates, these directions correspond to the radial, axial, and tangential directions, respectively.

This element is a higher order version of the 2-D, four-node element (PLANE25). It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The element is also a generalization of the axisymmetric version of PLANE82, the 2-D 8-node structural solid element, in that the loading need not be axisymmetric. Various loading cases are described in *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries. See PLANE83 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE83 Geometry



PLANE83 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE83 Geometry"*. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element input data is essentially the same as for PLANE82, except as follows: Z-direction material properties (EZ, ALPZ, etc.) may be input. MODE and ISYM are used to describe the harmonic loading condition (see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads* for more details).

Element loads are described in *Section 2.8: Node and Element Loads*. Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE83 Geometry"*. Positive pressures act into the element. Harmonically varying temperatures may be input as element body loads at the

nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The KEYOPT(3) parameter is used for temperature loading with MODE greater than zero and temperature dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If MODE equals zero, the material properties are always evaluated at the average element temperature. KEYOPT(4), (5), and (6) provide various element printout options (see *Section 2.2.2: Element Solution*).

A summary of the element input is given in *PLANE83 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

PLANE83 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Mode Number
Input mode number on **MODE** command

Loading Condition
Input for *ISYM* on **MODE** command

1--
Symmetric loading

-1--
Antisymmetric loading

Special Features

Stress stiffening
Birth and death

KEYOPT(1)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(3)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(4)
Extra stress output:

0 --
Basic element solution (not extra output)

1 --
Repeat basic solution for all integration points

2 --
Nodal stress solution

KEYOPT(5)
Combined stress output:

0 --
No combined stress solution

1 --
Combined stress solution at centroid and nodes

KEYOPT(6)
Extra surface output (surface solution is valid only for isotropic materials):

0 --
Basic element solution (no extra output)

1 --
Surface solution for face I-J also

2 --
Surface solution for both faces I-J and K-L also

PLANE83 Output Data

The solution output associated with the element is in two forms:

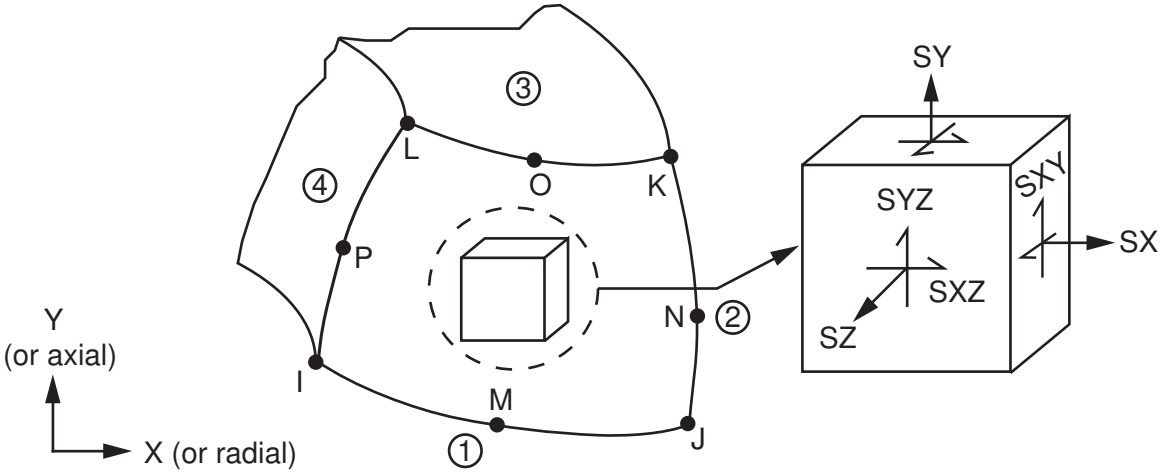
- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE83 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE83 Stress Output"*.

In the displacement printout, the UZ component is out-of-phase with the UX and UY components. For example, in the MODE = 1, ISYM = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. The same occurs for the reaction forces (FX, FY, etc.). We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see *Section 2.13: Axisymmetric Elements with Nonaxisymmetric Loads*.

The element stress directions are parallel to the element coordinate system. The sign convention on the surface shears is such that for a rectangular element that is lined up parallel to the axes with node J in the positive Y direction from node I, the shear stresses on surfaces I-J and K-L are analogous to the centroidal SYZ in both definition and sign. Stress components which are inherently zero for a load case are printed for clarity. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE83 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE83 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	-
MODE	Number of waves in loading	Y	Y
VOLU:	Volume	Y	Y
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
S:X, Y, Z	Direct stresses (radial, axial, hoop) at *PK ANG* locations	Y	Y
S:XY, YZ, XZ	Shear stresses (radial-axial, axial-hoop, radial-hoop) at *PK ANG* locations	Y	Y
S:1, 2, 3	Principal stresses at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*

Name	Definition	O	R
S:INT	Stress intensity at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*
S:EQV	Equivalent stress at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	*1*	*1*
EPEL:X, Y, Z, XY	Elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain [*4*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	Y	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
PK ANG	Angle where stresses have peak values: 0 and 90/MODE°. Blank if MODE = 0.	Y	Y
XC, YC	Location where results are reported	Y	*3*
FACE	Face label	*2*	*2*
TEMP	Surface average temperature	*2*	*2*
EPEL(PAR, PER, Z, SH)	Surface strains (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	*2*	*2*
S(PAR, PER, Z, SH)	Surface stresses (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	*2*	*2*

1. These items are output only if KEYOPT(5) = 1.
2. These items are printed only if KEYOPT(6) is greater than zero.
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2, "PLANE83 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE83 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "PLANE83 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,K,L
: sequence number for data at nodes I,J,K,L

Table 2 PLANE83 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
THETA = 0					
S1	NMISC	1	16	31	46
S2	NMISC	2	17	32	47
S3	NMISC	3	18	33	48
SINT	NMISC	4	19	34	49
SEQV	NMISC	5	20	35	50
THETA = 90/MODE					
S1	NMISC	6	21	36	51
S2	NMISC	7	22	37	52
S3	NMISC	8	23	38	53
SINT	NMISC	9	24	39	54
SEQV	NMISC	10	25	40	55
EXTR Values					
S1	NMISC	11	26	41	56
S2	NMISC	12	27	42	57
S3	NMISC	13	28	43	58
SINT	NMISC	14	29	44	59
SEQV	NMISC	15	30	45	60

Note

The NMISC items (1 thru 60) in the above table represent the combined stress solution, KEYOPT(5) = 1. If MODE = 0, their values are zero at THETA = 90/MODE and at EXTR.

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE83 Assumptions and Restrictions

- The area of the element must be positive.
- The element must be defined in the global X-Y plane as shown in *Figure 1, "PLANE83 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The element assumes a linear elastic material.
- Post-analysis superposition of results is valid only with other linear elastic solutions.
- The element should not be used with the large deflection option.
- The element may not be deactivated with the **EKILL** command.
- The element temperature is taken to be the average of the nodal temperatures.
- Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met.

Modeling hints:

- If shear effects are important in a shell-like structure, at least two elements through the thickness should be used.
- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.

PLANE83 Product Restrictions

There are no product-specific restrictions for this element.

SOLID87

3-D 10-Node Tetrahedral Thermal Solid

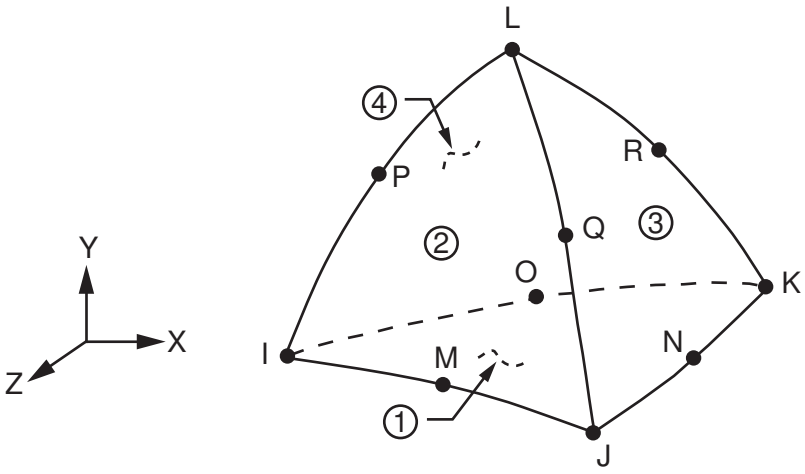
MP ME <> PR PRN DS <> <> <> <> <> PP VT
Product Restrictions

SOLID87 Element Description

SOLID87 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node.

The element is applicable to a 3-D, steady-state or transient thermal analysis. See SOLID87 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as SOLID92). A 20-node thermal solid element, SOLID90, is also available.

Figure 1 SOLID87 Geometry



SOLID87 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID87 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID87 Geometry"*. Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

For phase change problems, use KEYOPT(1) = 1 (diagonalized specific heat matrix). For convection regions with strong thermal gradients, use KEYOPT(5) = 1 (consistent convection matrix).

A summary of the element input is given in *SOLID87 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID87 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(5)
Surface convection matrix:

0 --
Diagonalized convection matrix

1 --
Consistent convection matrix

SOLID87 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID87 Element Output Definitions"*.

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID87 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Convection face label	*1*	-
NODES	Convection face corner nodes	*1*	-
AREA	Convection face area	*1*	*1*
HFILM	Film coefficient	*1*	-
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HFLUX	Heat flux at each node of face	*1*	-

1. Output if a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "SOLID87 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SOLID87 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SOLID87 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

FC*n*
: sequence number for solution items for element Face *n*

Table 2 SOLID87 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

SOLID87 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in *Figure 1, "SOLID87 Geometry"* or may have node L below the IJK plane.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

SOLID87 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

VISCO88

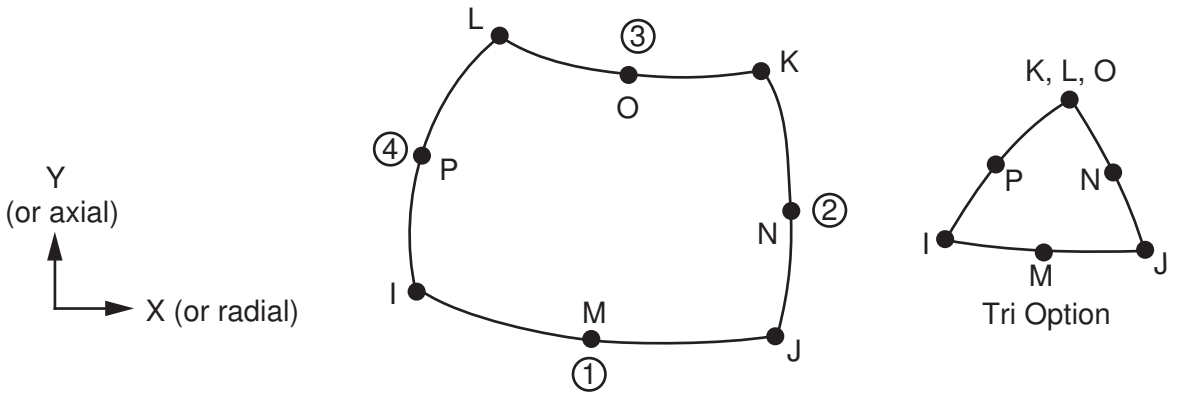
2-D 8-Node Viscoelastic Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

VISCO88 Element Description

VISCO88 is a quadratic isoparametric element. The element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane strain or as an axisymmetric element. The element has thermorheologically simple (TRS) viscoelastic and stress stiffening capabilities. Various printout options are also available. See VISCO88 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 VISCO88 Geometry



VISCO88 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "VISCO88 Geometry"*. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. VISCO88 uses a viscoelastic material model that is defined by the **TB** and **TBDATA** commands. The constant table is started by using the **TB** command with *Lab* = EVISC. Up to 95 constants may be defined with the **TBDATA** commands. Details are provided in *Section 2.5.4: Viscoelastic Material Constants*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "VISCO88 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis. KEYOPT(5) provides various element printout options (see *Section 2.2.2: Element Solution*).

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *VISCO88 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

VISCO88 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY

Real Constants
None

Material Properties
DAMP, DENS (see *Section 2.5.4: Viscoelastic Material Constants* for others)

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Viscoelasticity
Stress stiffening
Adaptive descent

KEYOPT(3)
Element behavior:

0, 2 --
Plane strain (Z strain = 0.0)

1 --
Axisymmetric

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic element printout for all integration points

2 --
Nodal stress printout

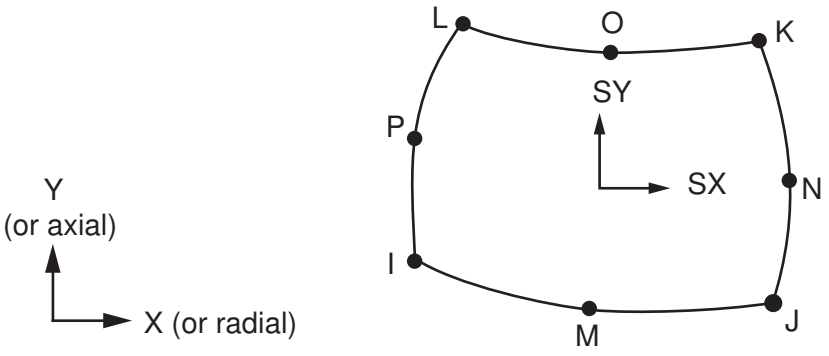
VISCO88 Output Data

The solution output associated with the element is in two forms:

- nodal displacements included in the overall nodal solution
- additional element output as shown in *Table 1, "VISCO88 Element Output Definitions (KEYOPT(5) = 0)"*.

The element stress directions are shown in *Figure 2, "VISCO88 Stress Output"*. The directions are parallel to the global Cartesian coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 VISCO88 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 VISCO88 Element Output Definitions (KEYOPT(5) = 0)

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
PRES	Pressures P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
S:X, Y, Z, XY	Stresses [*3*]	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Average elastic strain	-	Y
EPEL:EQV	Equivalent elastic strain [*4*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strain	-	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
GR STRAIN	Growth strain (recoverable and irrecoverable thermally induced effects)	Y	Y
FICT TEMP	Fictive or pseudo temperature	Y	Y
EFF BULK MOD	Effective bulk modulus	*1*	*1*
EFF SHEAR MOD	Effective shear modulus	*1*	*1*

1. Element solution output quantities EFF BULK MOD and EFF SHEAR MOD might not correspond to the effective bulk and shear moduli, respectively. The output values are actually intermediate quantities in the computation of bulk and shear moduli and do not represent any true tangible material properties. These quantities are also stored on the results files as nonsummable miscellaneous (NMISC) data items 25 through 32 (**ETABLE** command).
2. Available only at centroid as a ***GET** item.
3. For axisymmetric solutions, the X, Y, Z, and XY stress and strain outputs correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2 VISCO88 Miscellaneous Element Output (KEYOPT(5) = 1 or 2)

Description	Names of Items Output	O	R
Integration Point Solution (KEYOPT(5) = 1)	S(X, Y, Z, XY)[*1*], S(1, 2, 3), SINT, SEQV, GR STRAIN, FICT TEMP, EFF BULK MOD, EFF SHEAR MOD	Y	-
Nodal Stress Solution (KEYOPT(5) = 2)	TEMP, S(X, Y, Z, XY)[*1*], SINT, SEQV	Y	-

1. For axisymmetric solutions, the X, Y, Z, and XY stress and strain outputs correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively.

Note

Displacements and nodal forces are the total (not incremental) values.

Table 3, "VISCO88 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "VISCO88 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "VISCO88 Element Output Definitions (KEYOPT(5) = 0)"*

Item
: predetermined Item label for **ETABLE** command

I,J,K,L
: sequence number for data at nodes I,J,K,L

Table 3 VISCO88 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
S:1	NMISC	1	6	11	16
S:2	NMISC	2	7	12	17
S:3	NMISC	3	8	13	18

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
S:INT	NMISC	4	9	14	19
S:EQV	NMISC	5	10	15	20
FICT TEMP	NMISC	21	22	23	24
EFF BULK MOD	NMISC	25	26	27	28
EFF SHEAR MOD	NMISC	29	30	31	32
GR STRAIN	NMISC	33	34	35	36

VISCO88 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "VISCO88 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

VISCO88 Product Restrictions

There are no product-specific restrictions for this element.

VISCO89

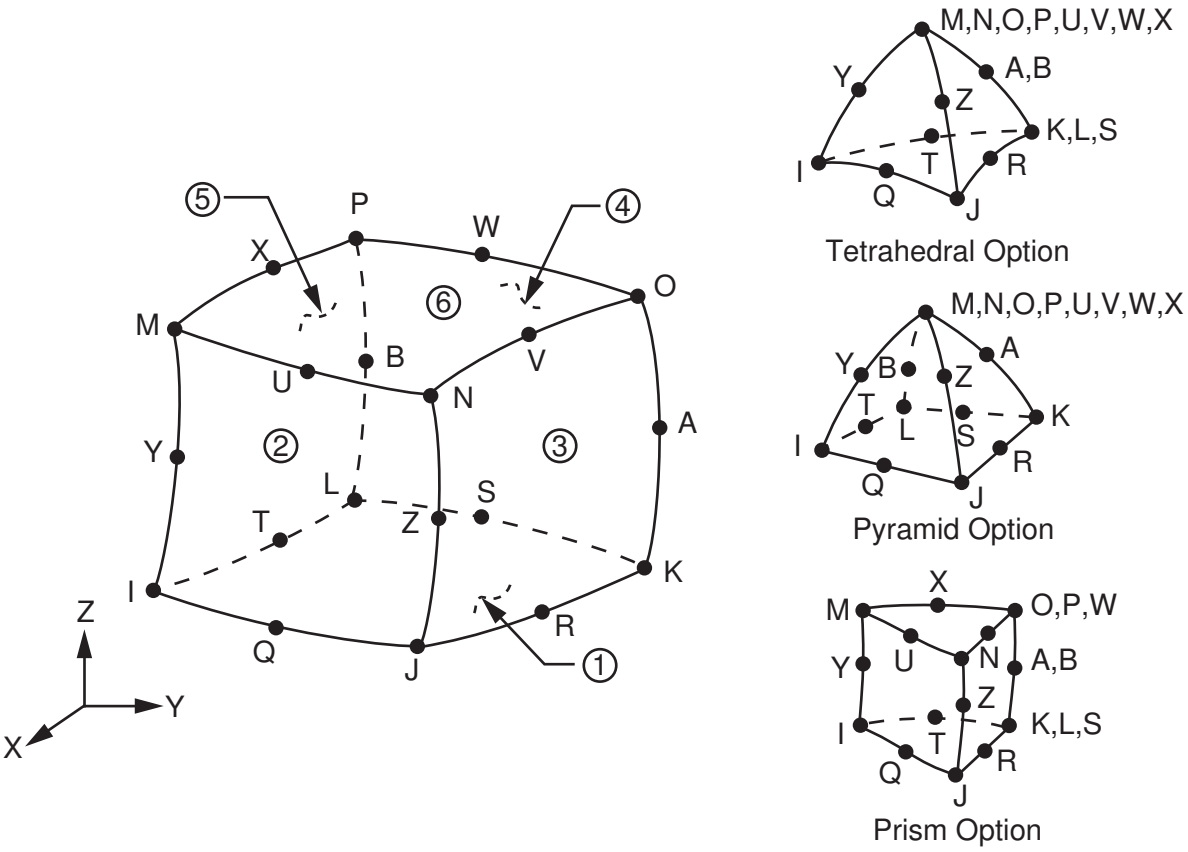
3-D 20-Node Viscoelastic Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

VISCO89 Element Description

VISCO89 is a quadratic isoparametric element. The element is defined by 20 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has thermorheologically simple (TRS) viscoelastic and stress stiffening capabilities. Various printout options are also available. See VISCO89 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 VISCO89 Geometry



VISCO89 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "VISCO89 Geometry"*. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

A prism-shaped element may be formed by defining the same node numbers for nodes K, L and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "VISCO89 Geometry"*. VISCO89 uses a viscoelastic material model that is defined by the **TB** and **TBDATA** commands. The constant table is started by using the **TB** command with *Lab* = EVISC. Up to 95 constants may be defined with the **TBDATA** commands. Details are provided in *Section 2.5.4: Viscoelastic Material Constants*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "VISCO89 Geometry"*. Positive pressures act into

the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

KEYOPT(5) provides various element printout options (see *Section 2.2.2: Element Solution*).

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *VISCO89 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

VISCO89 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
DAMP, DENS (see *Section 2.5.4: Viscoelastic Material Constants* for others)

Surface Loads

Pressures --
1-JILK, 2-IJNM, 3-JKON, 4-KLPO, 5-LIMP, 6-MNOP

Body Loads

Temperatures --
T(I), T(J), --, T(Z), T(A), T(B)

Special Features

Viscoelasticity
Stress stiffening
Adaptive descent

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic element printout for all integration points

2 --
Nodal stress printout

VISCO89 Output Data

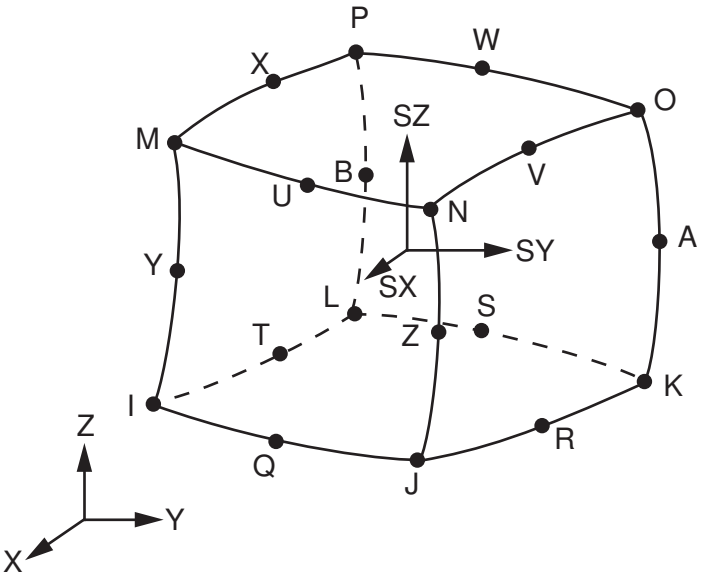
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

- Additional element output as shown in *Table 1, "VISCO89 Element Output Definitions"*

The element stress directions are shown in *Figure 2, "VISCO89 Stress Output"*. The directions are parallel to the global Cartesian coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 VISCO89 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 VISCO89 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
CORNER NODES	Corner nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
PRES	Pressures P1 at nodes J, I, L, K; P2 at nodes I, J, N, M; P3 at nodes J, K, O, N; P4 at nodes K, L, P, O; P5 at nodes L, I, M, P; P6 at nodes M, N, O, P	Y	Y
TEMP	T(I), T(J), --, T(Z), T(A), T(B)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y

Name	Definition	O	R
EPEL:X, Y, Z, XY, YZ, XZ	Average elastic strain	-	Y
EPEL:EQV	Equivalent elastic strain [*3*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strain	-	Y
EPTH:EQV	Equivalent thermal strain [*3*]	-	Y
GR STRAIN	Growth strain (recoverable and irrecoverable thermally induced effects)	Y	Y
FICT TEMP	Fictive or pseudo temperature	Y	Y
EFF BULK MOD	Effective bulk modulus	*1*	*1*
EFF SHEAR MOD	Effective shear modulus	*1*	*1*

1. Element solution output quantities EFF BULK MOD and EFF SHEAR MOD might not correspond to the effective bulk and shear moduli, respectively. The output values are actually intermediate quantities in the computation of bulk and shear moduli and do not represent any true tangible material properties. These quantities are also stored on the results files as nonsummable miscellaneous (NMISC) data items 49 through 64 (**ETABLE** command).
2. Available only at centroid as a ***GET** item.
3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2 VISCO89 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	S(X, Y, Z, XY, YZ, XZ), S(1, 2, 3), SINT, SEQV, GR STRAIN, FICT TEMP, EFF BULK MOD, EFF SHEAR MOD	*1*	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each node, if KEYOPT(5) = 2

Note

Displacements and nodal forces are the total (not incremental) values.

Table 3, "VISCO89 Item and Component Labels" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "VISCO89 Item and Component Labels"*:

Name
: output quantity as defined in the *Table 1, "VISCO89 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,...,P
: sequence number for data at nodes I,J,...,P

Table 3 VISCO89 Item and Component Labels

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40
FICT TEMP	NMISC	41	42	43	44	45	46	47	48
EFF BULK MOD	NMISC	49	50	51	52	53	54	55	56
EFF SHEAR MOD	NMISC	57	58	59	60	61	62	63	64
GR STRAIN	NMISC	65	66	67	68	69	70	71	72

VISCO89 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "VISCO89 Geometry"* or may have the planes IJKL and MNOP interchanged.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.

VISCO89 Product Restrictions

There are no product-specific restrictions for this element.

SOLID90

3-D 20-Node Thermal Solid

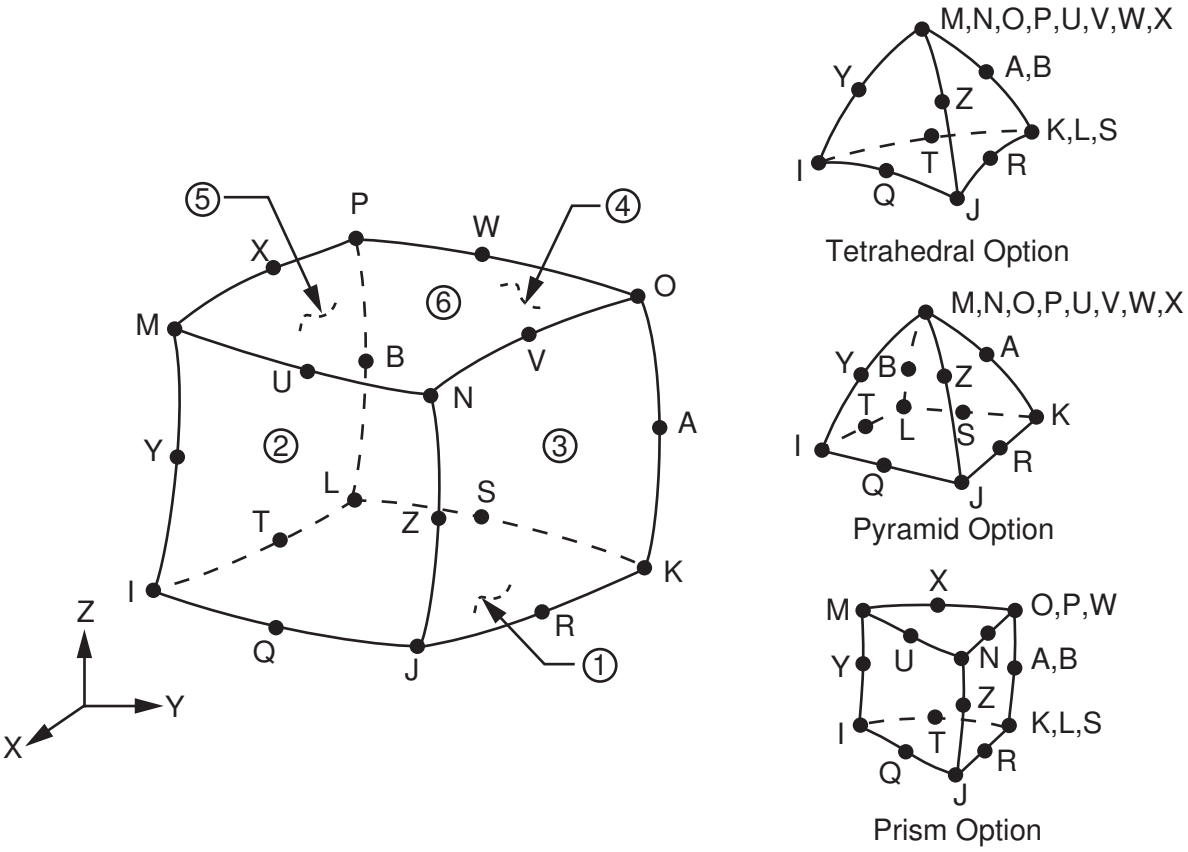
MP ME <> PR PRN DS <> <> <> <> <> PP VT
Product Restrictions

SOLID90 Element Description

SOLID90 is a higher order version of the 3-D eight node thermal element (SOLID70). The element has 20 nodes with a single degree of freedom, temperature, at each node. The 20-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The 20-node thermal element is applicable to a 3-D, steady-state or transient thermal analysis. See SOLID90 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as SOLID95).

Figure 1 SOLID90 Geometry



SOLID90 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID90 Geometry"*. The element is defined by 20 node points and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID90 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Specific heat and density are ignored for steady-state solutions. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID90 Geometry"*. Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

For phase change problems, use KEYOPT(1) = 1 (diagonalized specific heat matrix).

A summary of the element input is given in *SOLID90 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID90 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --

HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R),
HG(S), HG(T), HG(U), HG(V), HG(W), HG(X), HG(Y), HG(Z), HG(A), HG(B)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

SOLID90 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID90 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID90 Element Output Definitions

Label	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), ..., HG(Z), HG(A), HG(B)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	*1*	-
NODES	Corner nodes on this face	*1*	-
AREA	Face area	*1*	*1*
HFILM	Film coefficient	*1*	-
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HFLUX	Heat flux at each node of face	*1*	-
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*

1. Output only if a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "SOLID90 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The*

Item and Sequence Number Table of this manual for more information. The following notation is used in *Table 2, "SOLID90 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID90 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 2 SOLID90 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID90 Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "SOLID90 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The condensed face of a prism-shaped element should not be defined as a convection face.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal element is to be replaced by a SOLID95 structural element with surface stresses requested, the thermal element should be oriented such that face IJNM and/or face KLPO is a free surface.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- For transient solutions using the **THOPT**,QUASI option, the program removes the midside nodes from any face with a convection load. A temperature solution is not available for them. Do not use the midside nodes on these faces in constraint equations or with contact. If you use these faces for those situations, remove the midside nodes first.
- Degeneration to the form of pyramid should be used with caution.
- The element sizes, when degenerated, should be small in order to minimize the field gradients.

- Pyramid elements are best used as filler elements or in meshing transition zones.

SOLID90 Product Restrictions

ANSYS Professional

- No Birth and Death.

SHELL91

Nonlinear Layered Structural Shell

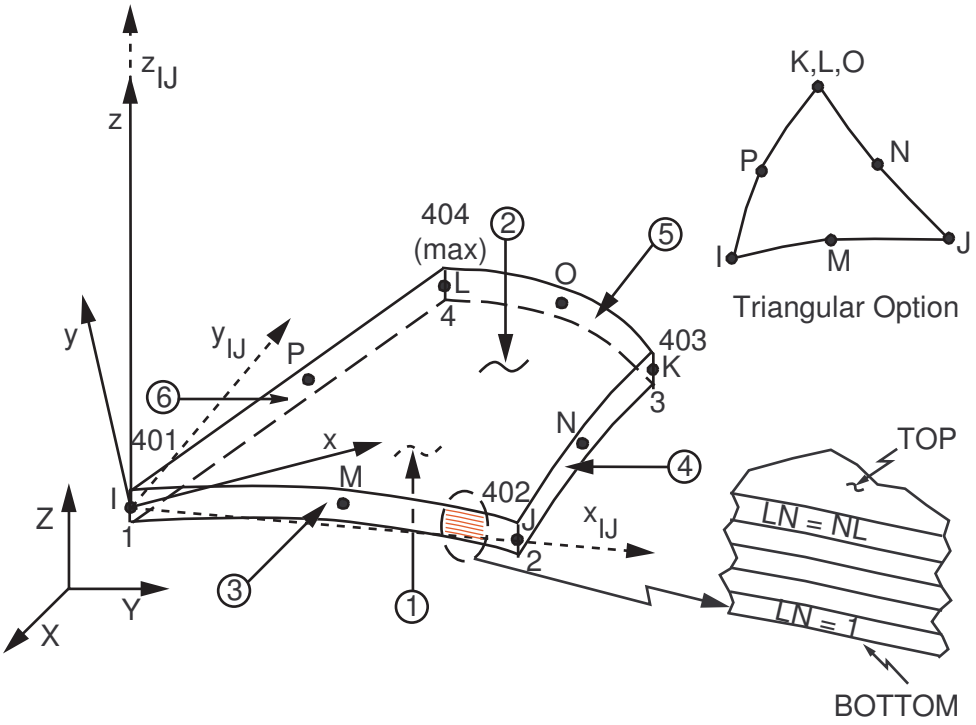
MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SHELL91 Element Description

SHELL91 may be used for layered applications of a structural shell model or for modeling thick sandwich structures. If applicable, SHELL99 is usually more efficient than SHELL91. Up to 100 different layers are permitted for applications with the sandwich option turned off. SHELL99 allows more layers, but no nonlinear materials. See SOLID46 for a description of a multi-layered solid.

The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. See SHELL91 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL91 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

LN = Layer Number

NL = Total Number of Layers

SHELL91 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL91 Geometry"*. The element is defined by eight nodes, layer thicknesses, layer material direction angles, and orthotropic material properties. Midside nodes may not be removed from this element. See Quadratic Elements (Midside

Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes. A triangular element may be formed by defining the same node number for nodes K, L and O.

When building a model using an element with *fewer* than three layers, SHELL91 is more efficient than SHELL99.

The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The local coordinate system for each layer is shown in *Figure 6, "SHELL91 Stress Output"*. In this local right-handed system, the layer x-axis is rotated an angle THETA (in degrees, specified as a real constant) from the element x-axis toward the element y-axis.

The total number of layers (up to 100) must be specified (NL). If the properties of the layers are symmetric about the midthickness of the element (LSYM = 1), only half the properties, up to and including those of the middle layer (if any), need to be entered. Otherwise (LSYM = 0), the properties of all layers should be entered.

Real constant ADMSUA is the added mass per unit area.

The material properties of each layer may be orthotropic in the plane of the element. The real constant MAT is used to define the layer material number instead of the element material number applied with the **MAT** command. MAT defaults to 1 if not input. The material X direction corresponds to the local layer xi direction.

Use the **BETAD** command to supply the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to supply the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

Each layer of the laminated shell element may have a variable thickness (TK). The thickness is assumed to vary bilinearly over the area of the layer, with the thickness input at the corner node locations. If a layer has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four corner thicknesses must be input using positive values. With nonlinear material properties, the thickness of any one layer may not exceed one-third of the total thickness of the element. The total thickness of each shell element *must* be less than twice the radius of curvature, and *should* be less than one-fifth the radius of curvature.

If the sandwich option is used (KEYOPT(9) = 1), the element uses "sandwich logic". This logic is specifically designed for sandwich construction with thin faceplates and a thick, relatively weak, core. The core is assumed to carry all of the transverse shear; the faceplates carry none. Conversely, the faceplates are intended to carry all (or almost all) of the bending load. Both faceplates are assumed to have the same number of layers, up to seven layers each. *Figure 2, "SHELL91 Sandwich Option - Before Deformation"* shows the element before deformation, and *Figure 3, "SHELL91 Sandwich Option - After Deformation"* shows the element after deformation. With the sandwich option, use of KEYOPT(5) = 1 is recommended since the best results are obtained at the midplane.

Figure 2 SHELL91 Sandwich Option - Before Deformation

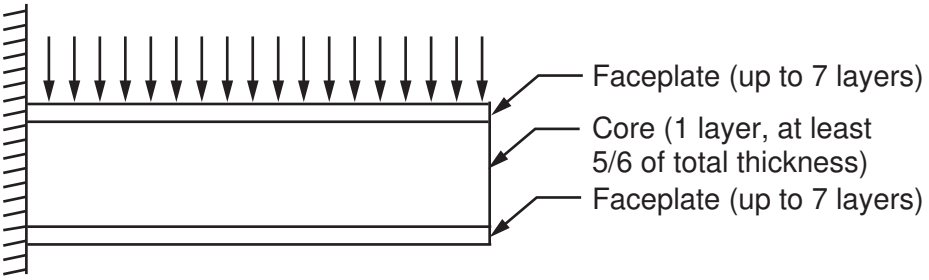
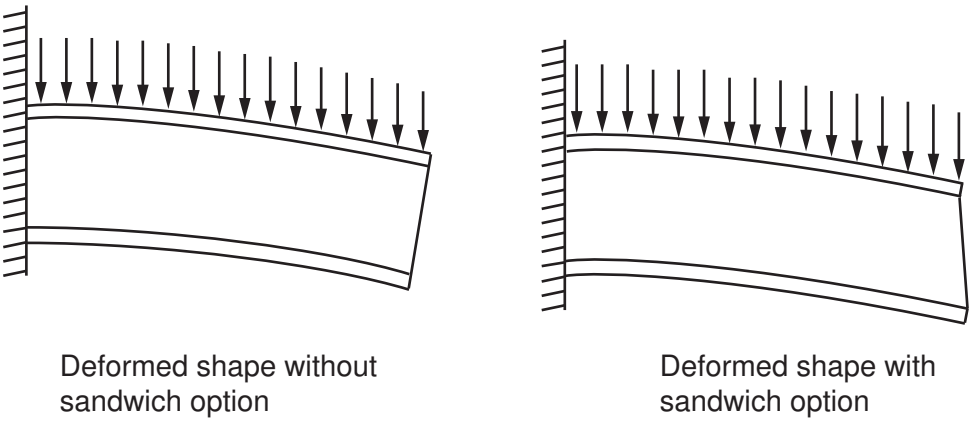
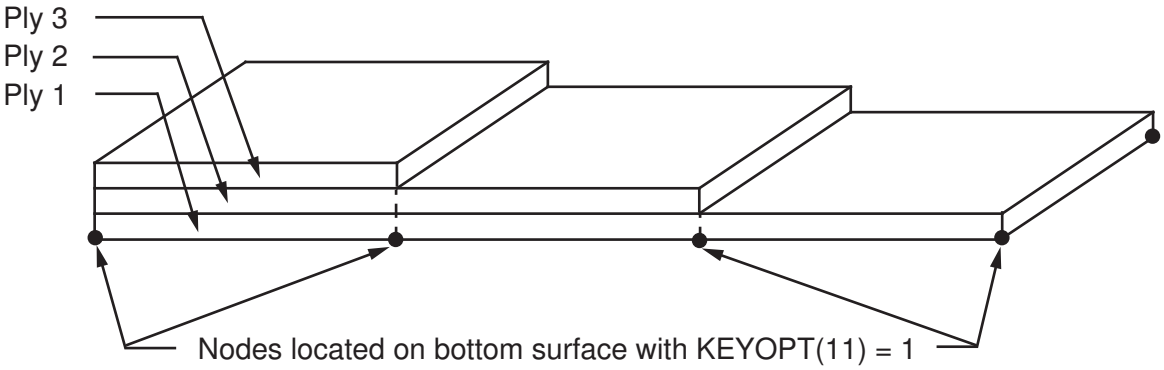


Figure 3 SHELL91 Sandwich Option - After Deformation



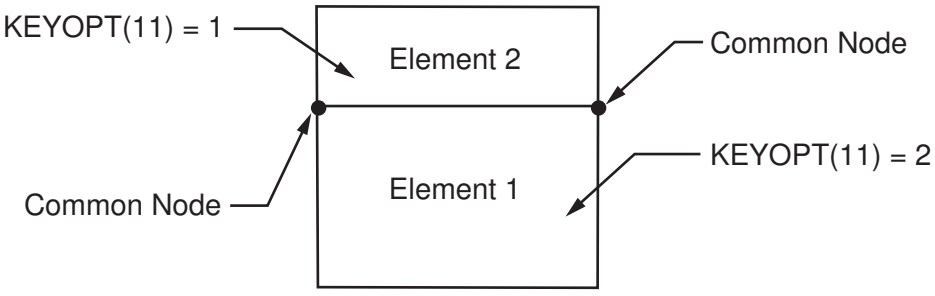
You can specify the nodes to be at the top, middle or bottom surface of the element. The choice is made through the node offset option, KEYOPT(11). This option is convenient, for example, when modeling laminated structures with ply drop-off where the location of the top or bottom surface may be better defined than the location of the midplane (as shown in *Figure 4, "SHELL91 Bottom Surface Nodes"*). Nodes in the nodal plane are shown at locations I through P in *Figure 1, "SHELL91 Geometry"* and by solid circles in *Figure 4, "SHELL91 Bottom Surface Nodes"*.

Figure 4 SHELL91 Bottom Surface Nodes



You can also define two elements that share the same nodes but have different settings of KEYOPT(11), as shown in *Figure 5, "SHELL91 Common Node Elements"*.

Figure 5 SHELL91 Common Node Elements



KEYOPT(10) controls printout of failure criteria. The failure criteria selection is input in the data table [**TB**], as described in *Table 2.2, "Orthotropic Material Failure Criteria Data"*. Three predefined criteria are available and up to six user-defined criteria may be entered with user subroutines. See Failure Criteria in the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the three predefined failure criteria. See *Guide to ANSYS User*

Programmable Features for an explanation of user subroutines. Failure criteria may also be computed in POST1 (using the **FC** commands). All references to failure criteria as part of element output data are based only on the **TB** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces at the nodal plane as shown by the circled numbers on *Figure 1, "SHELL91 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-404 maximum), as shown in *Figure 1, "SHELL91 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *SHELL91 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL91 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

Provide the following 12+(6*NL) constants:

NL, LSYM, (Blank), (Blank), (Blank), ADMSUA,
(Blank), (Blank), (Blank), (Blank), (Blank), (Blank),
MAT, THETA, TK(I), TK(J), TK(K), TK(L) for layer 1, etc. up to layer NL
See *Table 1, "SHELL91 Real Constants"* for a description of the real constants.

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, for layer 1, etc. up to layer *NL* (maximum number of material properties is 13**NL*)

Supply DAMP only once for the element (use **MAT** command to assign material property set). REFT may be supplied once for the element, or may be assigned on a per layer basis. See the discussion in the *Input Data* section for more details.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +z direction), face 2 (I-J-K-L) (top, in -z direction), face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer *NL* (4*(*NL*+1) maximum)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Adaptive descent
Swelling

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)

The maximum number of layers used by this element type for storage in the .ESAV and .0SAV files; default = 16. The first real constant (NL) must be no greater than the value you specify. The maximum number of layers may be no greater than 100.

KEYOPT(4)

Element coordinate system defined by:

0 --

No user subroutines to define element coordinate system

4 --

Element x-axis located by user subroutine USERAN

5 --

Element x-axis located by user subroutine USERAN and layer x-axes located by user subroutine USANLY

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(5)

Element output per layer:

0 --

Print average results at layer face farthest from element nodal plane

1 --

Print average results at layer middle

2 --

Print average results at layer top and bottom

3 --

Print results, including failure criterion, at layer top and bottom 4 integration points and averages

4 --

Print results at layer top and bottom 4 corner points and averages

KEYOPT(6)

Interlaminar shear stress output:

0 --
Do not print interlaminar shear stresses

1 --
Print interlaminar shear stresses

KEYOPT(8)
Storage of layer data:

0 --
Store data for bottom of bottom layer and top of top layer.

1 --
Store data for all layers

Caution

Volume of data may be excessive.

KEYOPT(9)
Thick sandwich option:

0 --
Do not use sandwich option

1 --
Use sandwich option

KEYOPT(10)
Failure criteria print summary:

0 --
Print summary of the maximum of all failure criteria

1 --
Print summary of all the failure criteria

KEYOPT(11)
Node offset option:

0 --
Nodes located at middle surface

1 --
Nodes located at bottom surface

2 --
Nodes located at top surface

For a complete discussion of failure criteria, please refer to *Section 2.2.2.12: Failure Criteria*.

Table 1 SHELL91 Real Constants

No.	Name	Description
Provide the following 12+(6*NL) constants:		
1	NL	Number of layers (100 maximum)
2	LSYM	Layer symmetry key
3 ... 5	(Blank)	
6	ADMSUA	Added mass/unit area

No.	Name	Description
7 ... 12	(Blank)	
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK(I)	Layer thickness at node I for layer 1
16	TK(J)	Layer thickness at node J for layer 1
17	TK(K)	Layer thickness at node K for layer 1
18	TK(L)	Layer thickness at node L for layer 1
19 ... 12+(6*NL)	MAT, THETA, etc.	Repeat MAT, THETA, TK(I), TK(J), TK(K), and TK(L) for each layer (up to NL layers)

SHELL91 Output Data

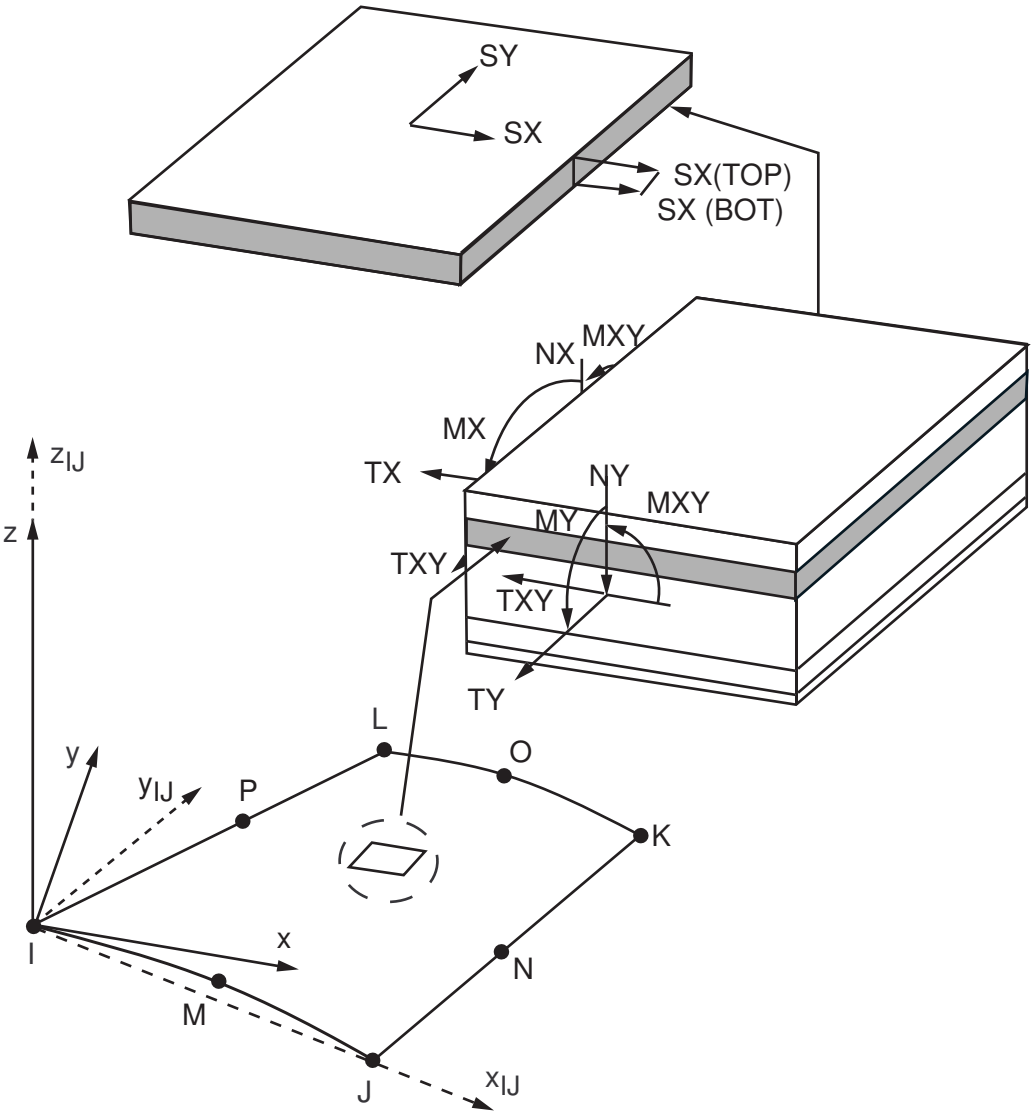
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL91 Element Output Definitions"*.

Several items are illustrated in *Figure 6, "SHELL91 Stress Output"*. The element stress directions correspond to the layer local coordinate directions.

Various layer printout options (KEYOPT(5)) are available. For integration point output, integration point 1 is nearest node I, 2 is nearest J, 3 is nearest K, and 4 is nearest L. Failure criterion output is evaluated only at the integration points (See the *Theory Reference for ANSYS and ANSYS Workbench*). After the layer printout, the in-plane forces and moments are listed for the entire element. These are shown in *Figure 6, "SHELL91 Stress Output"*. The forces and moments are calculated per unit length in the element coordinate system and are the combined sum for all layers. KEYOPT(8) controls the amount of data output on the results file for processing with the **LAYER** command. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 6 SHELL91 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL91 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y

Name	Definition	O	R
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*6*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T1, T2, T3, T4, T5, T6, T7, T8, T9, ...	Y	Y
LN	Layer number	Y	-
POS	Top, middle or bottom of layer	Y	-
LOC	Layer solution location	*1*	-
MAT	Material number of this layer	Y	-
S:X, Y, Z, XY, YZ, XZ	Stresses (in layer local coordinates)	Y	Y
S:1, 2, 3	Principal stress	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains (in layer local coordinates)	Y	Y
EPEL:1, 2, 3	Principal elastic strain	Y	-
EPEL:EQV	Equivalent elastic strain [*7*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain [*7*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains (in layer local coordinates)	*2*	*2*
EPPL:EQV	Equivalent plastic strain [*7*]	-	Y
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strain	Y	Y
EPCR:EQV	Equivalent creep strain [*7*]	-	Y
EPSW:	Swelling strain	-	Y
NL:EPEQ	Average equivalent plastic strain	*2*	*2*
NL:SRAT	Ratio of trial stress to stress on yield surface	*2*	*2*
NL:SEPL	Average equivalent stress from stress-strain curve	*2*	*2*
XC, YC, ZC	Global location of layer	Y	-
FC1, ..., FC6, FCMAX	Failure criterion values and maximum at each integration point, output only if KEYOPT(5) = 3	*3*	-
FC	Failure criterion number (FC1 to FC6, FCMAX)	*3*	Y
VALUE	Maximum value for this criterion (if value exceeds 9999.999, 9999.999 will be output)	*3*	Y
LN	Layer number where maximum occurs	*3*	Y
EPELF(X, Y, Z, XY, YZ, XZ)	Elastic strains (in layer local coordinates) causing the maximum value for this criterion in the element	*3*	Y
SF(X, Y, Z, XY, YZ, XZ)	Stresses (in layer local coordinates) causing the maximum value for this criterion in the element	*3*	Y
LAYERS	Interface location	*4*	-
ILSXZ	SXZ interlaminar shear stress	*4*	Y
ILSYZ	SYZ interlaminar shear stress	*4*	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	*4*	Y
ILSUM	Interlaminar shear stress vector sum	*4*	Y
LN1, LN2	Layer numbers which define location of maximum interlaminar shear stress (ILMAX)	*5*	Y

Name	Definition	O	R
ILMAX	Maximum interlaminar shear stress (occurs between LN1 and LN2)	*5*	Y
T(X, Y, XY)	Element total in-plane forces per unit length (in element coordinates)	Y	Y
M(X, Y, XY)	Element total moments per unit length (in element coordinates)	Y	Y
N(X, Y)	Out-of-plane element x and y shear forces	Y	Y

1. Layer solution location key:
   - Average - center location (if KEYOPT(5) = 0,1, or 2)
   - 1, 2, 3, or 4 - integration point location (if KEYOPT(5) = 3)
   - *NL* - corner node number (if KEYOPT(5) = 4)
2. Nonlinear solution (if KEYOPT(5) = 3 and the element has a nonlinear material)
3. Printed only if KEYOPT(5) = 3. Output of the elastic strains and stresses for each failure criterion and the maximum of all criteria (FCMAX).
4. Interlaminar stress solution (if KEYOPT(6) = 1)
5. Printed only if KEYOPT(6) $\neq$ 0 and significant shear stress is present.
6. Available only at centroid as a ***GET** item.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 3, "SHELL91 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SHELL91 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SHELL91 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J,...,L
: sequence number for data at nodes I,J,...,L

Table 3 SHELL91 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	(2*i)+7	(2*NL)+9
ILSYZ	SMISC	(2*i)+8	(2*NL)+10
ILSUM	NMISC	(2*i)+5	(2*NL)+7
ILANG	NMISC	(2*i)+6	(2*NL)+8

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	(2*NL)+11	(2*NL)+12	(2*NL)+13	(2*NL)+14
P2	SMISC	(2*NL)+15	(2*NL)+16	(2*NL)+17	(2*NL)+18
P3	SMISC	(2*NL)+20	(2*NL)+19	-	-
P4	SMISC	-	(2*NL)+22	(2*NL)+21	-
P5	SMISC	-	-	(2*NL)+24	(2*NL)+23
P6	SMISC	(2*NL)+25	-	-	(2*NL)+26

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TX	SMISC	1
TY	SMISC	2
TXY	SMISC	3
MX	SMISC	4
MY	SMISC	5
MXY	SMISC	6
NX	SMISC	7
NY	SMISC	8
FCMAX (over all layers)	NMISC	1
VALUE	NMISC	2
LN	NMISC	3
ILMAX	NMISC	4
LN1	NMISC	5
LN2	NMISC	6
ILSUM (between layers i and i+1)	NMISC	2i+7
ILANG	NMISC	2i+8
FCMAX (at layer i)	NMISC	(2*(NL+i))+7
VALUE (at layer i)	NMISC	(2*(NL+i))+8
FC	NMISC	(4*NL)+8+15(N-1)+1
VALUE	NMISC	(4*NL)+8+15(N-1)+2
LN	NMISC	(4*NL)+8+15(N-1)+3
EPELFX	NMISC	(4*NL)+8+15(N-1)+4
EPELFY	NMISC	(4*NL)+8+15(N-1)+5
EPELFZ	NMISC	(4*NL)+8+15(N-1)+6
EPELFXY	NMISC	(4*NL)+8+15(N-1)+7
EPELFYZ	NMISC	(4*NL)+8+15(N-1)+8
EPELFXZ	NMISC	(4*NL)+8+15(N-1)+9
SFX	NMISC	(4*NL)+8+15(N-1)+10
SFY	NMISC	(4*NL)+8+15(N-1)+11
SFZ	NMISC	(4*NL)+8+15(N-1)+12
SFXY	NMISC	(4*NL)+8+15(N-1)+13

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SFYZ	NMISC	(4*NL)+8+15(N-1)+14
SFXZ	NMISC	(4*NL)+8+15(N-1)+15

Note

The i in *Table 3, "SHELL91 Item and Sequence Numbers"* (where i = 1, 2, 3 ..., NL) refers to the layer number of the shell. NL is the maximum layer number as input for real constant NL ($1 \leq NL \leq 100$). N is the failure number as stored on the results file in compressed form, e.g., only those failure criteria requested will be written to the results file. For example, if only the maximum strain and the Tsai-Wu failure criteria are requested, the maximum strain criteria will be stored first (N = 1) and the Tsai-Wu failure criteria will be stored second (N = 2). In addition, if more than one criteria is requested, the maximum value over all criteria is stored last (N = 3 for this example).

SHELL91 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness layers are allowed only if a zero thickness is defined at all corners. For nonlinear materials, no layer can be thicker than 1/3 of the average element thickness.
- All inertial effects are assumed to be in the nodal plane, i.e., unbalanced laminate construction and offsets have no effect on the mass properties of the element.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers may not change between load steps.
- Under thermal loads, inaccurate results may be calculated for non-flat domains.
- The applied transverse thermal gradient is assumed to vary linearly through each layer and bilinearly over the element surface.
- The stress varies linearly through the thickness of each layer.
- Interlaminar transverse shear stresses are based on the assumption that no shear is carried at the top and bottom surfaces of an element. Further, these interlaminar shear stresses are only computed at the centroid and are not valid along the element boundaries. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- Only the lumped mass matrix is available. The mass matrix is assumed to act at the nodal plane.
- When you specify the sandwich option (KEYOPT(9) = 1), the following limits apply:
  - The ratio of the middle layer (core) thickness to the total thickness should be greater than 5/6, and must be greater than 5/7.
  - The ratio of the peak Young's modulus of the face over the Young's modulus of the core should be greater than 100 and must be greater than 4. Also, it should be less than 10,000 and must be less than 1,000,000.
  - For curved shells, the ratio of the radius of curvature to total thickness should be greater than 10 and must be greater than 8.
- For the node offset option (KEYOPT(11) $\neq$ 0):
  - Do not use shell-to-solid submodeling [**CBDOF**] or temperature interpolation [**BFINT**].

– Transverse shear stresses will not be valid if two elements share the same nodes but have different settings of KEYOPT(11) (such as in *Figure 5, "SHELL91 Common Node Elements"*). Also, POST1 nodal results in this case should be obtained from either the top or the bottom element, since nodal data averaging will not be valid if elements from both sides of the nodal plane are used.

SHELL91 Product Restrictions

There are no product-specific restrictions for this element.

SOLID92

3-D 10-Node Tetrahedral Structural Solid

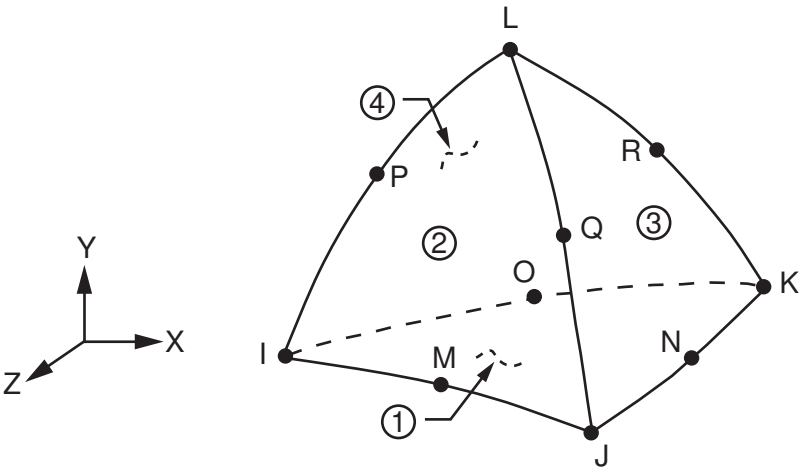
MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

SOLID92 Element Description

SOLID92 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). See SOLID95 for a 20-node brick shaped element.

The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. See SOLID92 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID92 Geometry



SOLID92 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID92 Geometry"*.

Beside the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID92 Geometry"*. Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

Initial state conditions previously handled via the **ISTRESS** command will be discontinued for this element. The **INISTATE** command will provide increased functionality, but only via the Current Technology elements (180,181, etc.). To continue using Initial State conditions in future versions of ANSYS, consider switching to the appropriate Current Technology element. For more information on setting Initial State values see the **INISTATE** command

and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. For more information on current -vs- legacy element technologies see *Section 2.17: Legacy vs. Current Element Technologies* in the *Elements Reference*

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID92 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID92 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Fluences --
FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P), FL(Q), FL(R)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO, CHABOCHE, HILL)
Creep (CREEP, RATE)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent
Initial stress import

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal stress printout

KEYOPT(6)
Extra surface output:

0 --
Basic element printout

4 --
Surface printout for faces with nonzero pressure

KEYOPT(9)
Initial stress subroutine option (available only through direct input of the **KEYOPT** command):

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)

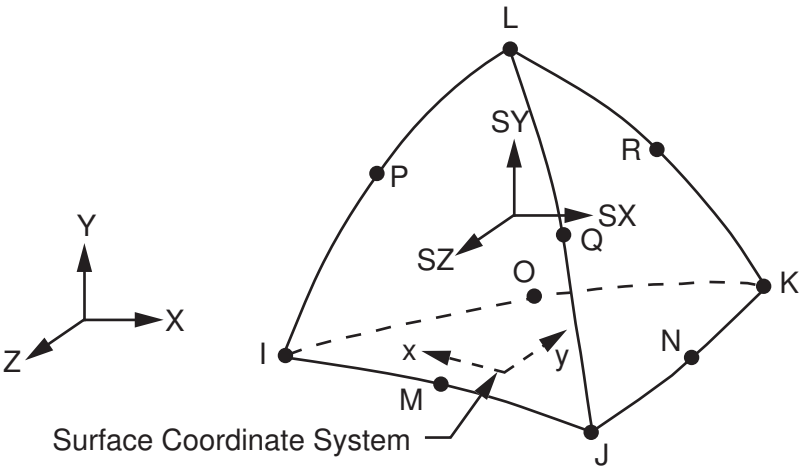
SOLID92 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID92 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID92 Stress Output"*. The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate system and are available for any face (KEYOPT(6)). The coordinate system for face J-I-K is shown in *Figure 2, "SOLID92 Stress Output"*. The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID92 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID92 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P), FL(Q), FL(R)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [*4*]	Y	-
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strains [*4*]	*1*	*1*
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*4*]	*1*	*1*

Name	Definition	O	R
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*4*]	*1*	*1*
EPSW:	Swelling strain	*1*	*1*
NL:EPEQ	Average equivalent plastic strain	*1*	*1*
NL:SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
NL:SEPL	Equivalent stress from stress-strain curve	*1*	*1*
NL:HPRES	Hydrostatic pressure	-	*1*
FACE	Face label	*2*	*2*
TRI	Nodes on this face	*2*	-
AREA	Face area	*2*	*2*
TEMP	Face average temperature	*2*	*2*
EPEL(X, Y, XY)	Surface elastic strains	*2*	*2*
PRES	Surface pressure	*2*	*2*
S(X, Y, XY)	Surface stresses	*2*	*2*
S(1, 2, 3)	Surface principal stresses	*2*	*2*
SINT	Surface stress intensity	*2*	*2*
SEQV	Surface equivalent stress	*2*	*2*
LOCI:X, Y, Z	Integration point locations	-	Y

1. Nonlinear solution (output if the element has a nonlinear material)
2. Surface output (if KEYOPT(6) = 4 and a nonzero pressure face)
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2 SOLID92 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Stress Solution	TEMP, SINT, SEQV, EPEL, S, EPPL, EPCR, EPSW, EPEQ, SRAT, SEPL, HPRES	*1*	-
Nodal Stress Solution	LOCATION, TEMP, SINT, SEQV, S	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each vertex node, if KEYOPT(5) = 2

Table 3, "SOLID92 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID92 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID92 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,R
sequence number for data at nodes I,J,...,R

Table 3 SOLID92 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-
S:1	NMISC	1	6	11	16	-
S:2	NMISC	2	7	12	17	-
S:3	NMISC	3	8	13	18	-
S:INT	NMISC	4	9	14	19	-
S:EQV	NMISC	5	10	15	20	-

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for the **ETABLE** command.

SOLID92 Assumptions and Restrictions

- The element must not have a zero volume. Elements may be numbered either as shown in *Figure 1, "SOLID92 Geometry"* or may have node L below the I-J-K plane.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for information about the use of midside nodes.

SOLID92 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- Fluence body loads are not applicable.
- The only special feature allowed is stress stiffening.

SHELL93

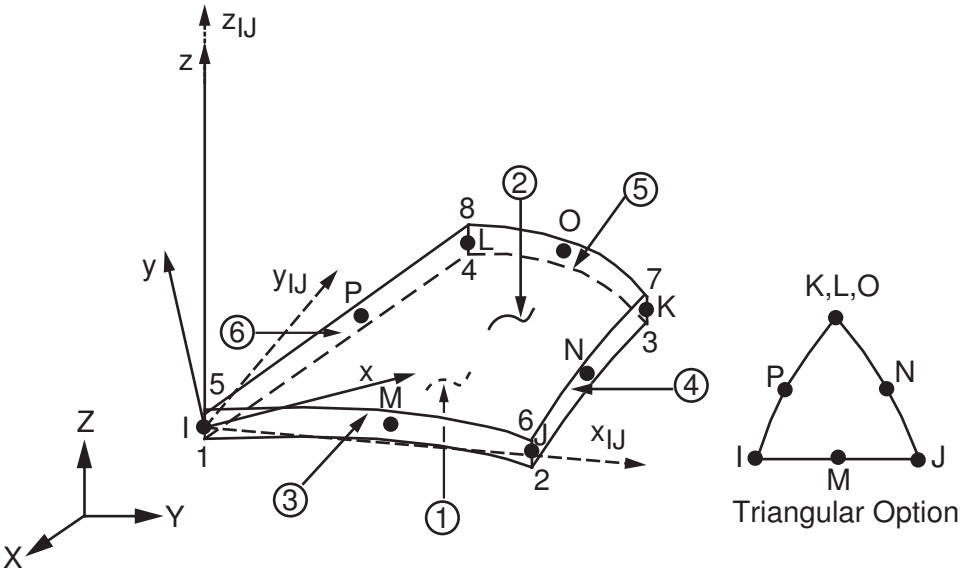
8-Node Structural Shell

MP ME ST PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

SHELL93 Element Description

SHELL93 is particularly well suited to model curved shells. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities. See SHELL93 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL93 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL93 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL93 Geometry"*. The element is defined by eight nodes, four thicknesses, and the orthotropic material properties. Midside nodes may not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for additional information about the use of midside nodes. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element x and y-axes are in the plane of the element. The x-axis may be rotated an angle THETA (in degrees) toward the y-axis.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the corner nodes. The thickness at the midside nodes is taken as the average of the corresponding corner nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input. If the total thickness of any shell element is greater than twice

the radius of curvature, ANSYS issues an error. If the total thickness is greater than one-fifth but less than twice the radius of curvature, ANSYS issues a warning. ADMSUA is the added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL93 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length. Temperatures may be input as element body loads at the "corner" locations (1-8) shown in *Figure 1, "SHELL93 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T1 is used for T1, T2, T3, and T4, while T2 (as input) is used for T5, T6, T7, and T8. For any other input pattern, unspecified temperatures default to TUNIF. Only the lumped mass matrix is available.

KEYOPT(8) = 2 is used to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

A summary of the element input is given in *SHELL93 Input Summary*. A general description of element input is given in *SHELL93 Input Summary*.

SHELL93 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, ADMSUA
See *Table 1, "SHELL93 Real Constants"* for a description of the real constants.

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction), face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperature --

T1, T2, T3, T4, T5, T6, T7, T8

Special Features

Plasticity
Creep
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent

Swelling

KEYOPT(4)

Element coordinate system defined by:

0 --

No user subroutine to define element coordinate system

4 --

Element x-axis located by user subroutine USERAN

Note

See the *Guide to ANSYS User Programmable Features* more information on user written subroutines

KEYOPT(5)

Extra stress output:

0 --

Basic element printout

1 --

Repeat basic solution for all integration points and top, middle and bottom surfaces

2 --

Nodal stress printout

KEYOPT(6)

Nonlinear integration point output:

0 --

Basic element printout

1 --

Nonlinear integration point printout

KEYOPT(8)

Specify data storage:

0 --

Store data for TOP and BOTTOM surfaces

2 --

Store data for TOP, BOTTOM, and MID surfaces.

Table 1 SHELL93 Real Constants

No.	Name	Description
1	TK(I)	Shell thickness at node I
2	TK(J)	Shell thickness at node J
3	TK(K)	Shell thickness at node K
4	TK(L)	Shell thickness at node L
5	THETA	Element X-axis rotation
6	ADMSUA	Added mass/unit area

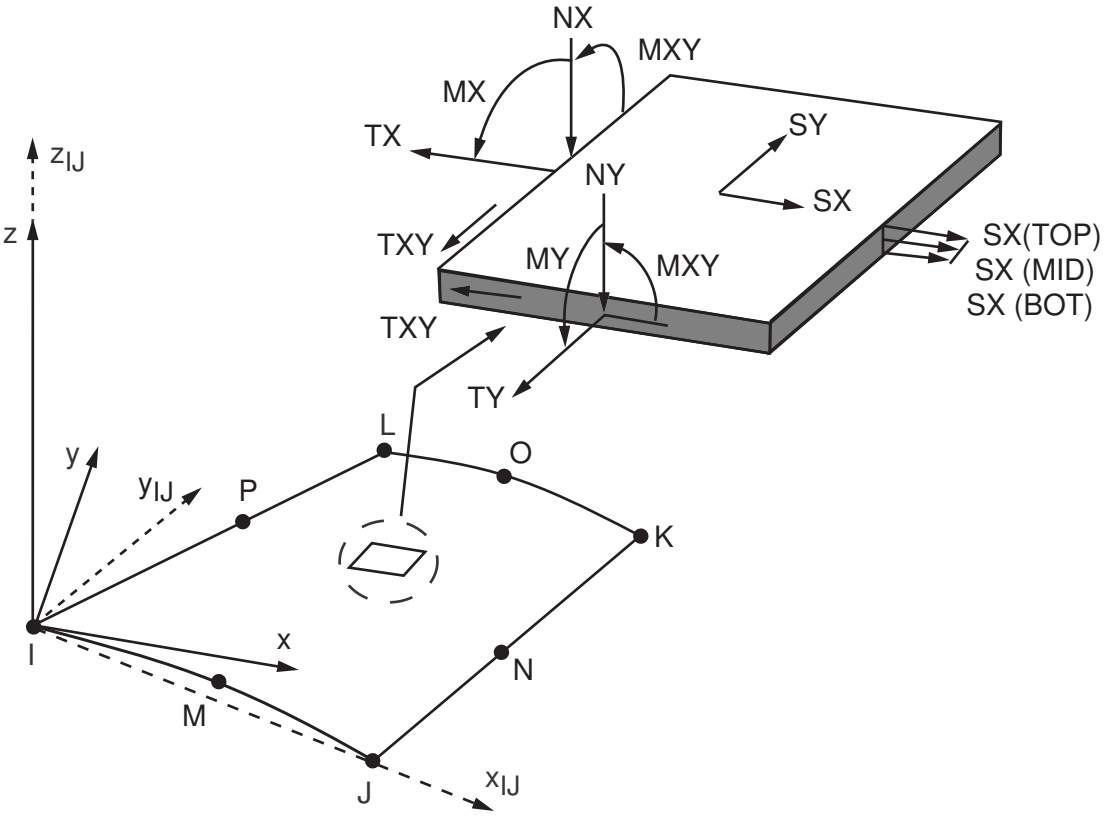
SHELL93 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL93 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL93 Stress Output"*. Printout includes the moments about the x face (MX), the moments about the y face (MY), and the twisting moment (MXY). The moments are calculated per unit length in the element coordinate system. The element stress directions and force resultants (NX, MX, TX, etc.) are parallel to the element coordinate system. The basic element printout is given at the center of the top of face IJKL, the element centroid, and at the center of the bottom of face IJKL. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL93 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL93 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
THICK	Average thickness	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes I,J,K,L; P2 at I,J,K,L; P3 at J,I; P4 at K,J; P5 at L,K; P6 at I,L	Y	Y
TEMP	T1, T2, T3, T4, T5, T6, T7, T8	Y	Y
LOC	TOP, MID, BOT, or integration point location	*1*	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*1*	*1*
S:1, 2, 3	Principal stress	*1*	*1*
S:INT	Stress intensity	*1*	*1*
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal stress	*1*	*1*
EPEL:EQV	Equivalent elastic strain [*4*]	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strain	Y	Y
EPTH:EQV	Equivalent thermal strain [*4*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	*2*	*2*
EPPL:EQV	Equivalent plastic strains [*4*]	-	*2*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains (X, Y, Z, XY, YZ, XZ)	*2*	*2*
EPCR:EQV	Equivalent creep strain [*4*]	-	*2*
EPSW:	Swelling strain	-	*2*
NL:EPEQ	Average equivalent plastic strain	*2*	*2*
NL:SRAT	Ratio of trial stress to stress on yield surface	*2*	*2*
NL:SEPL	Average equivalent stress from stress-strain curve	*2*	*2*
T(X, Y, XY)	In-plane element X, Y, and XY forces	Y	Y
M(X, Y, XY)	Element X, Y, and XY moments	Y	Y
N(X, Y)	Out-of-plane element X and Y shear forces	Y	Y

1. The stress solution item repeats for top, middle, and bottom surfaces (and for all integration points if KEYOPT(5) = 1)
2. Nonlinear solution (item output for top, middle, and bottom surfaces only if the element has a nonlinear material)
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 3 SHELL93 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Pt. Solution	EPPL, EPEQ, SRAT, SEPL	*1*	-

Description	Names of Items Output	O	R
Nodal Stress Solution	TEMP, S, SINT, SEQV	*2*	-

1. Output at each integration point, if the element has a nonlinear material and KEYOPT(6) = 1
2. Output at each corner node, if KEYOPT(5) = 2 (repeats each location)

Table 4, "SHELL93 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "SHELL93 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SHELL93 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,L
sequence number for data at nodes I,J,...,L

Table 4 SHELL93 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
TX	SMISC	1	-	-	-	-
TY	SMISC	2	-	-	-	-
TXY	SMISC	3	-	-	-	-
MX	SMISC	4	-	-	-	-
MY	SMISC	5	-	-	-	-
MXY	SMISC	6	-	-	-	-
NX	SMISC	7	-	-	-	-
NY	SMISC	8	-	-	-	-
THICK	NMISC	49	-	-	-	-
P1	SMISC	-	9	10	11	12
P2	SMISC	-	13	14	15	16
P3	SMISC	-	18	17	-	-
P4	SMISC	-	-	20	19	-
P5	SMISC	-	-	-	22	21
P6	SMISC	-	23	-	-	24
Top						
S:1	NMISC	-	1	6	11	16
S:2	NMISC	-	2	7	12	17
S:3	NMISC	-	3	8	13	18
S:INT	NMISC	-	4	9	14	19
S:EQV	NMISC	-	5	10	15	20
Bot						

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
S:1	NMISC	-	21	26	31	36
S:2	NMISC	-	22	27	32	37
S:3	NMISC	-	23	28	33	38
S:INT	NMISC	-	24	29	34	39
S:EQV	NMISC	-	25	30	35	40

SHELL93 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness.
- Shear deflections are included in this element.
- The out-of-plane (normal) stress for this element varies linearly through the thickness.
- The transverse shear stresses (SYZ and SXZ) are assumed to be constant through the thickness.
- The transverse shear strains are assumed to be small in a large strain analysis.
- This element may produce inaccurate stresses under thermal loads for doubly curved or warped domains.

SHELL93 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The special features allowed are stress stiffening and large deflection.
- KEYOPT(4) can only be set to 0 (default).

CIRCU94

Piezoelectric Circuit

MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

CIRCU94 Element Description

CIRCU94 is a circuit element for use in piezoelectric-circuit analyses. The element has two or three nodes to define the circuit component and one or two degrees of freedom to model the circuit response. The element may interface with the following piezoelectric elements:

PLANE13, KEYOPT(1) = 7 coupled-field quadrilateral solid
SOLID5, KEYOPT(1) = 0 or 3 coupled-field brick
SOLID98, KEYOPT(1) = 0 or 3 coupled-field tetrahedron
PLANE223, KEYOPT(1) = 1001, coupled-field 8-node quadrilateral
SOLID226, KEYOPT(1) = 1001, coupled-field 20-node brick
SOLID227, KEYOPT(1) = 1001, coupled-field 10-node tetrahedron

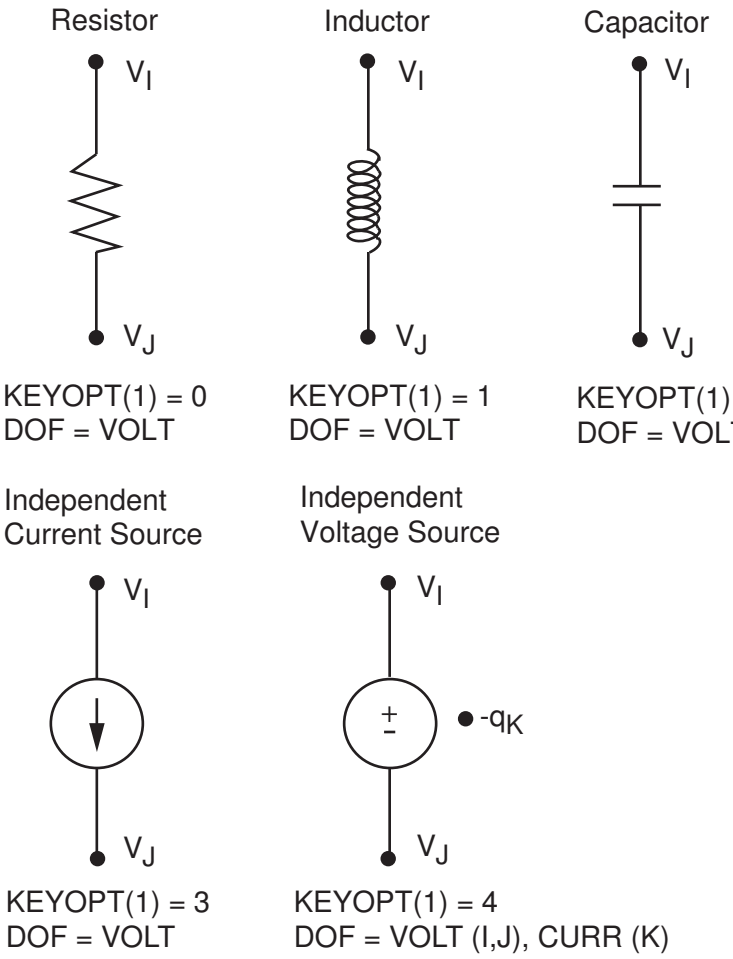
CIRCU94 is applicable to full harmonic and transient analyses. For these types of analyses, you can also use CIRCU94 as a general circuit element. See CIRCU94 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

CIRCU94 Input Data

The geometry, node definition, and degree of freedom options are shown in *Figure 1, "CIRCU94 Circuit Options"* . Active nodes I and J define the resistor, inductor, capacitor and independent current source. They are connected to the electric circuit. Active nodes I and J and a passive node K define the independent voltage source. The passive node is not connected to the electric circuit. It is associated with the CURR degree of freedom (which represents electric charge for this element).

KEYOPT(1) settings and the corresponding real constants define the circuit components. Real constant input is dependent on the element circuit option used. A summary of the element input options is given in *CIRCU94 Input Summary*. Real constants 15 (Graphical offset, GOFFST) and 16 (Element identification number, ID) are created for all components.

Figure 1 CIRCU94 Circuit Options



The independent current and voltage sources (KEYOPT(1) = 3 or 4) may be excited by constant load (transient) or constant amplitude load (harmonic), sinusoidal, pulse, exponential, or piecewise linear load functions as defined by KEYOPT(2); see *Figure 2, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.

The time-step size for a transient analysis is controlled by the **DELTIM** or **NSUBST** commands. The CIRCU94 element does not respond to automatic time stepping (**AUTOTS** command), but **AUTOTS** can be used as a mechanism for ramping the time step to its final value.

CIRCU94 is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

CIRCU94 Input Summary

Nodes
I, J, K

Degrees of Freedom
VOLT, CURR (charge) (see *Figure 1, "CIRCU94 Circuit Options"*)

Real Constants
Dependent on KEYOPT(1) and KEYOPT(2) settings. See *Table 1, "CIRCU94 Real Constants"* for details.

Material Properties
None

Surface Loads
None

Body Loads
See KEYOPT(2)

Special Features
This element works with the large deflection and stress stiffening capabilities of PLANE13, SOLID5, SOLID98, PLANE223, SOLID226, and SOLID227.

KEYOPT(1)
Circuit component type:

0 --
Resistor

1 --
Inductor

2 --
Capacitor

3 --
Independent Current Source

4 --
Independent Voltage Source

KEYOPT(2)
Body loads (only used for KEYOPT(1) = 3 and 4):

0 --
Constant load (transient) or constant amplitude load (harmonic)

1 --
Sinusoidal load

2 --
Pulse load

3 --
Exponential load

4 --
Piecewise Linear load

KEYOPT(6)
Electric charge reaction sign:

0 --
Negative

1 --
Positive

Table 1 CIRCU94 Real Constants

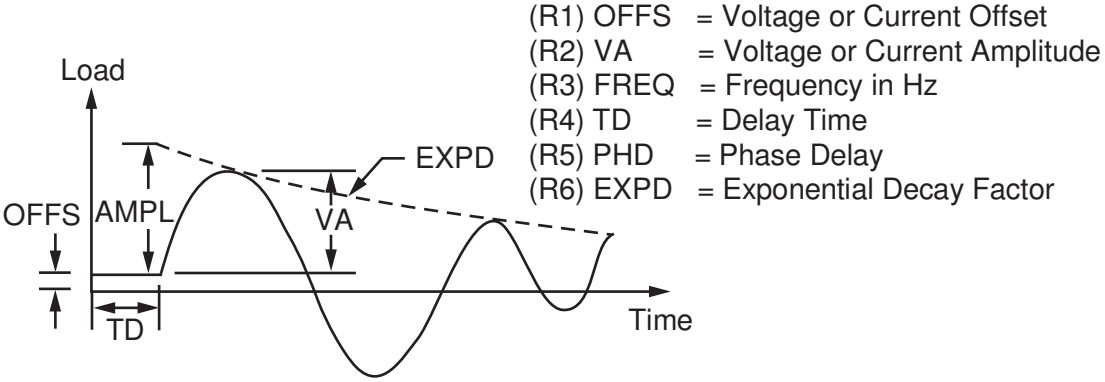
Circuit Component and Graphics Label	KEYOPT(1)	Real Constants
Resistor (R)	0	R1 = Resistance (RES)

Circuit Component and Graphics Label	KEYOPT(1)	Real Constants
Inductor (L)	1	R1 = Inductance (IND) R2 = Initial inductor current (ILO)
Capacitor (C)	2	R1 = Capacitance (CAP) R2 = Initial Capacitor Voltage (VCO)
Independent Current Source (I)	3	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0, see *Figure 2, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.
Independent Voltage Source (V)	4	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0, see *Figure 2, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.

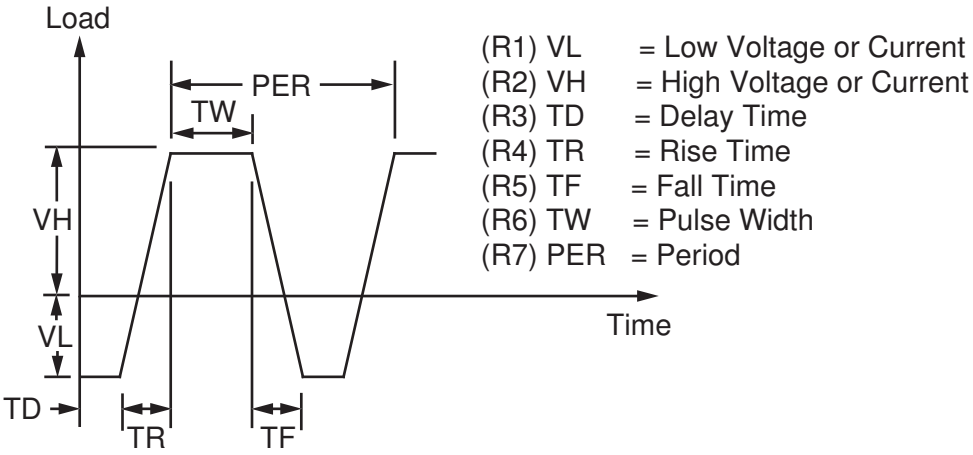
Note

For all above Circuit options, the GOFFST and ID real constants (numbers 15 and 16) are created by the Circuit Builder automatically:

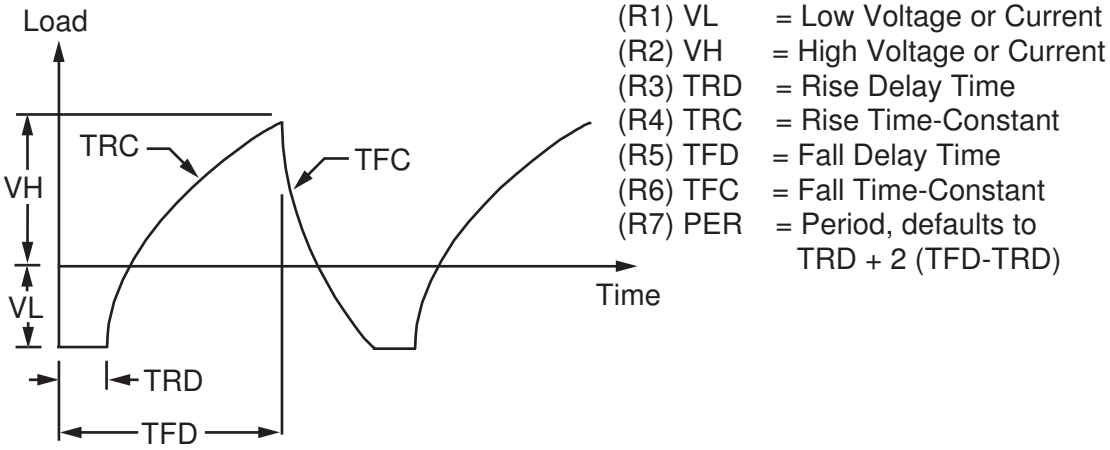
Figure 2 Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources



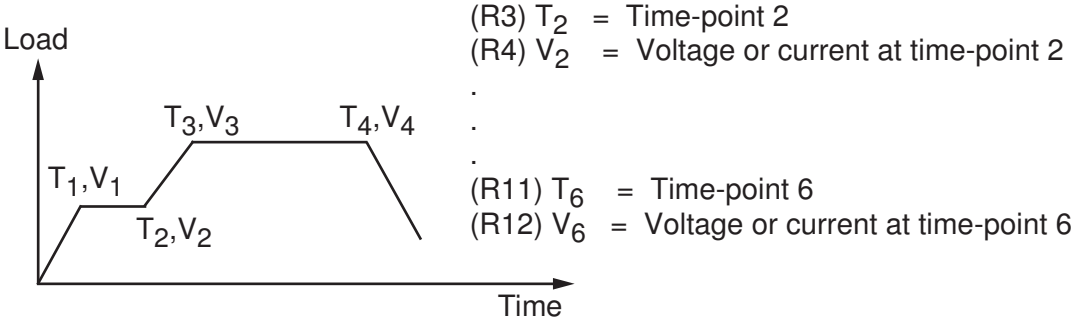
(a) Sinusoidal load, KEYOPT(2) = 1



(b) Pulse load, KEYOPT(2) = 2



(c) Exponential load, KEYOPT(2) = 3



(d) Piecewise linear load, KEYOPT(2) = 4

CIRCU94 Output Data

The element output for this element is dependent on the circuit option selected. *Table 2,"CIRCU94 Element Output Definitions"* summarizes the element output data.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CIRCU94 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Resistor			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
RES	Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
For KEYOPT(1) = 1: Inductor			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
IND	Inductance	Y	Y
IL0	Initial current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 2: Capacitor			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
CAP	Capacitance	Y	Y
VC0	Initial voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
	For KEYOPT(1) = 3: Independent Current Source		
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
CURRENT SOURCE	Real or imaginary component of applied current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
	For KEYOPT(1) = 4: Independent Voltage Source		
EL	Element Number	Y	Y
NODES	Nodes-I,J,K	Y	Y
VOLTAGE SOURCE	Real or imaginary component of applied voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y

Name	Definition	O	R
POWER	Power (loss if positive, output if negative)	Y	Y

Table 3, "CIRCU94 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CIRCU94 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 2, "CIRCU94 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 CIRCU94 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1
CURRENT	SMISC	2
POWER	NMISC	1
SOURCE (real)	NMISC	2
SOURCE (imaginary)	NMISC	3

CIRCU94 Assumptions and Restrictions

- CIRCU94 is applicable only to full harmonic and transient analyses. You cannot use CIRCU94 in a static analysis or in a transient analysis with time integration effects turned off (**TIMINT**,OFF).
- Only MKS units are allowed (**EMUNIT** command).
- Only the sparse solver is available for problems using the independent voltage source circuit option.
- This element may not be compatible with other elements with the VOLT degree of freedom. For example, it is not compatible with CIRCU124 or CIRCU125. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

CIRCU94 Product Restrictions

There are no product-specific restrictions for this element.

SOLID95

3-D 20-Node Structural Solid

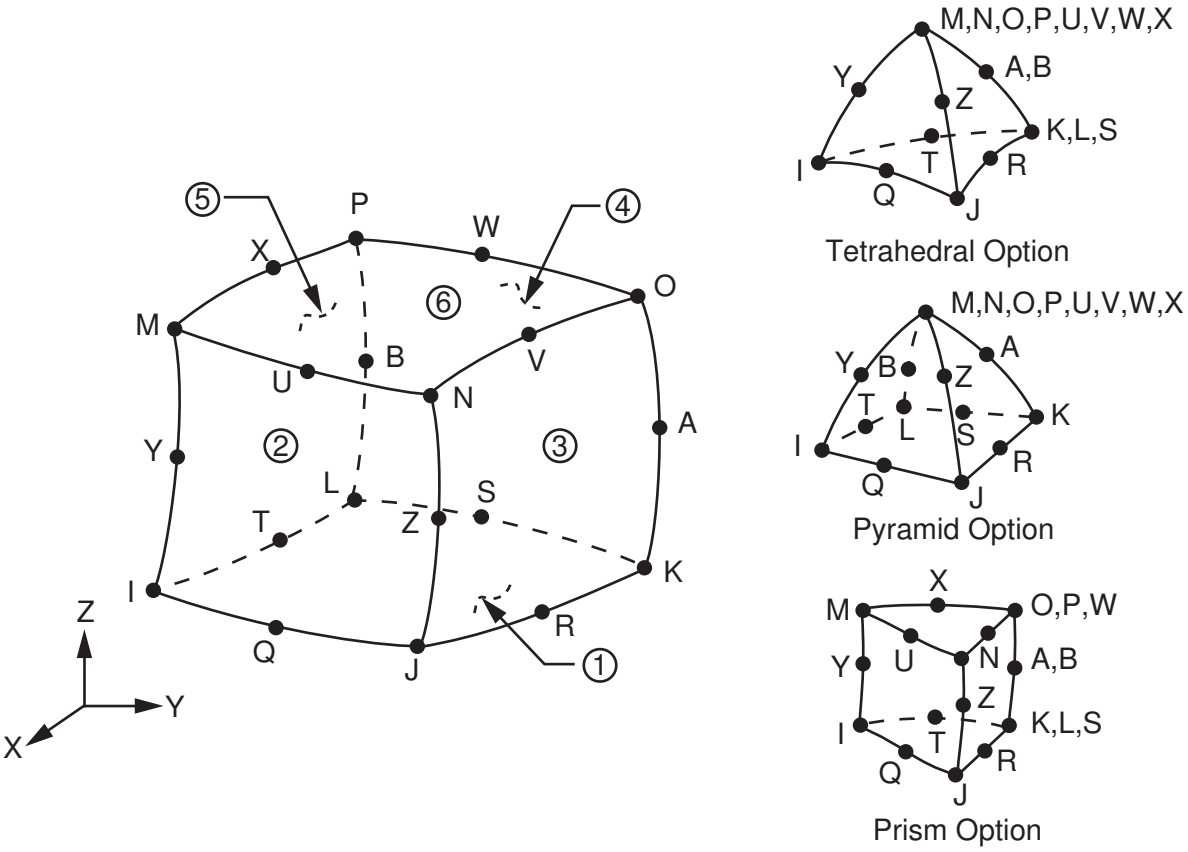
MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

SOLID95 Element Description

SOLID95 is a higher order version of the 3-D 8-node solid element SOLID45. It can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries.

The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. SOLID95 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities. Various printout options are also available. See SOLID95 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

Figure 1 SOLID95 Geometry



SOLID95 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID95 Geometry"*. A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID95 Geometry"*. A similar, but 10-node tetrahedron, element is SOLID92.

Besides the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID95 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

When using KEYOPT(1) = 1, this element acts in many regards as a shell element. Multiple elements through the thickness can be used to model a composite laminate in detail. Material properties are oriented the same way as for a shell element (using the plane through the midside nodes Y-Z-A-B) when you set KEYOPT(1) = 1. The element z-axis is normal to this plane and the element x-axis is determined by projecting the x-axis (set with **ESYS**) onto the midside node plane. If needed, the x-axis can be adjusted by using THETA, an optional real constant. THETA cannot be changed between load steps. In POST1, the command **LAYER**,1 is needed to get correct results in the material system, even though there is only one layer.

A lumped mass matrix formulation, which may be useful for certain analyses, may be obtained with **LUMPM**. While the consistent matrix gives good results for most applications, the lumped matrix may give better results with reduced analyses using Guyan reduction. The KEYOPT(5) and (6) parameters provide various element printout options (see *Section 2.2.2: Element Solution*).

Initial state conditions previously handled via the **ISTRESS** command will be discontinued for this element. The **INISTATE** command will provide increased functionality, but only via the Current Technology elements (180,181, etc.). To continue using Initial State conditions in future versions of ANSYS, consider switching to the appropriate Current Technology element. For more information on setting Initial State values see the **INISTATE** command and *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. For more information on current -vs- legacy element technologies see *Section 2.17: Legacy vs. Current Element Technologies* in the *Elements Reference*

You can include the effects of pressure load stiffness using **SOLCONTROL**,,,INCP. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID95 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID95 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

THETA - x-axis adjustment (used only when KEYOPT(1) = 1)

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),

face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), ..., T(Z), T(A), T(B)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO, CHABOCHE, HILL)
Creep (CREEP, RATE)
Swelling (SWELL)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent
Initial stress import

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)
Element coordinate system:

0 --
(default)

1 --
Orient material properties using plane created by midside nodes (Y-Z-A-B) with the z-axis normal to that plane and the x-axis (from **ESYS**) projected onto that plane.

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal stress printout

KEYOPT(6)
Extra surface output:

0 --
Basic element printout

1 --
Surface printout for face I-J-N-M

2 --
Surface printout for face I-J-N-M and face K-L-P-O (Surface printout valid for linear materials only)

3 --
Nonlinear printout at each integration point

4 --
Surface printout for faces with nonzero pressure

KEYOPT(9)
Initial stress subroutine option (available only through direct input of the **KEYOPT** command):

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)

KEYOPT(11)
Integration rule:

0 --
No reduced integration (default)

1 --
2 x 2 x 2 reduced integration option for brick shape

See Failure Criteria in the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the three predefined failure criteria. For a complete discussion of failure criteria, please refer to *Section 2.2.2.12: Failure Criteria*.

SOLID95 Output Data

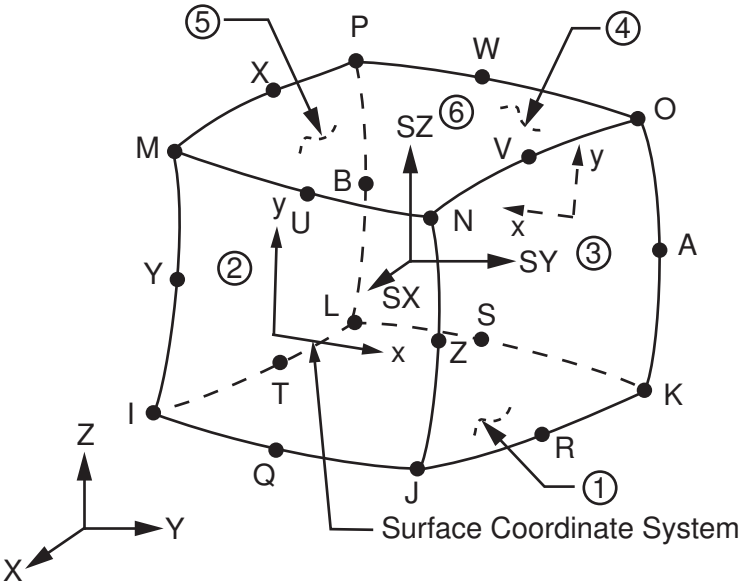
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID95 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID95 Stress Output"*.

The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate systems and are available for any face (KEYOPT(6)). The coordinate systems for faces I-J-N-M and K-L-P-O are shown in *Figure 2, "SOLID95 Stress Output"*. The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met. The SXY component is the in-plane shear stress on that face. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID95 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID95 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
CORNER NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*5*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [*6*]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strain [*6*]	*1*	*1*
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	*1*	*1*

Name	Definition	O	R
EPPL:EQV	Equivalent plastic strain [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strain [*6*]	*1*	*1*
EPSW:	Swelling strain	*1*	*1*
NL:EPEQ	Average equivalent plastic strain	*1*	*1*
NL:SRAT	Ratio of trial stress to stress on yield surface	*1*	*1*
NL:SEPL	Average equivalent stress from stress-strain curve	*1*	*1*
NL:HPRES	Hydrostatic pressure	-	*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
TEMP	Face average temperature	*2*	*2*
EPEL(X, Y, XY)	Surface elastic strains	*2*	*2*
PRES	Surface pressure	*2*	*2*
S(X, Y, XY)	Surface stresses (X-axis parallel to line defined by first two nodes which define the face)	*2*	*2*
S(1, 2, 3)	Surface principal stresses	*2*	*2*
SINT	Surface stress intensity	*2*	*2*
SEQV	Surface equivalent stress	*2*	*2*
FC1, ..., FC6, FCMAX	Failure criterion values and maximum at each integration point	*3*	-
FC	Failure criterion number (FC1 to FC6, FCMAX)	*4*	Y
VALUE	Maximum value for this criterion (if value exceeds 9999.999, 9999.999 will be printed)	*4*	Y
LN	Layer number where maximum occurs	*4*	Y
EPELF(X, Y, Z, XY, YZ, XZ)	Elastic strains (in layer local coordinates) causing the maximum value for this criterion in the element.	*4*	Y
SF(X, Y, Z, XY, YZ, XZ)	Stresses (in layer local coordinates) causing the maximum value for this criterion in the element.	*4*	Y
LOCI:X, Y, Z	Integration point locations	-	Y

1. Nonlinear solution (output only if the element has a nonlinear material)
2. Surface output (if KEYOPT(6) is 1, 2, or 4)
3. Output only if KEYOPT(1) = 1, KEYOPT (5) = 1, and failure criteria was specified (**TB**,FAIL)
4. Summary of failure criteria calculation. Output of the elastic strains and stresses for each failure criterion and the maximum of all criteria (FCMAX). Output only if KEYOPT(1) = 1 and failure criteria was specified (**TB**,FAIL).
5. Available only at centroid as a ***GET** item
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2 SOLID95 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nonlinear Integration Pt. Solution	EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR	*1*	-
Integration Point Stress Solution	TEMP, S, SINT, SEQV, EPEL	*2*	-

Description	Names of Items Output	O	R
Nodal Stress Solution	TEMP, S, SINT, SEQV, EPEL	*3*	-

1. Output at each integration point, if the element has a nonlinear material and KEYOPT(6) = 3
2. Output at each integration point, if KEYOPT(5) = 1
3. Output at each node, if KEYOPT(5) = 2

Table 3, "SOLID95 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID95 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "SOLID95 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I,J,...,P

Table 3 SOLID95 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40

The following output items are available only if KEYOPT(1) = 1 and the failure criteria information (**TB**,FAIL) was specified.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	Number
FCMAX	NMISC	61
VALUE	NMISC	62
FC	NMISC	62 + 15(N-1) + 1
VALUE	NMISC	62 + 15(N-1) + 2
LN (=1)	NMISC	62 + 15(N-1) + 3
EPELFX	NMISC	62 + 15(N-1) + 4
EPELFY	NMISC	62 + 15(N-1) + 5

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	Number
EPELFZ	NMISC	62 + 15(N-1) + 6
EPELFXY	NMISC	62 + 15(N-1) + 7
EPELFYZ	NMISC	62 + 15(N-1) + 8
EPELFXZ	NMISC	62 + 15(N-1) + 9
SFX	NMISC	62 + 15(N-1) + 10
SFY	NMISC	62 + 15(N-1) + 11
SFZ	NMISC	62 + 15(N-1) + 12
SFXY	NMISC	62 + 15(N-1) + 13
SFYZ	NMISC	62 + 15(N-1) + 14
SFXZ	NMISC	62 + 15(N-1) + 15

Note

N refers to the failure criterion number: N = 1 for the first failure criterion, N = 2 for the second failure criterion, and so on.

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for the **ETABLE** command.

SOLID95 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in *Figure 1, "SOLID95 Geometry"* or may have the planes IJKL and MNOP interchanged.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- If using **MSAVE**,ON, set KEYOPT(1) = 0. **MSAVE**,ON will not apply to any SOLID95 elements with KEYOPT(1) = 1.

SOLID95 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- The only special feature allowed is stress stiffening.
- KEYOPT(6) = 3 is not applicable.

SOLID96

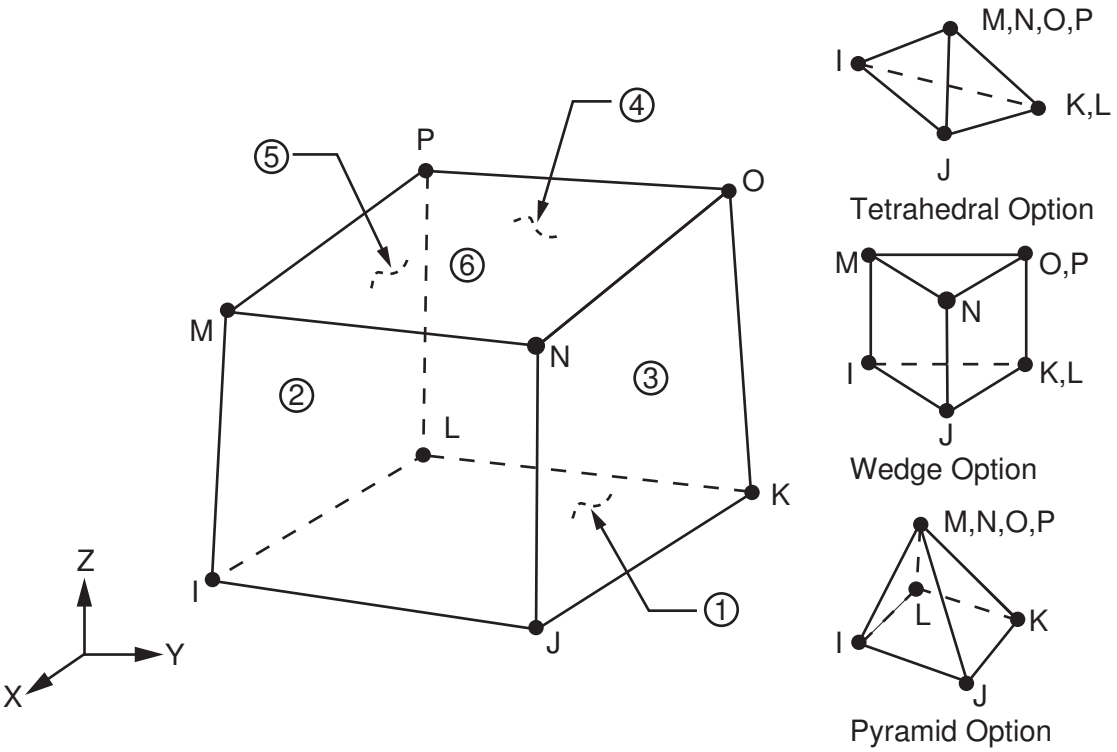
3-D Magnetic Scalar Solid

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID96 Element Description

SOLID96 has the capability of modeling 3-D magnetic fields. Scalar potential formulations (reduced (RSP), difference (DSP), or general (GSP)) are available [**MAGOPT**] for modeling magnetic fields in a static analysis. See SOLID96 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID96 Geometry



SOLID96 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID96 Geometry"*. The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID96 Geometry"*. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4\pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is available and is specified through the MURX, MURY, and MURZ material options.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Nonlinear magnetic B-H properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be

specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (MAG) and *VALUE* corresponds to the value (magnetic scalar potential). With the **F** command, the *Lab* variable corresponds to the force (FLUX) and *VALUE* corresponds to the value (magnetic flux).

Element loads are described in *Section 2.8: Node and Element Loads*. Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID96 Geometry"* using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Maxwell forces may be made available for a subsequent structural analysis with companion elements (**LDREAD** command).

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. In general, unspecified nodal values of temperature default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

Current for the magnetic scalar potential options are defined with the SOURC36 element, the command macro RACE, or through electromagnetic coupling. The various types of magnetic scalar potential solution options are defined with the **MAGOPT** command.

A summary of the element input is given in *SOLID96 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID96 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
MAG

Real Constants
None

Material Properties
MUZERO, MURX, MURY, MURZ, MGXX, MGYY, MGZZ plus BH data table (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Maxwell Force Flags --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T (M), T(N), T(O), T(P)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

EF --
EFX, EFY, EFZ. See *SOLID96 Assumptions and Restrictions*.

Special Features

Requires an iterative solution if nonlinear material properties are defined
Birth and death
Adaptive descent

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

SOLID96 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID96 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID96 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
MUX, MUY, MUZ	Magnetic permeability	*1*	*1*
H:X, Y, Z	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y, Z	Magnetic flux density components	*1*	*1*

Name	Definition	O	R
B:SUM	Vector magnitude of B	*1*	*1*
FMX	Maxwell magnetic force components (X, Y, Z)	*1*	-
FVW	Virtual work force components (X, Y, Z)	*1*	*1*
Combined (FJB or FMX) force components	Combined force components	-	*1*

1. The solution value is printed only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.

Table 2 SOLID96 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	*1*	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "SOLID96 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID96 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "SOLID96 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 SOLID96 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
FVWX	NMISC	4
FVWY	NMISC	5
FVWZ	NMISC	6
FVWSUM	NMISC	7

SOLID96 Assumptions and Restrictions

- When using SOLID96 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfils boundary conditions for the total field.
- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID96 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The difference magnetic scalar potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar, and general scalar potential options do not have this restriction.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.

SOLID96 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

SOLID97

3-D Magnetic Solid

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID97 Element Description

SOLID97 models 3-D magnetic fields. The element is defined by eight nodes, and has up to five degrees of freedom per node out of six defined DOFs; that is, the magnetic vector potential (AX, AY, AZ), the time-integrated electric potential (VOLT - classical formulation) or the electric potential (VOLT - solenoidal formulation), the electric current (CURR), and the electromotive force (EMF). SOLID97 is based on the magnetic vector potential formulation with the Coulomb gauge, and is applicable to the following low-frequency magnetic field analyses: magnetostatics, eddy currents (AC time harmonic and transient analyses), voltage forced magnetic fields (static, AC time harmonic and transient analyses), and electromagnetic-circuit coupled fields (static, AC time harmonic and transient analyses). The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. See SOLID97 in the *Theory Reference for ANSYS and ANSYS Workbench* for details about this element. Elements with similar capability are PLANE53, SOLID62 (but without voltage forced and magnetic-circuit coupled capability), and SOLID117.

Formulations

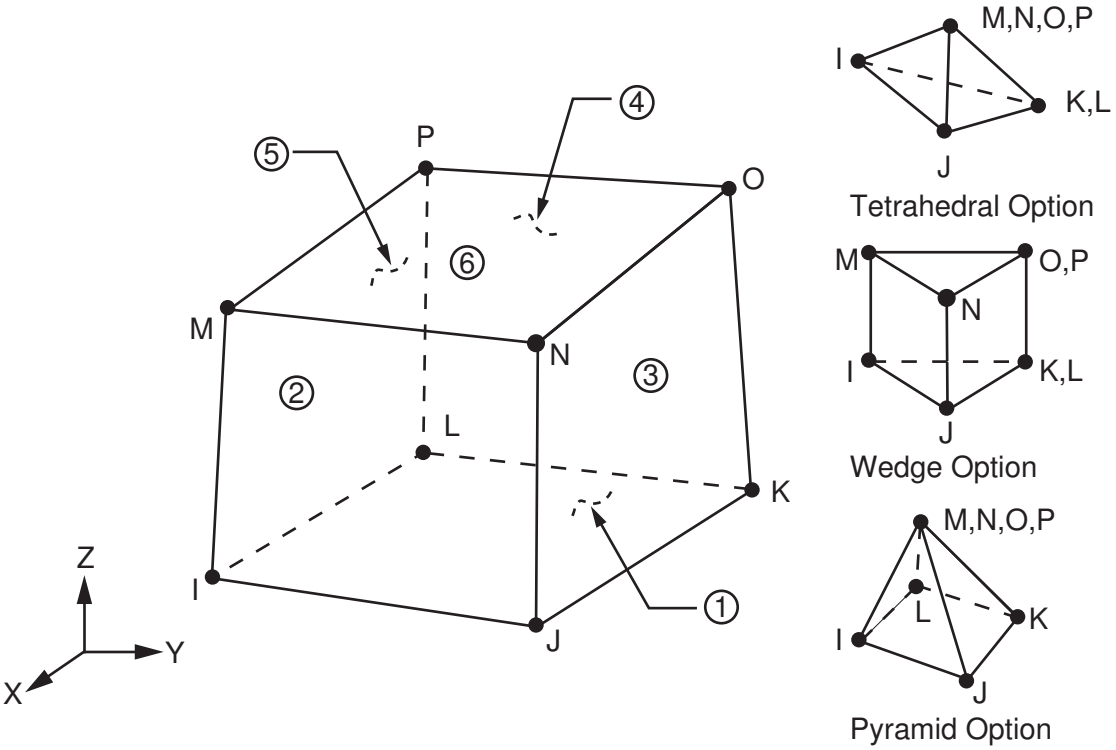
SOLID97 has two formulation options: classical and solenoidal. The classical formulation requires you to specify current density for current source loading. You must ensure that solenoidal conditions (div J = 0) are satisfied, otherwise an erroneous solution might develop. The solenoidal formulation automatically satisfies the solenoidal condition by directly solving for current (density) using a coupled current conduction and electromagnetic field solution. The solenoidal formulation is applicable to sources that are eddy current free (such as stranded coils).

Note

Both formulations may be used simultaneously, depending on the physics requirements of the model. See 3-D Nodal-Based Analyses (Static, Harmonic, and Transient) in the *Low-Frequency Electromagnetic Analysis Guide* for information on applying these formulations to different physics regions of a model.

The SOLID97 solenoidal option is compatible with CIRCU124, CIRCU125, and TRANS126 elements allowing circuit coupling. The nonlinear symmetric solenoidal formulation is applicable to static and transient analyses. The linear unsymmetric solenoidal formulation is applicable to harmonic analysis. For more information, see 3-D Circuit Coupled Solid Source Conductor in the *Coupled-Field Analysis Guide*

Figure 1 SOLID97 Geometry



SOLID97 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID97 Geometry"*. The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID97 Geometry"*. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4\pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is available and is specified through the MURX, MURY, and MURZ material options. Orthotropic resistivity is specified through RSVX, RSVY, and RSVZ material property labels.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Nonlinear magnetic B-H properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

When SOLID97 is used for voltage forced or magnetic-circuit coupled analyses, the following real constants apply for coils or massive conductors:

CARE
Coil cross-sectional area.

TURN
Total number of coil turns, required for stranded coil only. Defaults to 1.

VOLU
Modeled coil volume, required for stranded coil only.

DIRX, DIRY, DIRZ
x, y, and z components of a unit vector (in the element coordinate system) representing the direction of current. Required for a stranded coil only.

CSYM
Coil symmetry factor: CSYM*VOLU = total volume of the coil. Required for stranded coil only. Defaults to 1.

FILL
Coil fill factor, required for stranded coil only. Defaults to 1.

When velocity effects of a conducting body (KEYOPT(2) = 1) are considered, the following real constants apply:

VELOX, VELOY, VELOZ
Velocity components in the Global Cartesian Coordinate system X, Y, and Z direction respectively.

OMEGAX, OMEGAY, OMEGAZ
Angular (rotational) velocity (Hz, cycles/sec) about the Global Cartesian system X, Y, and Z-axes respectively, located at the pivot point location (XLOC, YLOC, ZLOC).

XLOC, YLOC, ZLOC
Global Cartesian coordinate point locations of the rotating body in the X, Y, and Z directions respectively.

Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (A_ and VOLT) and *VALUE* corresponds to the value (vector magnetic potential or the time-integrated electric potential (classical formulation) or the electric potential (solenoidal formulation). The electric potential may or may not be time integrated depending on the KEYOPT(1) selection. With the **F** command, the *Lab* variable corresponds to the force (CSG or Amps) and *VALUE* corresponds to the value (magnetic current segment or current).

Element loads are described in *Section 2.8: Node and Element Loads*. Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID97 Geometry"* using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Lorentz and Maxwell forces may be made available for a subsequent structural analysis with companion elements [**LDREAD**].

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. Source current density (classical formulation) and voltage body loads may be applied to an area or volume [**BFA** or **BFV**] or input as an element value [**BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. The vector components of the current density are with respect to the element coordinate system (see *SOLID97 Assumptions and Restrictions* for solenoidal restriction). Joule heating may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

A summary of the element input is given in *SOLID97 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID97 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
See KEYOPT(1)

Real Constants
None if KEYOPT(1) = 0 and KEYOPT(2) = 0.

For KEYOPT(1) = 2, 3, 5, or 6, and KEYOPT(2) = 0:

CARE, TURN, VOLU, DIRX, DIRY, DIRZ,
CSYM, FILL

For KEYOPT(1) = 4 and KEYOPT(2) = 0:

CARE - Coil cross-sectional area

For KEYOPT(1) = 0 or 1 and KEYOPT(2) = 1:

(blank), (blank), (blank), (blank), (blank), (blank),
(blank), (blank), VELOX, VELOY, VELOZ, OMEGAX,
OMEGAY OMEGAZ, XLOC, YLOC, ZLOC

See *Table 1, "SOLID97 Real Constants"* for a description of the real constants.

Material Properties
MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ, MGXX, MGYY, MGZZ plus BH data table (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Maxwell Force Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Magnetic-Circuit Interface Flags, if KEYOPT(1) = 4:

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

Source Current Density, if KEYOPT(1) = 0 or 1: (See *SOLID97 Assumptions and Restrictions* for solenoidal restriction)

JSX(I), JSY(I), JSZ(I), PHASE(I), JSX(J), JSY(J), JSZ(J), PHASE(J), JSX(K), JSY(K), JSZ(K), PHASE(K),
JSX(L), JSY(L), JSZ(L), PHASE(L), JSX(M), JSY(M), JSZ(M), PHASE(M), JSX(N), JSY(N), JSZ(N), PHASE(N),
JSX(O), JSY(O), JSZ(O), PHASE(O), JSX(P), JSY(P), JSZ(P), PHASE(P)

Voltage Loading, if KEYOPT(1) = 2:

VLTG(I), PHASE(I), VLTG(J), PHASE(J), VLTG(K), PHASE(K), VLTG(L), PHASE(L), VLTG(M), PHASE(M), VLTG(N), PHASE(N), VLTG(O), PHASE(O), VLTG(P), PHASE(P)

EF --
EFX, EFY, EFZ. See *SOLID97 Assumptions and Restrictions*.

Special Features

Requires an iterative solution if nonlinear material properties are defined
Birth and death
Adaptive descent

KEYOPT(1)
Element degrees of freedom and formulation selection:

Classical Formulation

0 --
AX, AY, AZ degrees of freedom: static domain, source domain

1 --
AX, AY, AZ, VOLT degrees of freedom: eddy current domain, velocity effect domain

2 --
AX, AY, AZ, CURR degrees of freedom: voltage-fed stranded coil

3 --
AX, AY, AZ, CURR, EMF degrees of freedom: circuit-coupled stranded coil

4 --
AX, AY, AZ, VOLT, CURR degrees of freedom: circuit-coupled massive conductor

Solenoidal Formulation

5 --
AX, AY, AZ, VOLT degrees of freedom: nonlinear symmetric solenoidal formulation applicable to static and transient analyses

6 --
AX, AY, AZ, VOLT degrees of freedom: linear unsymmetric solenoidal formulation applicable to harmonic analyses

Note

For KEYOPT(1) = 1 and 4, the VOLT degree of freedom is time integrated (classical formulation). For KEYOPT(1) = 5 and 6, the VOLT degree of freedom is not time integrated (solenoidal formulation).

KEYOPT(2)
Element conventional velocity:

0 --
Velocity effects ignored

1 --
Conventional velocity formulation (not available if KEYOPT(1) = 2, 3, or 4)

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

Table 1 SOLID97 Real Constants

No.	Name	Description
KEYOPT(1) = 2, 3, 5, 6 and KEYOPT(2) = 0		
1	CARE	Coil cross-sectional area
2	TURN	Total number of coil turns
3	VOLU	Modeled coil volume
4	DIRX	Directional cosine for X component of the current
5	DIRY	Directional cosine for Y component of the current
6	DIRZ	Directional cosine for Z component of the current
7	CSYM	Coil symmetry factor
8	FILL	Coil fill factor
KEYOPT(1) = 0 or 1 and KEYOPT(2) = 1		
1, ..., 8	(blank)	Unused for these settings
9, 10, 11	VELOX, VELOY, VELOZ	Velocity specification in the x, y, and z directions
12, 13, 14	OMEGAX, OMEGAY, OMEGAZ	Angular velocity about the X, Y, and Z axes
15, 16, 17	XLOC, YLOC, ZLOC	Pivot point location (x, y, z)

SOLID97 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 2, "SOLID97 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SOLID97 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Input temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
MUX, MUY, MUZ	Magnetic secant permeability	*1*	*1*
H:X, Y, Z	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y, Z	Magnetic flux density components	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
JS:X, Y, Z	Source current density components in the global Cartesian coordinate system, valid for static analysis only	*1*	*1*
JT(X, Y, Z)	Total current density components in the global Cartesian coordinate system.	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
FJB(X, Y, Z)	Lorentz magnetic force components	*1*	-
FMX(X, Y, Z)	Maxwell magnetic force components	*1*	-
FVW(X, Y, Z)	Virtual work force components	*1*	*1*
Combined (FJB or FMX) force components	Combined (FJB or FMX) force components	-	*1*
ERES	Element resistance value (for stranded coils only)	-	*1*
EIND	Element inductance value (for stranded coils only)	-	*1*
DMUXX, DMUYY, DMUZZ	Differential permeability	*1*	*1*
V:X, Y, Z	Velocity components	*1*	*1*
V:SUM	Vector magnitude of V	*1*	-
MRE	Magnetic Reynolds number	*1*	*1*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For harmonic analysis, joule losses (JHEAT) and forces (FJB(X, Y, Z), FMX(X, Y, Z), FVW(X, Y, Z)) represent time-average values. These values are stored in both the "Real" and "Imaginary" data sets. The macros **POWERH** and **FMAGSUM** can be used to retrieve this data.

Inductance values (EIND) obtained for KEYOPT(1) = 2, 3, or 4 are only valid under the following conditions: the problem is linear (constant permeability), there are no permanent magnets in the model, and only a single coil exists in the model.

2. Available only at centroid as a ***GET** item.

Table 3 SOLID97 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	*1*	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	*2*	-

1. Output at each integration point, if (KEYOPT(5) = 1)
2. Output at each corner node, if (KEYOPT(5) = 2)

Table 4, "SOLID97 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "SOLID97 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 2, "SOLID97 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 4 SOLID97 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JSX	SMISC	1
JSY	SMISC	2
JSZ	SMISC	3
JSSUM	SMISC	4
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
FVWX	NMISC	4
FVWY	NMISC	5
FVWZ	NMISC	6
FVWSUM	NMISC	7
JTX	NMISC	12
JTY	NMISC	13
JTZ	NMISC	14
JTSUM	NMISC	15
ERES	NMISC	16
EIND	NMISC	17
DMUXX	NMISC	18
DMUYY	NMISC	19

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DMUZZ	NMISC	20
VX	NMISC	21
VY	NMISC	22
VZ	NMISC	23
MRE	NMISC	28

SOLID97 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID97 Geometry"* or may have the planes IJKL and MNOP interchanged.
- The continuity equation must be satisfied for a proper electromagnetic analysis as explained in the *Theory Reference for ANSYS and ANSYS Workbench*. For this reason the source current density, JS, must be solenoidal (that is, $\nabla \cdot JS = 0$). You should verify that this condition is satisfied when prescribing the source current density load. If this condition is not satisfied SOLID97 can produce erroneous solutions without warning. Refer to Source Current Density (JS) in the *Low-Frequency Electromagnetic Analysis Guide* for information on how to obtain solenoidal currents when the source current density is not constant. To have ANSYS compute the current density for voltage or circuit coupled problems, apply the solenoidal formulation (KEYOPT(1) = 5 or 6).
- For models containing materials with different permeabilities, the 3-D nodal-based vector potential formulation (either static or time-dependent) is not recommended. The solution has been found to be inaccurate when the normal component of the vector potential is significant at the interface between elements of different permeability. To obtain the normal component of the vector potential in postprocessing, issue **PLVECT**,A or **PRVECT**,A in a rotated coordinate system [**RSYS**] that orients one of the vector potential components normal to the material interface.
- Current density loading (**BFE**,,JS) is only valid for the AX, AY, AZ option (KEYOPT(1) = 0). For the AX, AY, AZ, VOLT option (KEYOPT(1) = 1, 5, or 6) use **F**,,AMPS. Solenoidal loading is recommended. For more information, see 3-D Magnetostatics and Fundamentals of Edge-Based Analysis and 3-D Nodal-Based Analyses (Static, Harmonic, and Transient) in the *Low-Frequency Electromagnetic Analysis Guide*.
- When this element does not have the VOLT degree of freedom, for a harmonic or transient analysis, it acts as a stranded conductor.
- Permanent magnets are not permitted in a harmonic analysis.
- You cannot use this element in a nonlinear harmonic analysis.
- The VOLT degree of freedom (KEYOPT(1) = 1) is required in all non-source regions with a specified nonzero resistivity. This allows eddy currents to be computed.
- For source conducting regions (RSVX $\neq$ 0), current loading should be applied as nodal loads (AMPS) using the solenoidal formulation. Current density loading (JS) is allowed (classical formulation), but solenoidal loading is recommended. Node coupling of the VOLT DOF may be required at symmetry planes and locations where the current is applied.
- The ANSYS product does not support the analysis of coupled velocity and circuit effects.
- For voltage forced magnetic field (KEYOPT(1) = 2) and circuit coupled problems (KEYOPT(1) = 3,4), note the following additional restrictions:
  - Only MKS units are allowed.

  - The permeability and conductivity are isotropic and constant.
  - The element coordinate system is used for specifying the current direction vector (DIRX, DIRY, DIRZ) for a stranded coil. Also, the cross sectional area of the stranded coil should not change.
  - For (KEYOPT(1) = 2 or 3), all CURR degrees of freedom in a coil region must be coupled (**CP** command), and all EMF degrees of freedom in a coil region must be coupled.
  - For (KEYOPT(1) = 4), all CURR degrees of freedom on the input face and output face of a massive conductor must be coupled.
- For circuit coupled transient analyses, use THETA = 1.0, the default value, on the **TINTP** command to specify the backward Euler method. For more information, refer to the *Theory Reference for ANSYS and ANSYS Workbench*, as well as the description of the **TINTP** command in the *Commands Reference*.
- For velocity effects (KEYOPT(2) = 1), note the following restrictions:
  - Velocity effects are valid only for the AX, AY, AZ, VOLT DOF option.
  - Velocity effects cannot be included in a static analysis. To simulate a static analysis, execute a harmonic analysis at a very low frequency and retrieve the "real" results for the solution.
  - Velocity effects are available only in a linear analysis.
  - Isotropic resistivity.
  - Solution accuracy may degrade if the element magnetic Reynolds number is much greater than 1.0. (See the discussion of magnetic fields in the *Low-Frequency Electromagnetic Analysis Guide*.)
- If (KEYOPT(1) = 2, 3,4, or 6) or (KEYOPT(2) $\geq$ 1), unsymmetric matrices are produced.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- The solenoidal formulations do not model eddy current effects.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.

SOLID97 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

SOLID98

Tetrahedral Coupled-Field Solid

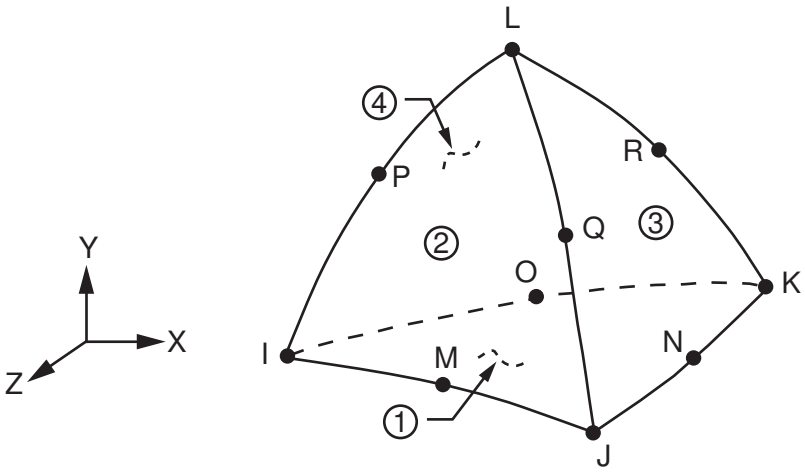
MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID98 Element Description

SOLID98 is a 10-node tetrahedral version of the 8-node SOLID5 element. The element has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). When used in structural and piezoelectric analyses, SOLID98 has large deflection and stress stiffening capabilities.

The element is defined by ten nodes with up to six degrees of freedom at each node (see KEYOPT(1)). See SOLID98 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. The 3-D magnetic, thermal, electric, piezoelectric, and structural field capability is similar to that described for SOLID5.

Figure 1 SOLID98 Geometry



SOLID98 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID98 Geometry"*. The element input data is essentially the same as for SOLID5 except that there are 10 nodes instead of 8.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (UX, UY, UZ, TEMP, VOLT, MAG) and *VALUE* corresponds to the value (displacements, temperature, voltage, scalar magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (FX, FY, FZ, HEAT, AMPS, FLUX) and *VALUE* corresponds to the value (force, heat flow, current or charge, magnetic flux). Nonlinear magnetic B-H, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in *Section 2.5: Data Tables - Implicit Analysis*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID98 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load

commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

The body loads; temperature, heat generation rate and magnetic virtual displacement may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 0, 1 or 8), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal values of temperatures and heat generation rate default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) is applied in subsequent iterations as heat generation rate loading.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

A summary of the element input is given in *SOLID98 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID98 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

UX, UY, UZ, TEMP, VOLT, MAG if KEYOPT(1) = 0
TEMP, VOLT, MAG if KEYOPT(1) = 1
UX, UY, UZ if KEYOPT(1) = 2
UX, UY, UZ, VOLT if KEYOPT(1) = 3
TEMP if KEYOPT(1) = 8
VOLT if KEYOPT(1) = 9
MAG if KEYOPT(1) = 10

Real Constants

None

Material Properties

EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP, KXX, KYY, KZZ, C,
ENTH, MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ,
MGXX, MGYY, MGZZ, PERX, PERY, PERZ, plus BH, ANEL, and PIEZ data tables (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags -- face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P), VD(Q), VD(R)

EF --
EFX, EFY, EFZ. See *SOLID98 Assumptions and Restrictions*.

Special Features

Requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
Large deflections
Stress stiffening
Birth and death
Adaptive descent

KEYOPT(1)
Degree of freedom selection:

0 --
UX, UY, UZ, TEMP, VOLT, MAG

1 --
TEMP, VOLT, MAG

2 --
UX, UY, UZ

3 --
UX, UY, UZ, VOLT

8 --
TEMP

9 --
VOLT

10 --
MAG

KEYOPT(3)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(5)
Extra element output:

0 --
Basic element printout

2 --
Nodal stress or magnetic field printout

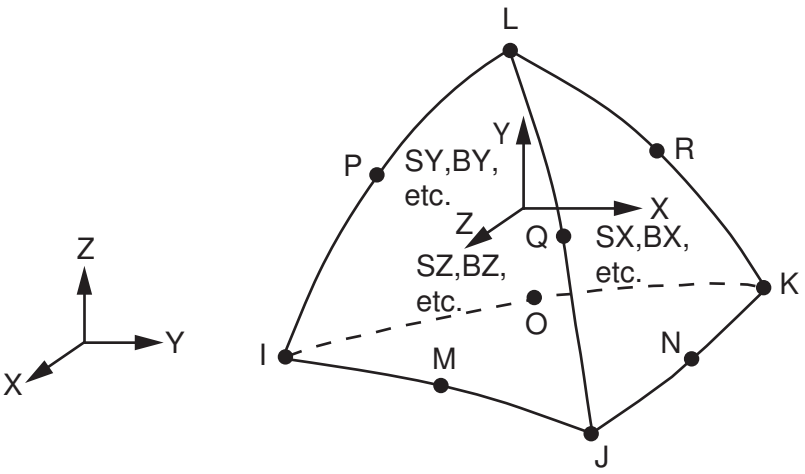
SOLID98 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID98 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID98 Element Output"*. The component output directions are parallel to the element coordinate system. The reaction forces, heat flow, current, and magnetic flux at the nodes can be printed with the **OUTPR** command. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID98 Element Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID98 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	Y	Y
TEMP(INPUT)	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
HGEN(INPUT)	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)	Y	-
S:X, Y, Z, XY, YZ, XZ	Stresses	*1*	*1*
S:1, 2, 3	Principal stresses	*1*	*1*
S:INT	Stress intensity	*1*	*1*

Name	Definition	O	R
S:EQV	Equivalent stress	*1*	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	*1*	*1*
EPEL:1, 2, 3	Principal elastic strains	*1*	-
EPEL:EQV	Equivalent elastic strains [*4*]	*1*	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*1*	*1*
EPTH:EQV	Equivalent thermal strain [*4*]	*1*	*1*
LOC	Output location (X, Y, Z)	*1*	*1*
MUX, MUY, MUZ	Magnetic permeability	*1*	*1*
H:X, Y, Z	Magnetic field intensity components	*1*	*1*
H:SUM	Vector magnitude of H	*1*	*1*
B:X, Y, Z	Magnetic flux density components	*1*	*1*
B:SUM	Vector magnitude of B	*1*	*1*
FJB	Lorentz magnetic force components (X, Y, Z)	*1*	-
FMX	Maxwell magnetic force components (X, Y, Z)	*1*	-
FVW	Virtual work force components (X, Y, Z)	*1*	*1*
Combined (FJB or FMX) force components	Combined (FJB or FMX) force components	-	*1*
EF:X, Y, Z	Electric field components	*1*	*1*
EF:SUM	Vector magnitude of EF	*1*	*1*
JS:X, Y, Z	Source current density components	*1*	*1*
JSSUM	Vector magnitude of JS	*1*	*1*
JHEAT:	Joule heat generation per unit volume	*1*	*1*
D:X, Y, Z	Electric flux density components	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
U(E, D, M)	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	*1*	*1*
TG:X, Y, Z	Thermal gradient components	*1*	*1*
TG:SUM	Vector magnitude of TG	*1*	*1*
TF:X, Y, Z	Thermal flux components	*1*	*1*
TF:SUM	Vector magnitude of TF (Heat flow rate/unit cross-section area)	*1*	*1*
FACE	Face label	*2*	*2*
AREA	Face area	*2*	*2*
NODES	Face nodes	*2*	-
HFILM	Film coefficient at each node of face	*2*	-
TBULK	Bulk temperature at each node of face	*2*	-
TAVG	Average face temperature	*2*	*2*
HEAT RATE	Heat flow rate across face by convection	*2*	*2*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*2*	-
HFLUX	Heat flux at each node of face	*2*	-
HFAVG	Average film coefficient of the face	*2*	*2*
TBAVG	Average face bulk temperature	-	*2*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*2*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Output only if a surface load is input.
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 2 SOLID98 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Stress Solution	LOC, SINT, SEQV, EPEL(X, Y, Z, XY, YZ, XZ), EPEL(1, 2, 3), S(X, Y, Z, XY, YZ, XZ), S(1, 2, 3)	*1*	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	*2*	-

1. Output at each vertex node, if KEYOPT(5) = 2 and structural DOF
2. Output at each vertex node, if KEYOPT(5) = 2 and magnetic DOF

Table 3, "SOLID98 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID98 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "SOLID98 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,L
sequence number for data at nodes I,J,...,L

FC*n* -
sequence number for solution items for element Face *n*

Table 3 SOLID98 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P1	SMISC	-	2	1	3	-
P2	SMISC	-	4	5	-	6
P3	SMISC	-	-	7	8	9
P4	SMISC	-	11	-	10	12
MUX	NMISC	1	-	-	-	-
MUY	NMISC	2	-	-	-	-
MUZ	NMISC	3	-	-	-	-
FVWX	NMISC	4	-	-	-	-
FVWY	NMISC	5	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
FVWZ	NMISC	6	-	-	-	-
FVWSUM	NMISC	7	-	-	-	-
UE	NMISC	16	-	-	-	-
UD	NMISC	17	-	-	-	-
UM	NMISC	18	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	19	25	31	37
HFAVG	NMISC	20	26	32	38
TAVG	NMISC	21	27	33	39
TBAVG	NMISC	22	28	34	40
HEAT RATE	NMISC	23	29	35	41
HFLXAVG	NMISC	24	30	36	42

SOLID98 Assumptions and Restrictions

- When using SOLID98 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfils boundary conditions for the total field.
- The element must not have a zero volume. Elements may be numbered either as shown in *Figure 1, "SOLID98 Geometry"* or may have node L below the IJK plane. in the *Modeling and Meshing Guide*
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) for more information about the use of midside nodes.
- The difference scalar magnetic potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar and general scalar potential options do not have this restriction.
- Temperatures and heat generation rates, if internally calculated, include any user defined heat generation rates.
- Large deflection capabilities available for KEYOPT(1) = 2 and 3 are not available for KEYOPT(1) = 0. Stress stiffening is available for KEYOPT(1) = 0, 2, and 3.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.

- If you used the MAG degree of freedom, you cannot restart a job in ANSYS Mechanical using Jobname.DB and Jobname.ESAV files that were created by ANSYS Multiphysics.

SOLID98 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic capability.
- The MAG degree of freedom is not active.
- KEYOPT(1) cannot be set to 10. If KEYOPT(1) = 0 (default) or 1, the MAG degree of freedom is inactive.
- The magnetic material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags and magnetic virtual displacements body loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are MAG and VOLT.
- If KEYOPT(1) = 1, the TEMP degree of freedom is inactive. KEYOPT(1) settings of 0, 2, 3 and 8 are not allowed.
- The only allowable material properties are the magnetic and electric properties (MUZERO through PERZ, plus the BH data table).
- The only applicable surface loads are Maxwell force flags. The only applicable body loads are temperatures (for material property evaluation only) and magnetic virtual displacements.
- The element does not have stress stiffening or birth and death features.
- KEYOPT(3) is not applicable.

SHELL99

Linear Layered Structural Shell

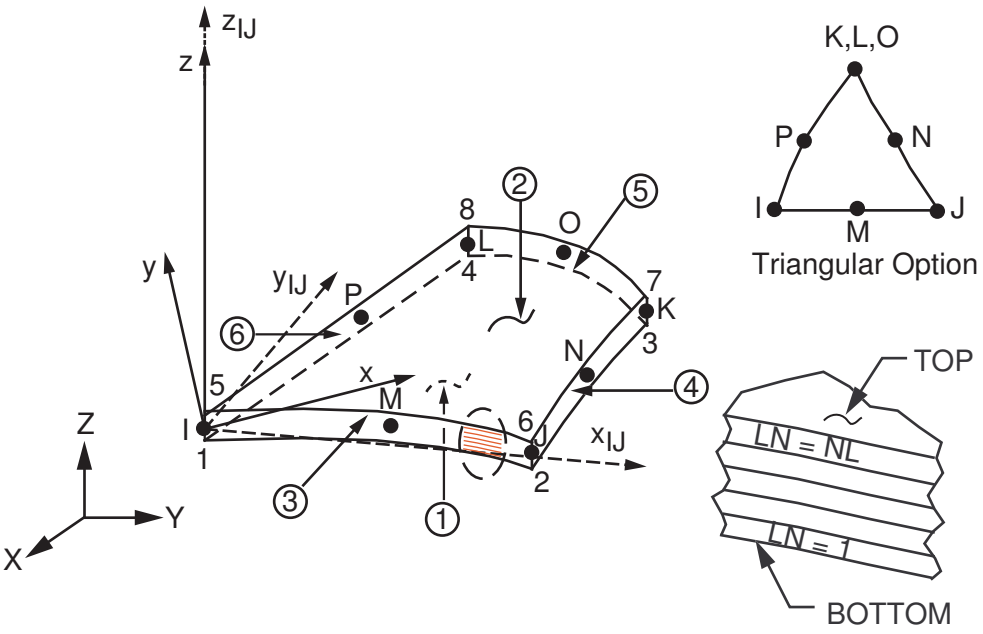
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL99 Element Description

SHELL99 may be used for layered applications of a structural shell model. While SHELL99 does not have some of the nonlinear capabilities of SHELL91, it usually has a smaller element formulation time. SHELL99 allows up to 250 layers. If more than 250 layers are required, a user-input constitutive matrix is available.

The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. See SHELL99 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL99 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

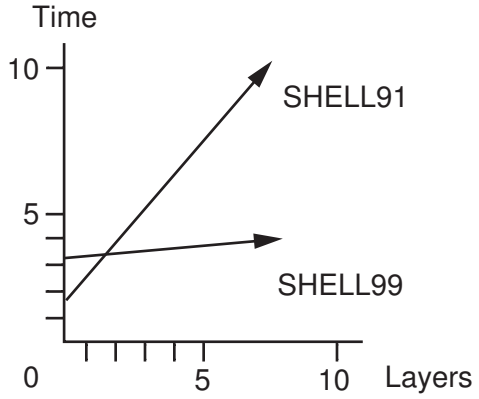
LN = Layer Number

NL = Total Number of Layers

SHELL99 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL99 Geometry"*. The element is defined by eight nodes, average or corner layer thicknesses, layer material direction angles, and orthotropic material properties. Midside nodes may not be removed from this element. See Quadratic Elements (Midside Nodes) of the *Modeling and Meshing Guide* for more information about the use of midside nodes. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

The following graph shows element formation and stress recovery time as a function of the number of layers. While SHELL91 uses less time for elements of under three layers, SHELL99 uses less time for elements with three or more layers.



The elastic foundation stiffness (EFS) is defined as the pressure required to produce a unit normal deflection of the foundation. The elastic foundation capability is bypassed if EFS is less than, or equal to, zero. ADMSUA is the added mass per unit area.

The input may be either in matrix form or layer form, depending upon KEYOPT(2). If matrix form, the matrices must be computed outside of the ANSYS program or from a prior ANSYS run using KEYOPT(10). See the *Theory Reference for ANSYS and ANSYS Workbench*. Briefly, the force-strain and moment-curvature relationships defining the matrices for a linear variation of strain through the thickness (KEYOPT(2) = 2) for a flat shell may be defined as:

$$\begin{Bmatrix} N \\ \overline{M} \end{Bmatrix} = \begin{bmatrix} A & B \\ \overline{B} & \overline{D} \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \overline{\kappa} \end{Bmatrix} - \begin{Bmatrix} MT \\ \overline{BT} \end{Bmatrix}$$

where these terms are defined in the *Theory Reference for ANSYS and ANSYS Workbench*. The submatrix [A] is input by real constants as:

$$[A]_{6x6} = \begin{bmatrix} A_1 & A_2 & A_3 & A_4 & A_5 & A_6 \\ A_2 & A_7 & A_8 & A_9 & A_{10} & A_{11} \\ A_3 & A_8 & A_{12} & A_{13} & A_{14} & A_{15} \\ A_4 & A_9 & A_{13} & A_{16} & A_{17} & A_{18} \\ A_5 & A_{10} & A_{14} & A_{17} & A_{19} & A_{20} \\ A_6 & A_{11} & A_{15} & A_{18} & A_{20} & A_{21} \end{bmatrix} \quad \text{or} \quad [A]_{3x3} = \begin{bmatrix} A_1 & A_2 & A_3 \\ A_2 & A_4 & A_5 \\ A_3 & A_5 & A_6 \end{bmatrix}$$

Submatrices [B] and [D] are input similarly. Note that all submatrices are symmetric. {MT} and {BT} are for thermal effects. Real constants also include the element average density (AVDENS) and the element average thickness (THICK). As flat elements have been seen to give better results than curved elements for KEYOPT(2) = 2 or 4, midside nodes are internally redefined for this case to be on a straight line connecting the corner nodes midway between the nodes for geometric computations. If KEYOPT(2) = 3, quadratic effects are also included with matrices [E], [F], and {QT}, and midside nodes are not redefined for curved elements. For this case, the relationships defining the matrices are expanded to be:

$$\begin{Bmatrix} N \\ M \\ L \end{Bmatrix} = \begin{bmatrix} A & B & \\ B & D & E \\ & E & F \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \\ \omega \end{Bmatrix} - \begin{Bmatrix} MT \\ BT \\ QT \end{Bmatrix} \quad \textbf{(1)}$$

If KEYOPT(2) = 4, the transverse shear terms are, for example, A_6 * TRSHEAR where TRSHEAR is input and defaults to 1000.0. *SHELL99 Assumptions and Restrictions* provides a limitation on the use of matrix input. No stresses, thermal strains, or failure criteria are available with matrix input.

For layer (non-matrix) input, the element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The local coordinate system for each layer is defined as shown in *Figure 2, "SHELL99 Stress Output"*. In this local right-handed system, the x'-axis is rotated an angle THETA(LN) (in degrees) from the element x-axis toward the element y-axis.

The total number of layers must be specified (NL). If KEYOPT(2) = 0, the maximum number of layers is 250; if KEYOPT(2) = 1, the maximum is 125. The properties of all layers should be entered (LSYM = 0). If the properties of the layers are symmetrical about the midthickness of the element (LSYM = 1), only half of properties of the layers, up to and including the middle layer (if any), need to be entered. While all layers may be printed, two layers may be specifically selected to be output (LP1 and LP2, with LP1 usually less than LP2).

The material properties of each layer may be orthotropic in the plane of the element. The real constant MAT defines the layer material number instead of the element material number applied with the **MAT** command. MAT defaults to 1 if no input exists. The material X direction corresponds to the local layer x' direction.

You can input layer information via the real constants as described, or by using a combination of section information and data from a FiberSIM `.xml` file. To learn more about using FiberSIM data in your ANSYS simulation, see *Section 13.2: The FiberSIM-ANSYS Interface* in the *Structural Analysis Guide*.

Use **TREF** and **BETAD** to supply global values for reference temperature and damping, respectively. Alternatively, use the **MAT** command to specify element-dependent values for reference temperature (**MP**,REFT) or damping (**MP**,DAMP); layer material numbers are ignored for this purpose.

Each layer of the laminated shell element may have a variable thickness (TK) by selecting KEYOPT(2) = 1. The thickness is assumed to vary bilinearly over the area of the layer, with the thickness input at the corner node locations. If the layer has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four corner thicknesses must be input using positive values. The total thickness of each shell element must be less than twice the radius of curvature, and should be less than one-fifth the radius of curvature.

You can specify the nodes to be at the top, middle or bottom surface of the element. The choice is made through the node offset option (KEYOPT(11)). This option is very convenient, for example, when modeling laminated structures with ply drop-off, where the location of the top or bottom surface may be better defined than the location of the midplane as shown in *Figure 4, "SHELL91 Bottom Surface Nodes"*.

You can also define two elements that share the same nodes, but with each element having a different setting of KEYOPT(11), as shown in *Figure 5, "SHELL91 Common Node Elements"*.

The failure criteria selection is input in the data table [**TB**], as described in *Table 2.2, "Orthotropic Material Failure Criteria Data"*. Three predefined criteria are available and up to six user-defined criteria may be entered with user subroutines. See Failure Criteria in the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the three predefined failure criteria. See *Guide to ANSYS User Programmable Features* for an explanation of user subroutines. Failure criteria may also be computed in POST1 (using the **FC** commands). All references to failure criteria as part of element output data are based only on the **TB** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces. The edge pressures act at the nodal plane as shown by circled numbers 3 through 6 on *Figure 1, "SHELL99 Geometry"*. The mass matrix is also assumed to act at the nodal plane. Depending on KEYOPT(11), the nodal plane may be at the midsurface, or at the top or bottom surface. Positive pressures act into the element. Edge pressures are input as force per unit length. Temperatures may be input as element body loads at the "corner" locations (1-8) shown in *Figure 1, "SHELL99 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T1 is used for T1, T2, T3, and T4, while T2 (as input) is used for T5, T6, T7, and T8. For any other input pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *SHELL99 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL99 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

See *Table 1, "SHELL99 Real Constants"* for a description of the real constants.

Material Properties

If KEYOPT(2) = 0 or 1, supply the following 13*NM properties where NM is the number of materials (maximum is NL):

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, for each of the NM materials.

If KEYOPT(2) = 2, 3 or 4, supply none of the above.

Supply DAMP and REFT only once for the element (use **MAT** command to assign material property set). See the discussion in *SHELL99 Input Data* for more details.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

T1, T2, T3, T4, T5, T6, T7, T8 if KEYOPT(2) = 0 or 1
None if KEYOPT(2) = 2, 3 or 4

Special Features

Stress stiffening
Large deflection

KEYOPT(2)

Form of input:

0 --
Constant thickness layer input (250 layers maximum)

1 --
Tapered layer input (125 layers maximum)

2 --
6 x 6 matrix input using linear logic

3 --
6 x 6 matrix input using quadratic logic

4 --
3 x 3 matrix input using linear logic

KEYOPT(3)
Extra element output:

0 --
Basic element printout

1 --
Integration point strain printout

2 --
Nodal force and moment printout in element coordinates

3 --
Force and moment per unit length printout (available only if KEYOPT(2) = 0 or 1)

4 --
Combination of all three options

KEYOPT(4)
Element coordinate system defined by:

0 --
No user subroutines used to define element coordinate system

4 --
Element x-axis located by user subroutine USERAN

5 --
Element x-axis located by user subroutine USERAN and layer x-axes located by user subroutine USANLY

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(5)
Strains or stresses output (will be used with KEYOPT(6)):

0 --
Strain results will be used

1 --
Stress results will be used

2 --
Both strain and stress results will be used

KEYOPT(6)
Extra element output (for layer input only) (used for printout control):

0 --
Basic element printout, as well as the summary of the maximum of all the failure criteria

1 --
Same as 0 but also print the summary of all the failure criteria and the summary of the maximum of the interlaminar shear stress

2 --
Same as 1 but also print the layer solution at the integration points in the bottom layer (or LP1) and the top layer (or LP2)

3 --
Same as 1 but also print the layer solution at the element centroid for all layers, as well as the interlaminar shear stress solution between layers

4 --
Same as 1 but also print the layer solution at the corner nodes for all layers, as well as the interlaminar shear stress solution between layers

5 --
Same as 1 but also print the layer solution with the failure criterion values at the integration points for all layers, as well as the interlaminar shear stress solution between layers

Note

No stresses, thermal strains, or failure criteria are available with matrix input.

KEYOPT(8)
Storage of layer data:

0 --
Store data for bottom of bottom layer (or LP1) and top of top layer (or LP2). Also store data for maximum failure criteria layer.

1 --
Store data for all layers.

Caution

Volume of data stored may be excessive.

KEYOPT(9)
Determines where strains, stresses, and failure criteria are evaluated (available only if KEYOPT(2) = 0 or 1 with NL > 1):

0 --
Evaluate strains and stresses at top and bottom of each layer

1 --
Evaluate at midthickness of each layer

KEYOPT(10)
Material property matrix output:

0 --
No material property matrices printed

1 --
Print material property matrices integrated through thickness for element number 1, if element number 1 is a SHELL99 element

2 --
Same as 1 but if KEYOPT(2) = 0 or 1, also write matrices as **RMODIF** commands for use with KEYOPT(2) = 2

3 --
Same as 1 but if KEYOPT(2) = 0 or 1, also write matrices as **RMODIF** commands for use with KEYOPT(2) = 3

KEYOPT(11)
Node offset option:

0 --
Nodes located at midsurface

1 --
Nodes located at bottom surface

2 --
Nodes located at top surface

Table 1 SHELL99 Real Constants

No.	Name	Description
If KEYOPT(2) = 0, supply the following 12+(3*NL) constants:		
1	NL	Number of layers (250 maximum)
2	LSYM	Layer symmetry key
3	LP1	First layer for output
4	LP2	Second layer for output
5	EFS	Elastic foundation stiffness
6	ADMSUA	Added mass/unit area
7 ... 12	(Blank)	- -
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK	Layer thickness for layer 1
16	MAT	Material number for layer 2
17	THETA	x-axis rotation for layer 2
18	TK	Layer thickness for layer 2
19 ... 12+(3*NL)	MAT, THETA, etc.	Repeat MAT, THETA, and TK for each layer (up to NL layers)
If KEYOPT(2) = 1, supply the following 12+(6*NL) constants:		
1	NL	Number of layers (250 maximum)
2	LSYM	Layer symmetry key
3	LP1	First layer for output
4	LP2	Second layer for output
5	EFS	Elastic foundation stiffness
6	ADMSUA	Added mass/unit area
7 ... 12	(Blank)	- -
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK(I)	Layer thickness at node I for layer 1
16	TK(J)	Layer thickness at node J for layer 1

No.	Name	Description
17	TK(K)	Layer thickness at node K for layer 1
18	TK(L)	Layer thickness at node L for layer 1
19 ... 12+(6*NL)	MAT, THETA, etc.	Repeat MAT, THETA, TK(I), TK(J) , TK(K), and TK(L) for each layer (up to NL layers)
If KEYOPT(2) = 2, supply the following 79 constants:		
1 ... 21	A(1) ... A(21)	Submatrix A
22 ... 42	B(1) ... B(21)	Submatrix B
43 ... 63	D(1) ... D(21)	Submatrix D
64 ... 69	MT(1) ... MT(6)	MT array
70 ... 75	BT(1) ... BT(6)	BT array
76	AVDENS	Element average density
77	THICK	Element average thickness
78	EFS	Elastic foundation stiffness
79	ADMSUA	Added mass/unit area
If KEYOPT(2) = 3, supply the following 127 constants:		
1 ... 21	A(1) ... A(21)	Submatrix A
22 ... 42	B(1) ... B(21)	Submatrix B
43 ... 63	D(1) ... D(21)	Submatrix D
64 ... 84	E(1) ... E(21)	Submatrix E
85 ... 105	F(1) ... F(21)	Submatrix F
106 ... 111	MT(1) ... MT(6)	MT array
112 ... 117	BT(1) ... BT(6)	BT array
118 ... 123	QT(1) ... QT(6)	QT array
124	AVDENS	Element average density
125	THICK	Element average thickness
126	EFS	Elastic foundation stiffness
127	ADMSUA	Added mass/unit area
If KEYOPT(2) = 4, supply the following 30 constants:		
1 ... 6	A(1) ... A(6)	Submatrix A
7 ... 12	B(1) ... B(6)	Submatrix B
13 ... 18	D(1) ... D(6)	Submatrix D
19 ... 21	MT(1) ... MT(3)	MT array
22 ... 24	BT(1) ... BT(3)	BT array
25	AVDENS	Element average density
26	THICK	Element average thickness
27	EFS	Elastic foundation stiffness
28	ADMSUA	Added mass/unit area
29	(Blank)	- -
30	TRSHEAR	Transverse shear

For more information on real constants and other input data, see *SHELL91* . A discussion on failure criteria is found in *Section 2.2.2.12: Failure Criteria*.

SHELL99 Output Data

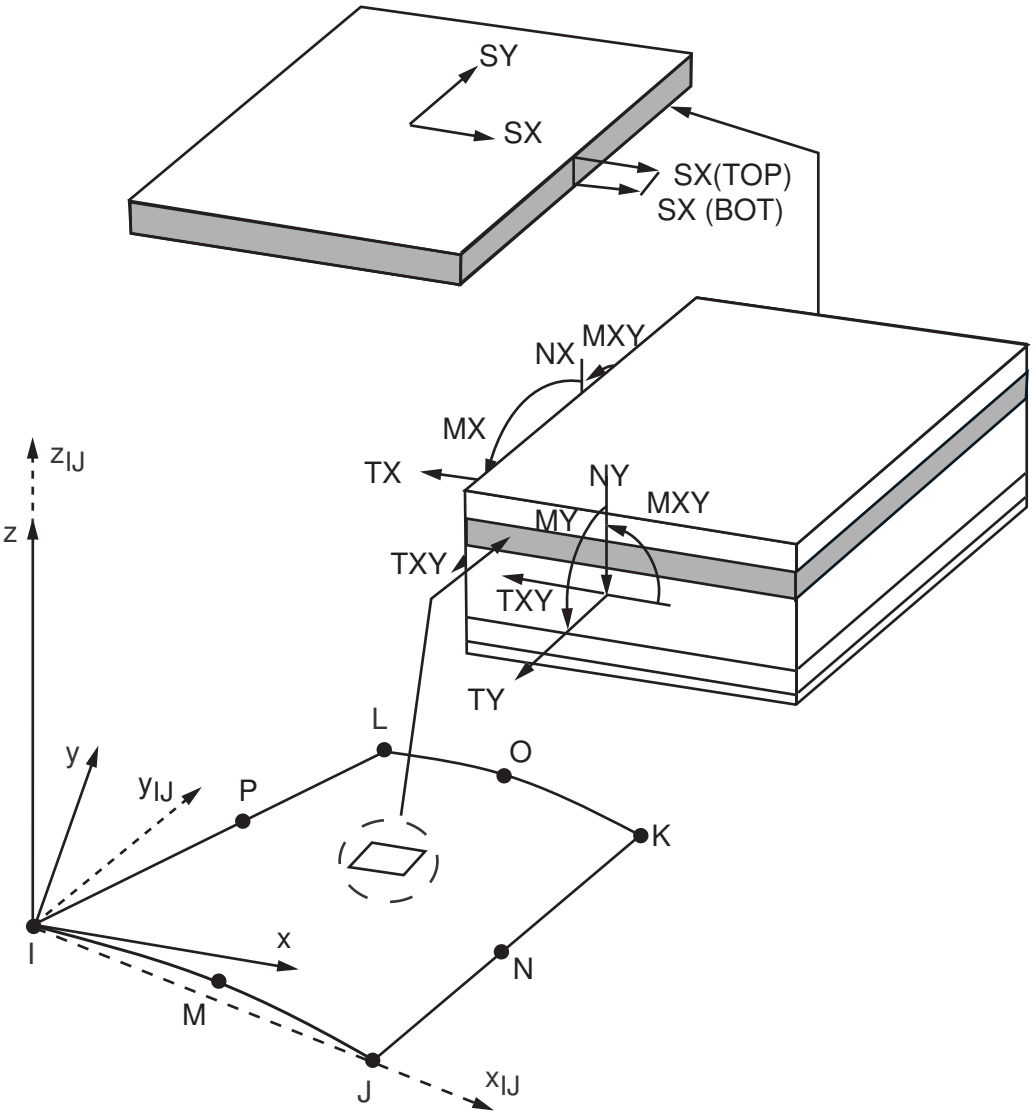
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL99 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "SHELL99 Stress Output"*. The element stress directions correspond to the layer local coordinate directions.

Various layer printout options are available. For integration point output, integration point 1 is nearest node I, 2 nearest J, 3 nearest K, and 4 nearest L. Failure criterion output is evaluated only at the in-plane integration points. (See the *Theory Reference for ANSYS and ANSYS Workbench*). After the layer printout, the in-plane forces and moments are listed for the entire element if KEYOPT(3) = 3 or 4. These are shown in *Figure 2, "SHELL99 Stress Output"*. The moments include the moment about the x-face (MX), the moment about the y-face (MY), and the twisting moment (MXY). The forces and moments are calculated per unit length in the element coordinate system and are the combined sum for all layers. If KEYOPT(3) = 2 or 4 for this element, the 6 member forces and moments are also printed for each node (in the element coordinate system). KEYOPT(8) controls the amount of data output on the postdata file for processing with the **LAYER** or **LAYERP26** command. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SHELL99 Stress Output



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL99 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y

Name	Definition	O	R
VOLU:	Volume	Y	Y
TTOP, TBOT	Average temperatures at top and bottom faces	*1*	-
XC, YC, ZC	Element centroid	Y	*11*
PRES	Pressures: P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T1, T2, T3, T4, T5, T6, T7, T8	Y	Y
INT	Integration point number	*2*	-
POS	Top (TOP), Bottom (BOT), Midthickness (MID) of element	*2*	-
XI, YI, ZI	Global X, Y, Z location of integration point	*2*	-
NUMBER	Layer number	*1*, *3*	-
MAT	Material number of this layer	*1*, *3*	-
THETA	Material direction angle for layer (THETA)	*1*, *3*	-
AVE THICK	Average thickness of layer	*3*	-
ACC AVE THICK	Accumulative average thickness (thickness of element from layer 1 to this layer)	*1*, *3*	-
AVE TEMP	Average temperature of layer	*3*	-
POS	Top (TOP), Bottom (BOT), Midthickness (MID) of layer (see KEYOPT(9) for control options)	*3*	-
LOC	Center location (average) (if KEYOPT(6) = 3)	*1*, *3*	-
NODE	Corner node number (if KEYOPT(6) = 4)	*1*, *3*	-
INT	Integration point number (if KEYOPT(6) = 2 or 5)	*1*, *3*	-
S:X, Y, Z, XY, YZ, XZ	Stresses (in layer local coordinates)	*1*, *4*	Y
S:1, 2, 3	Principal stress	*1*, *4*	-
S:INT	Stress intensity	*1*, *4*	-
S:EQV	Equivalent stress	*1*, *4*	-
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains (in layer local coordinates), Total strain if KEYOPT(2) = 2 or 3	*4*	Y
EPEL:EQV	Equivalent elastic strains (in layer local coordinates) [*12*]	*4*	-
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains (in layer local coordinates), Total strain if KEYOPT(2) = 2 or 3	*4*	Y
EPTH:EQV	Equivalent thermal strains (in layer local coordinates) [*12*]	*4*	-
EPTO:X, Y, Z, XY, YZ, XZ	Total strains (no thermal strain adjustment if KEYOPT(2) = 0 or 1) in element coordinates	*2*	-
EPTO:EQV	Total equivalent strains (no thermal strain adjustment if KEYOPT(2) = 0 or 1) in layer local coordinates	*2*	-
FC1 ... FC6, FCMAX	Failure criterion values and maximum at each integration point, output only if KEYOPT(6) = 5	*1*, *4*	-
FC	Failure criterion number (FC1 to FC6, FCMAX)	*1*, *5*	*1*
VALUE	Maximum value for this criterion (if value exceeds 9999.999, 9999.999 will be output)	*1*, *5*	*1*
LN	Layer number where maximum occurs	*1*, *5*	*1*
EPELF(X, Y, Z, XY, YZ, XZ)	Elastic strains (in layer local coordinates) causing the maximum value for this criterion in the element	*1*, *5*	*1*

Name	Definition	O	R
SF(X, Y, Z, XY, YZ, XZ)	Stresses (in layer local coordinates) causing the maximum value for this criterion in the element	*1, 5*	*1*
LAYERS	Interface location	*1, 6*	-
ILSXZ	SXZ interlaminar shear stress	*1, 6*	*1*
ILSYZ	SXZ interlaminar shear stress	*1, 6*	*1*
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	*1, 6*	*1*
ILSUM	Interlaminar shear stress vector sum	*1, 6*	*1*
LN1, LN2	Layer numbers which define location of maximum interlaminar shear stress (ILMAX)	*1, 7*	*1*
ILMAX	Maximum interlaminar shear stress (occurs between LN1 and LN2)	*1, 7*	*1*
T(X, Y, XY)	Element total in-plane forces per unit length (in element coordinates)	*8*	-
N(X, Y)	Out-of-plane element X and Y shear forces	*8*	-
M(X, Y, XY)	Element total moments per unit length (in element coordinates)	*9*	-
MFOR(X, Y, Z)	Member forces for each node in the element coordinate system	*10*	-
MMOM(X, Y, Z)	Member moments for each node in the element coordinate system	*10*	-

1. If KEYOPT(2) = 0 or 1
2. Integration point strain solution (if KEYOPT(3) = 1 or 4)
3. Layer solution (if KEYOPT(6) > 1)
4. The item output is controlled with KEYOPT(5)
5. Summary of failure criteria calculation: if KEYOPT(6) = 0, only maximum of all failure criteria (FCMAX) in element is output;

   Output of the elastic strains and/or stresses (depending on KEYOPT(5)) for each failure criterion and the maximum of all criteria (FCMAX).
6. Interlaminar stress solution (if KEYOPT(6) > 2)
7. Printed only if KEYOPT(6) $\neq$ 0
8. Output at the corner nodes only if KEYOPT(3) = 3 or 4
9. Output at the corner nodes only if KEYOPT(3) = 3 or 4, and KEYOPT(9) $\neq$ 1
10. Output only if KEYOPT(3) = 2 or 4
11. Available only at centroid as a ***GET** item.
12. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 3, "SHELL99 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SHELL99 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SHELL99 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,L
sequence number for data at nodes I,J,...,L

Table 3 SHELL99 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	(2*i)+7	(2*NL)+9
ILSYZ	SMISC	(2*i)+8	(2*NL)+10
ILSUM	NMISC	(2*i)+5	(2*NL)+7
ILANG	NMISC	(2*i)+6	(2*NL)+8

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	(2*NL)+11	(2*NL)+12	(2*NL)+13	(2*NL)+14
P2	SMISC	(2*NL)+15	(2*NL)+16	(2*NL)+17	(2*NL)+18
P3	SMISC	(2*NL)+20	(2*NL)+19		
P4	SMISC		(2*NL)+22	(2*NL)+21	
P5	SMISC			(2*NL)+24	(2*NL)+23
P6	SMISC	(2*NL)+25			(2*NL)+26

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TX	SMISC	1
TY	SMISC	2
TXY	SMISC	3
MX	SMISC	4
MY	SMISC	5
MXY	SMISC	6
NX	SMISC	7
NY	SMISC	8
FCMAX (over all layers)	NMISC	1
VALUE	NMISC	2
LN	NMISC	3
ILMAX	NMISC	4
LN1	NMISC	5

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
LN2	NMISC	6
ILSUM (between layers i and i+1)	NMISC	2i+7
ILANG	NMISC	2i+8
FCMAX (at layer i)	NMISC	2*(NL+i)+7
VALUE (at layer i)	NMISC	2*(NL+i)+8
FC	NMISC	(4*NL)+8+15(N-1)+1
VALUE	NMISC	(4*NL)+8+15(N-1)+2
LN	NMISC	(4*NL)+8+15(N-1)+3
EPELFX	NMISC	(4*NL)+8+15(N-1)+4
EPELFY	NMISC	(4*NL)+8+15(N-1)+5
EPELFZ	NMISC	(4*NL)+8+15(N-1)+6
EPELFXY	NMISC	(4*NL)+8+15(N-1)+7
EPELFYZ	NMISC	(4*NL)+8+15(N-1)+8
EPELFXZ	NMISC	(4*NL)+8+15(N-1)+9
SFX	NMISC	(4*NL)+8+15(N-1)+10
SFY	NMISC	(4*NL)+8+15(N-1)+11
SFZ	NMISC	(4*NL)+8+15(N-1)+12
SFXY	NMISC	(4*NL)+8+15(N-1)+13
SFYZ	NMISC	(4*NL)+8+15(N-1)+14
SFXZ	NMISC	(4*NL)+8+15(N-1)+15

Note

The i in *Table 3, "SHELL99 Item and Sequence Numbers"* (where i = 1, 2, 3 ..., NL) refers to the layer number of the shell. NL is the maximum layer number as input for real constant NL (1 3 NL 3 250). N is the failure number as stored on the results file in compressed form, e.g., only those failure criteria requested will be written to the results file. For example, if only the maximum strain and the Tsai-Wu failure criteria are requested, the maximum strain criteria will be stored first (N = 1) and the Tsai-Wu failure criteria will be stored second (N = 2). In addition, if more than one criteria is requested, the maximum value over all criteria is stored last (N = 3 for this example).

SHELL99 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness layers are allowed only if a zero thickness is defined at all corners. Tapering down to zero is not allowed.
- If KEYOPT(11) = 0, all nodes are assumed to be at the midthickness of the element.
- All inertial effects are assumed to be in the nodal plane, i.e., unbalanced laminate construction and offsets have no effect on the mass properties of the element.

- No slippage is assumed between the element layers. Shear deflections are included in the element, however, normals to the center plane before deformation are assumed to remain straight after deformation.
- This element may produce inaccurate results under thermal loads for non-flat domains.
- The applied transverse thermal gradient is assumed to be linear through the element and over the element surface.
- The stress varies linearly through the thickness of each layer.
- Interlaminar transverse shear stresses are based on the assumption that no shear is carried at the top and bottom surfaces of an element. Further, these interlaminar shear stresses are only computed at the centroid and are not valid along the element boundaries. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- The element matrices are reformed every iteration unless option 1 of the **KUSE** command is active. Only the lumped mass matrix is available. The mass matrix is assumed to act at the nodal plane.
- The large deflection option for SHELL99 is not as convergent as it is for SHELL91 (the nonlinear layered shell element). SHELL91 may be the preferred element type when constructing models that include large deflection

If you have defined the element using the node offset option (KEYOPT(11) $\neq$ 0), be aware of the following:

- You should not use shell-to-solid submodeling [**CBDOF**] or temperature interpolation [**BFINT**].
- You should not use the matrix input option (KEYOPT(2) = 2 or 3).
- The transverse shear stresses will not be valid if two elements share the same nodes but have different settings of KEYOPT(11) (for example, as shown in *Figure 5, "SHELL91 Common Node Elements"*). Also, POST1 nodal results in this case should be obtained from either the top or the bottom element, since nodal data averaging will not be valid if elements from both sides of the nodal plane are used.

SHELL99 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element is limited to 20 constant thickness layers, or 10 tapered layers, and does not allow the user-input constitutive matrix option (that is, KEYOPT(2) = 2 or 3 is not valid).
- The DAMP material property is not allowed.
- KEYOPT(4) can only be set to 0 (default).
- The six user-defined failure criteria (subroutines USRFC1 through USRFC6) are not allowed.

VISCO106

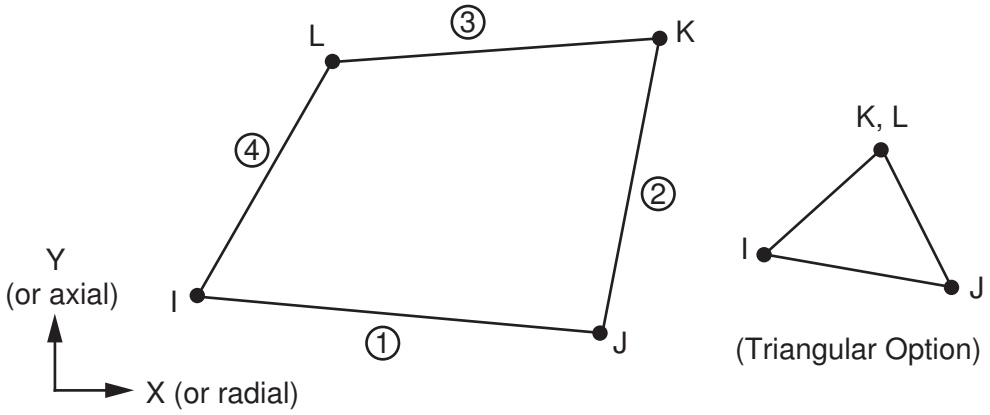
2-D 4-Node Viscoplastic Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

VISCO106 Element Description

VISCO106 is used for 2-D modeling of solid structures. It can be used either as a plane strain or as an axisymmetric element, and is defined by four nodes having up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element is designed to solve both isochoric (volume preserving) rate-independent and rate-dependent large strain plasticity problems. Iterative solution procedures must be used with VISCO106 because it is used to represent highly nonlinear behavior. Large deflections [**NLGEOM**] must be active in order to update the geometry in each substep. See VISCO106 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A midside node version of this element is VISCO108.

Figure 1 VISCO106 Geometry



VISCO106 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "VISCO106 Geometry"*. The element input data includes four nodes and the linear and nonlinear material properties.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown in *Figure 1, "VISCO106 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a planar analysis and on a full 360° basis for an axisymmetric analysis.

KEYOPT(5) and KEYOPT(6) provide various element printout options (see *Section 2.2.2: Element Solution*).

A summary of the element input is given in *Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

Input Summary

Nodes
I, J, K, L

Degrees of Freedom

UX, UY if KEYOPT(3) = 0
UX, UY, UZ if KEYOPT(3) = 1

Real Constants
None

Material Properties
EX, PRXY (or NUXY), ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features

Rate-dependent plasticity (ANAND)
Stress stiffening
Large deflection
Large strain
Adaptive descent

KEYOPT(3)
Element behavior:

0, 2 --
Plane strain (Z strain = 0.0)

1 --
Axisymmetric

KEYOPT(5)
Extra element output:

0 --
Centroidal solution

1 --
Centroidal and integration point solution

2 --
Centroidal and integration point solution plus state variable and total plastic work

KEYOPT(6)
Strain output at integration points:

0 --
No strain output

1 --
Total strain at integration points

KEYOPT(7)
Type of stress update:

0 --
Scalar consistent stress update

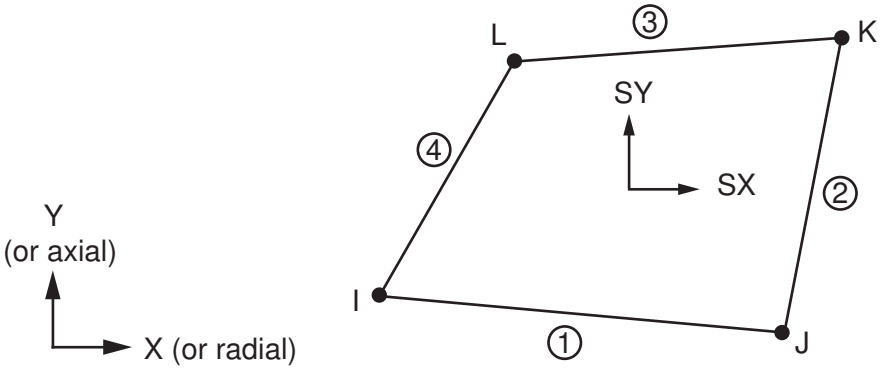
1 --
Euler backward stress update

VISCO106 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "VISCO106 Element Output Definitions"*.

Figure 2 VISCO106 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 VISCO106 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
CENT:X, Y, PSV, NL, EPTO	Global X, Y location	Y	Y
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses (SYZ = SXZ = 0.0 for plane strain)	Y	Y
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [*1*]	-	Y

Name	Definition	O	R
EPTH:X, Y, Z, XY	Average thermal strain	-	Y
EPTH:EQV	Equivalent thermal strain [*1*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	-	Y
EPPL:EQV	Equivalent plastic strain [*1*]	-	Y
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL)	Y	Y
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL)	-	Y
NL:PSV	Plastic state variable	Y	Y
NL:PLWK	Plastic work-per-volume	Y	Y
URS	Right stretch tensor (X, Y, Z, XY, YZ, XZ)	-	Y

1. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic this value is set at 0.5.

Table 2 VISCO106 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	TEMP, S, SINT, SEQV	*1*	-
Integration Point Solution	PSV, PLWK	*2*	-
Integration Point Strain Solution	EPTO	*3*	-

1. Output at each integration point, if KEYOPT(5) = 1 or 2
2. Output at each integration point if KEYOPT(5) = 2
3. Output at each integration point if KEYOPT(6) = 1

Note

For axisymmetric solutions with KEYOPT(3) = 1, the X, Y, Z, XY, YZ and XZ stress and strain outputs correspond to the radial, axial hoop, in-plane, and out-of-plane torsional shear stresses, respectively.

Table 3, "VISCO106 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "VISCO106 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "VISCO106 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,K,L
: sequence number for data at nodes I,J,K,L

Table 3 VISCO106 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
S:1	NMISC	1	6	11	16
S:2	NMISC	2	7	12	17
S:3	NMISC	3	8	13	18
S:INT	NMISC	4	9	14	19
S:EQV	NMISC	5	10	15	20
URSX	NMISC	21	27	33	39
URSY	NMISC	22	28	34	40
URSZ	NMISC	23	29	35	41
URSXY	NMISC	24	30	36	42
URSYZ	NMISC	25	31	37	43
URSXZ	NMISC	26	32	38	44

VISCO106 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "VISCO106 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- Only the isotropic and Anand material laws (BISO, MISO, ANAND on the **TB** command) are valid for this element.

VISCO106 Product Restrictions

There are no product-specific restrictions for this element.

VISCO107

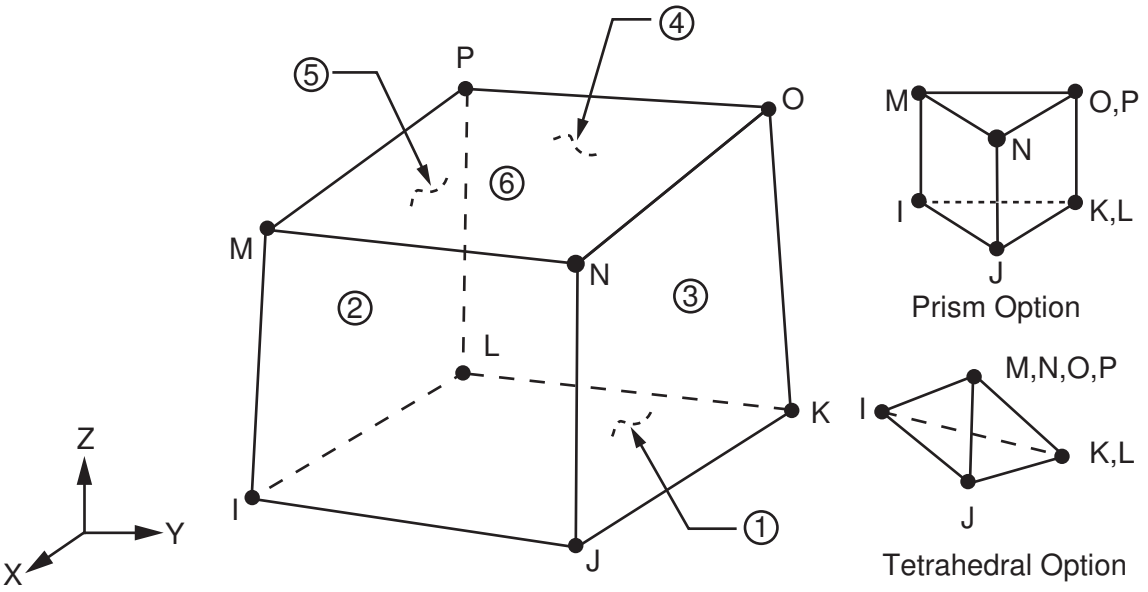
3-D 8-Node Viscoplastic Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

VISCO107 Element Description

VISCO107 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element is designed to solve both isochoric (volume preserving) rate-independent and rate-dependent large strain plasticity problems. Iterative solution procedures must be used with VISCO107 since it is used to represent highly nonlinear behavior. Large deflections [**NLGEOM**] must be active in order to update the geometry each substep. See VISCO107 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 VISCO107 Geometry



VISCO107 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "VISCO107 Geometry"*. The element input data includes eight nodes and linear and nonlinear material properties.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown in *Figure 1, "VISCO107 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(5) and KEYOPT(6) parameters provide various element printout options (see *Section 2.2.2: Element Solution*).

A summary of the element input is given in *VISCO107 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

VISCO107 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
EX, PRXY (or NUXY), ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Rate-dependent plasticity (ANAND)
Stress stiffening
Large deflection
Large strain
Adaptive descent

KEYOPT(5)
Extra element output:

0 --
Centroidal solution

1 --
Centroidal and integration point solution

2 --
Centroidal and integration point solution plus state variable and total plastic work

KEYOPT(6)
Strain output at integration points:

0 --
No strain output

1 --
Total strain at integration points

KEYOPT(7)
Type of stress update:

0 --
Scalar consistent stress update

1 --
Euler backward stress update

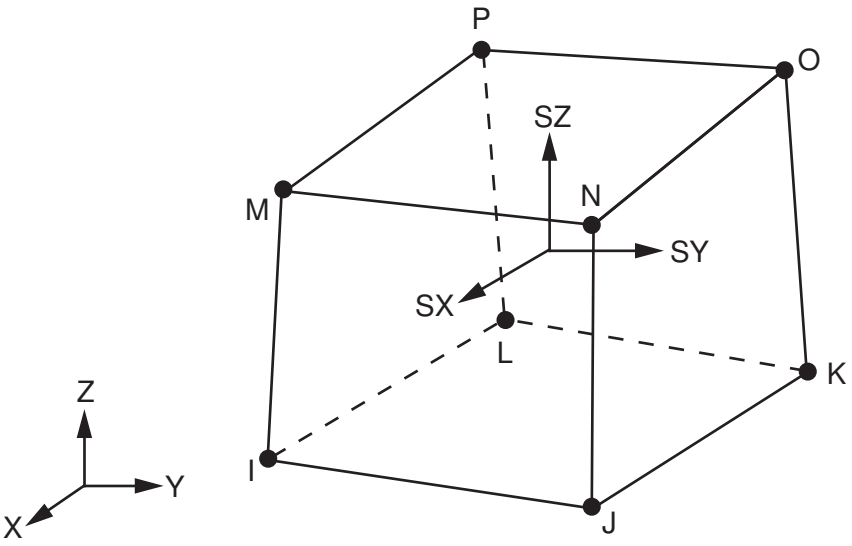
VISCO107 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "VISCO107 Element Output Definitions"*.

The element output directions are parallel to the rotated (see *Section 2.3: Coordinate Systems*) element coordinate system (see *Figure 2, "VISCO107 Stress Output"*). General solution output is described in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 VISCO107 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 VISCO107 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*1*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y

Name	Definition	O	R
S:1, 2, 3	Principal stresses	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, XY, YZ, XZ	Elastic strains	-	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [*2*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	-	Y
EPTH:EQV	Equivalent thermal strain [*2*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	-	Y
EPPL:EQV	Equivalent plastic strain [*2*]	-	Y
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL)	Y	Y
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL)	-	Y
NL:PSV	Plastic state variable	Y	Y
NL:PLWK	Plastic work/volume	Y	Y
URS	Right stretch tensor (X, Y, Z, XY, YZ, XZ)	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic this value is set at 0.5.

Table 2 VISCO107 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	TEMP, S, SINT, SEQV	*1*	-
Integration Point Solution	PSV, PLWK	*2*	-
Integration Point Strain Solution	EPTO	*3*	-

1. Output at each integration point, if KEYOPT(5) = 1 or 2
2. Output at each integration point if KEYOPT(5) = 2
3. Output at each integration point if KEYOPT(6) = 1

Table 3, "VISCO107 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "VISCO107 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "VISCO107 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,...,P
: sequence number for data at nodes I,J,...,P

Table 3 VISCO107 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40
URSX	NMISC	41	47	53	59	65	71	77	83
URSY	NMISC	42	48	54	60	66	72	78	84
URSZ	NMISC	43	49	55	61	67	73	79	85
URSXY	NMISC	44	50	56	62	68	74	80	86
URSYZ	NMISC	45	51	57	63	69	75	81	87
URSXZ	NMISC	46	52	58	64	70	76	82	88

VISCO107 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1,"VISCO107 Geometry"* or may have the planes IJKL and MNOP interchanged
- The element should not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- A tetrahedron shape is also available.
- Only the isotropic and Anand material laws (BISO, MISO, ANAND on the **TB** command) are valid for this element.

VISCO107 Product Restrictions

There are no product-specific restrictions for this element.

VISCO108

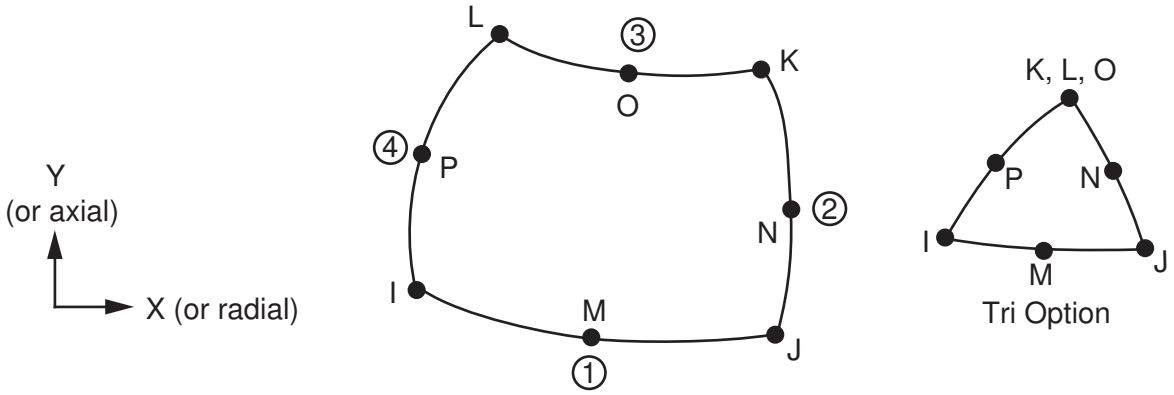
2-D 8-Node Viscoplastic Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

VISCO108 Element Description

VISCO108 is used for 2-D modeling of solid structures. It can be used either as a plane strain or as an axisymmetric element, and is defined by eight nodes having up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element is designed to solve both isochoric (volume preserving) rate-independent and rate-dependent large strain plasticity problems. Iterative solution procedures must be used with VISCO108 since it is used to represent highly nonlinear behavior. Large deflections [**NLGEOM**] must be active in order to update the geometry each substep. See VISCO108 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 VISCO108 Geometry



VISCO108 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "VISCO108 Geometry"*. The element input data includes eight nodes and linear and nonlinear material properties.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown in *Figure 1, "VISCO108 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a planar analysis and on a full 360° basis for an axisymmetric analysis.

KEYOPT(5) and KEYOPT(6) provide various element printout options (see *Section 2.2.2: Element Solution*).

An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A summary of the element input is given in *VISCO108 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

VISCO108 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY if KEYOPT(3) = 0
UX, UY, UZ if KEYOPT(3) = 1

Real Constants
None

Material Properties
EX, PRXY (or NUXY), ALPX (or CTEX *or* THSX), DENS, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Rate-dependent plasticity (ANAND)
Stress stiffening
Large deflection
Large strain
Adaptive descent

KEYOPT(3)
Element behavior:

0, 2 --
Plane strain (Z strain = 0.0)

1 --
Axisymmetric

KEYOPT(5)
Extra element output:

0 --
Centroidal solution

1 --
Centroidal and integration point solution

2 --
Centroidal and integration point solution plus state variable and total plastic work

KEYOPT(6)
Strain output at integration points:

0 --
No strain output

1 --
Total strain at integration points

KEYOPT(7)
Type of stress update:

0 --
Scalar consistent stress update

1 --
Euler backward stress update

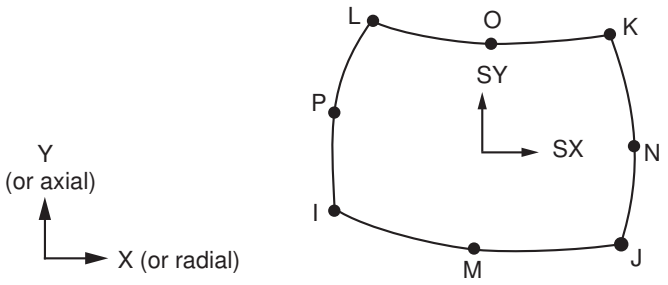
VISCO108 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "VISCO108 Element Output Definitions"*.

The element output directions are parallel to the rotated (see *Section 2.3: Coordinate Systems*) element coordinate system (see *Figure 2, "VISCO108 Stress Output"*). General solution output is described in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 VISCO108 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 VISCO108 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*1*
PRES	Pressures P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Stresses (SYZ = SXZ = 0.0 for plane strain)	Y	Y
S:1, 2, 3	Principal stresses	Y	Y

Name	Definition	O	R
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain [*2*]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	-	Y
EPTH:EQV	Equivalent thermal strain [*2*]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	-	Y
EPPL: EQV	Equivalent plastic strain [*2*]	-	Y
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL)	Y	Y
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL)	-	Y
NL:PSV	Plastic state variable	Y	Y
NL:PLWK	Plastic work-per-volume	Y	Y
URS	Right stretch tensor (X, Y, Z, XY, YZ, XZ)	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic this value is set at 0.5.

Table 2 VISCO108 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	TEMP, S, SINT, SEQV	*1*	-
Integration Point Solution	PSV, PLWK	*2*	-
Integration Point Strain Solution	EPTO	*3*	-

1. Output at each integration point, if KEYOPT(5) = 1 or 2
2. Output at each integration point if KEYOPT(5) = 2
3. Output at each integration point if KEYOPT(6) = 1

Note

For axisymmetric solutions with KEYOPT(3) = 1, the X, Y, Z, XY, YZ and XZ stress and strain outputs correspond to the radial, axial, hoop, in-plane, and out-of-plane torsional shear stresses, respectively.

Table 3, "VISCO108 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "VISCO108 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "VISCO108 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,...,L
: sequence number for data at nodes I,J,...,L

Table 3 VISCO108 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
S:1	NMISC	1	6	11	16
S:2	NMISC	2	7	12	17
S:3	NMISC	3	8	13	18
S:INT	NMISC	4	9	14	19
S:EQV	NMISC	5	10	15	20
URSX	NMISC	21	27	33	39
URSY	NMISC	22	28	34	40
URSZ	NMISC	23	29	35	41
URSXY	NMISC	24	30	36	42
URSYZ	NMISC	25	31	37	43
URSXZ	NMISC	26	32	38	44

VISCO108 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "VISCO108 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K, L and O node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- Only the isotropic and Anand material laws (BISO, MISO, ANAND on the **TB** command) are valid for this element.

VISCO108 Product Restrictions

There are no product-specific restrictions for this element.

TRANS109

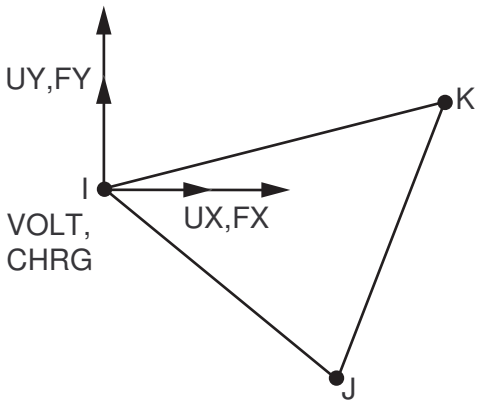
2-D Electromechanical Transducer

MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

TRANS109 Element Description

TRANS109 is a triangular element used in fully coupled electromechanical analysis. It has three degrees of freedom at each node: translation in the nodal x and y directions (UX and UY) and electric potential (VOLT). This element is useful for simulating the electromechanical response of micro-electromechanical systems (MEMS) such as electrostatic comb drives and optical switches. TRANS109 is applicable to large signal static and transient analyses, but not to small signal modal or harmonic analyses (prestressed). See TRANS109 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 TRANS109 Geometry



TRANS109 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "TRANS109 Geometry"*. Element input data includes the relative isotropic permittivity, which must be temperature independent.

TRANS109 uses a segregated solution algorithm to morph the initial mesh. KEYOPT(1) provides the morphing options. If KEYOPT(1) = 0, morphing is unweighted. If KEYOPT(1) = 1, morphing is area weighted.

KEYOPT(3) allows you to input a thickness. If KEYOPT(3) = 0, the thickness is input as unity. If KEYOPT(3) = 3, the thickness is input as the real constant THICKNESS.

The element supports nodal displacements and voltage (**D** command) as well as nodal forces (**F** command). Nodal forces should be input per unit of depth. When applying a nonzero initial starting voltage, use both the **D** command and the **IC** command to input the value.

Free-space permittivity must be set using the **EMUNIT** command. See System of Units for free-space permittivity values and conversion factors useful for micro-electromechanical systems (MEMS).

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

TRANS109 Input Summary

Nodes
I, J, K

Degrees of Freedom
UX, UY, VOLT

Real Constants

None, if KEYOPT (3) = 0
THICKNESS - if KEYOPT (3) = 3

Material Properties
PERX (must be 1.0)

Surface Loads
None

Body Loads
None

Special Features

Large deflection
Large strain

KEYOPT(1)
Select Laplacian morphing:

0 --
Use unweighted morphing

1 --
Use area weighted morphing

KEYOPT(3)
Element behavior:

0 --
Use a thickness of unity

2 --
Plane strain (Z strain = 0.0)

3 --
Use a thickness equal to the real constant THICKNESS

TRANS109 Output Data

The solution output associated with the element is shown in *Table 1, "TRANS109 Element Output Definitions"*.

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* in the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 TRANS109 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J, K	Y	Y
MAT	Material Number	Y	Y
EF:X, Y	Electric field components	Y	Y
EF:SUM	Vector magnitude of EF	Y	Y
D:X, Y	Electric flux density components	Y	Y
D:SUM	Vector magnitude of D	Y	Y

TRANS109 Assumptions and Restrictions

- You cannot use TRANS109 in small signal modal or harmonic analyses.
- You cannot generate a superelement from TRANS109 elements.
- Only isotropic permittivity, independent of temperature, is allowed.
- The element works with 2-D mechanical elements assuming negligible strain in the thickness direction (plane strain).
- TRANS109 will not work with TRANS126, PLANE121, INFIN110, CIRCU94, CIRCU124, or CIRCU125.

TRANS109 Product Restrictions

There are no product-specific restrictions for this element.

INFIN110

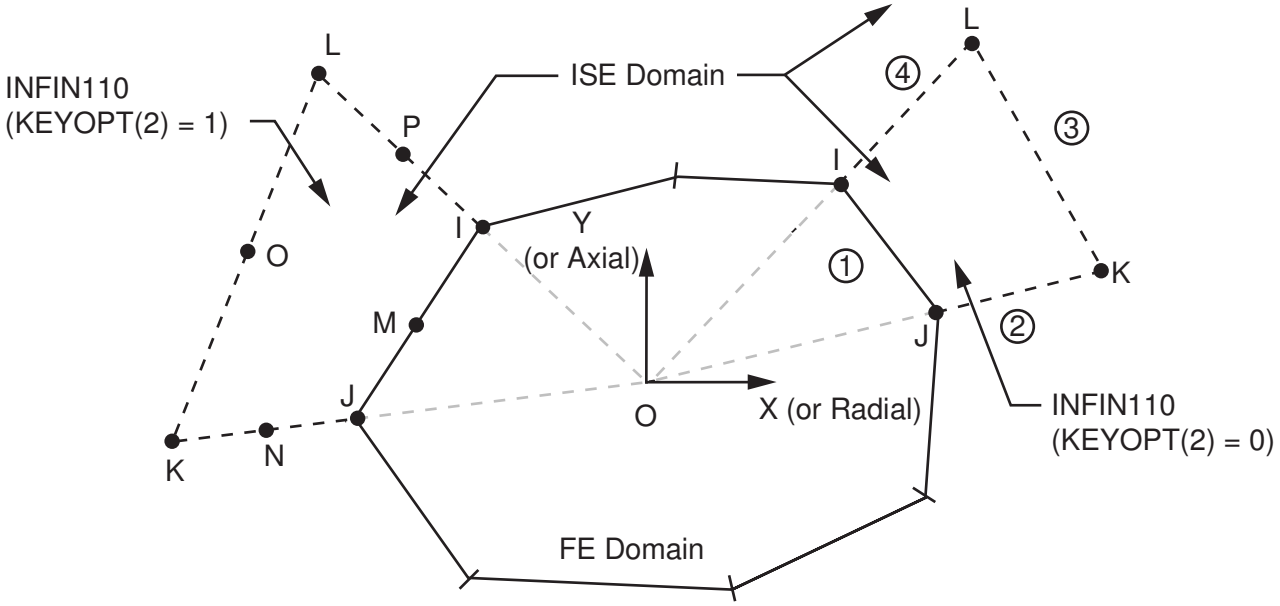
2-D Infinite Solid

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

INFIN110 Element Description

INFIN110 models an open boundary of a 2-D unbounded field problem. A single layer of elements is used to represent an exterior sub-domain of semi-infinite extent. The layer models the effect of far-field decay in magnetic, electrostatic, thermal, or electric current conduction analyses. For information about enclosed elements and analysis types, see *Table 1, "INFIN110 Analyses"*. INFIN110 may be used for planar and axisymmetric analyses. See INFIN110 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 INFIN110 Geometry



INFIN110 Input Data

The geometry, node locations, and the coordinate system for the element are shown in *Figure 1, "INFIN110 Geometry"*. KEYOPT(1) specifies the degree of freedom to be used. KEYOPT(2) specifies whether a 4-node or 8-node element is used.

Only one layer of INFIN110 elements should be used between the finite element model and the exterior (infinite) surface. The nodes may be input starting at any corner node, but the face opposite of the finite element model (the exterior face) must be flagged as an infinite surface. This is usually done by selecting the nodes at the outer surface and issuing the **SF**,all,INF command. The other faces have no meaning. For best results, edges connecting the inner and outer surfaces of the infinite element should be radial from the center of the model.

A summary of the element input is given in *INFIN110 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

INFIN110 Input Summary

Nodes

I, J, K, L (if KEYOPT(2) = 0)
I, J, K, L, M, N, O, P (if KEYOPT(2) = 1)

Degrees of Freedom
Set by KEYOPT(1). See *Table 1, "INFIN110 Analyses"*.

Real Constants
None

Material Properties
See *Table 2, "INFIN110 Material Properties"*.

Surface Loads

Infinite Surface Flags --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads
None

Special Features
None

KEYOPT(1)
Element degrees of freedom. See *Table 1, "INFIN110 Analyses"*.

KEYOPT(2)
Element definition:

0 --
4-node quadrilateral

1 --
8-node quadrilateral

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

Analysis categories are shown in the following table. KEYOPT(1) specifies the element degree of freedom.

Table 1 INFIN110 Analyses

Analysis Category	KEYOPT(1)	DOF Label	Reaction Solution	Enclosed Elements	Analysis Type
Magnetic	0	AZ	Magnetic Current Segment (F label = CSG)	PLANE13 PLANE53	Static Harmonic Transient
Electrostatic	1	VOLT	Electric Charge (F label = CHRG)	PLANE121	Static Harmonic
Thermal	2	TEMP	Heat Flow (F label = HEAT)	PLANE35 PLANE55 PLANE77	Steady-state Transient
Electric Current Conduction	3	VOLT	Electric Current (F label = AMPS)	PLANE230	Steady-state Harmonic Transient

INFIN110 material properties are shown in the following table. Nonzero material properties must be defined. Material properties are defined with the **MP**, **MPDATA** and **EMUNIT** commands.

Table 2 INFIN110 Material Properties

Analysis Category	KEYOPT(1)	Material Properties
Magnetic	0	MUZERO
Electrostatic	1	PERX, PERY, LSST, RSVX, RSVY
Thermal	2	KXX, KYY, DENS, C
Electric Current Conduction	3	RSVX, RSVY, LSST, PERX, PERY

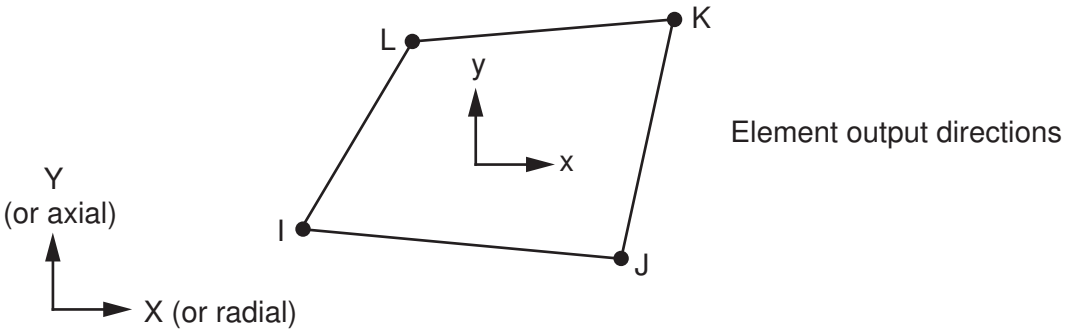
INFIN110 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 3, "Element Output Definitions"*

Several items are illustrated in *Figure 2, "INFIN110 Element Output"*. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 INFIN110 Element Output



The following notation is used in *Table 3, "Element Output Definitions"*:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L (KEYOPT(2) = 0) Nodes - I, J, K, L, M, N, O, P (KEYOPT(2) = 1)	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*4*
MUZERO	Magnetic permeability of free space	*1*	*1*
PERX, PERY	Electric relative permittivity (element coordinates)	*2*	*2*
KXX, KYY	Thermal conductivity (element coordinates)	*3*	*3*

1. If KEYOPT(1) = 0
2. If KEYOPT(1) = 1
3. If KEYOPT(1) = 2
4. Available only at centroid as a ***GET** item.

Table 4, "INFIN110 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "INFIN110 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 3, "Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 4 INFIN110 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUZERO	NMISC	1
PERX	NMISC	1
PERY	NMISC	2
KXX	NMISC	1
KYY	NMISC	2

INFIN110 Assumptions and Restrictions

- The area of the quadrilateral infinite element must be nonzero.
- The element cannot degenerate to a triangle.
- The exterior surface (for example, KL or KOL in *Figure 1, "INFIN110 Geometry"*) of the element must be flagged using the *INF* option on the **SF** family of commands.
- Only one layer of infinite elements can be used on the exterior boundary of the finite element model.
- The lines JK and IL of the infinite element IJKL (in *Figure 2, "INFIN110 Element Output"*) should either be parallel or divergent from each other. That is, the enclosed surface should be convex and the infinite domain must be represented by one layer of infinite elements without overlap or gap. Ideally, the length OJ should equal JK, and OI should equal IL. The point "O" is the "pole" of mapping for the infinite element. The pole is chosen arbitrarily, and may or may not coincide with the origin of the coordinate system. For best results, the poles should be placed at the centers of disturbances (loads). There can be multiple poles for a problem. See the *Theory Reference for ANSYS and ANSYS Workbench* for more about poles.
- Although this element can have 8 nodes (KEYOPT(2) = 1), for theoretical reasons (see the *Theory Reference for ANSYS and ANSYS Workbench*), only 5 nodes are included in the solution.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.
- The infinite elements are not included in solution result displays but may be viewed in element displays [**EPLOT**].
- There are considerations in the application of INFIN110 that will lead to optimal performance in the analysis of your model. These consideration are covered in detail in the *Low-Frequency Electromagnetic Analysis Guide*.
- When used in a model with the higher-order elements PLANE35, PLANE53, PLANE77, PLANE121, and PLANE230 use the higher-order setting for INFIN110 (KEYOPT(2) = 1).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. KEYOPT(6) sets the electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

INFIN110 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

These restrictions apply unless the Emag option is enabled.

- This element does not have magnetic, electrostatic, or electric current conduction capability.
- The AZ and VOLT degrees of freedom are not active. KEYOPT(1) defaults to 2 (TEMP) instead of 0 and cannot be changed.
- The material properties MUZERO, PERX, PERY, RSVX, RSVY, and LSST are not allowed.

ANSYS Emag

- This element has only magnetic, electrostatic, and electric current conduction capability, and does not have thermal capability.
- The only active degrees of freedom are AZ and VOLT. KEYOPT(1) can only be set to 0 or 1.
- The only allowable material properties are MUZERO, PERX, PERY, RSVX, RSVY, and LSST.

INFIN111

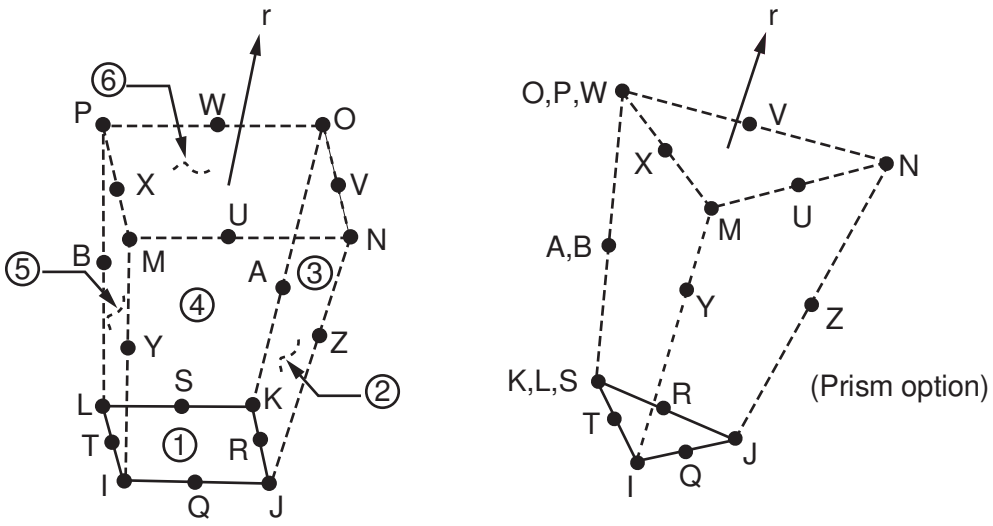
3-D Infinite Solid

MP ME <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

INFIN111 Element Description

INFIN111 models an open boundary of a 3-D unbounded field problem. A single layer of elements is used to represent an exterior sub-domain of semi-infinite extent. The layer models the effect of far-field decay in magnetic, electrostatic, thermal, or electric current conduction analyses. For information about enclosed elements and analysis types, see *Table 1, "INFIN111 Analyses"*. See INFIN111 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 INFIN111 Geometry



INFIN111 Input Data

The geometry, node locations, and the coordinate system for the element are shown in *Figure 1, "INFIN111 Geometry"*. KEYOPT(1) specifies the degree(s) of freedom to be used. KEYOPT(2) specifies whether a 8-node or 20-node element is used.

Only one layer of INFIN111 elements should be used between the finite element model and the exterior (infinite) surface. The nodes may be input starting at any corner node, but the face opposite of the finite element model (the exterior face) must be flagged as an infinite surface. This is usually done by selecting the nodes at the outer surface and issuing the **SF**,all,INF command. The other faces have no meaning. For best results, edges connecting the inner and outer surfaces of the infinite element should be radial from the center of the model.

A summary of the element input is given in *INFIN111 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

INFIN111 Input Summary

Nodes

I, J, K, L, M, N, O, P (if KEYOPT(2) = 0)
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B (if KEYOPT(2) = 1)

Degrees of Freedom
Set by KEYOPT(1). See *Table 1, "INFIN111 Analyses"*.

Real Constants
None

Material Properties
See *Table 2, "INFIN111 Material Properties"*.

Surface Loads

Infinite Surface Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads
None

Special Features
None

KEYOPT(1)
Element degree of freedom. See *Table 1, "INFIN111 Analyses"*.

KEYOPT(2)
Element definition:

0 --
8-node brick

1 --
20-node brick

KEYOPT(6)
Electric charge reaction sign:

0 --
Negative

1 --
Positive

Analysis categories are shown in the following table. KEYOPT(1) specifies the element degree of freedom.

Table 1 INFIN111 Analyses

Analysis Category	KEYOPT(1)	DOF Label	Reaction Solution	Enclosed Elements	Analysis Type
Magnetic	0	MAG	Magnetic Flux (F label = FLUX)	SOLID5 SOLID96 SOLID98	Static
Magnetic	1	AX, AY, AZ	Magnetic Current Segments (F label = CSG)	SOLID62 SOLID97	Static Harmonic Transient
Electrostatic	2	VOLT	Electric Charge	SOLID122	Static

Analysis Category	KEYOPT(1)	DOF Label	Reaction Solution	Enclosed Elements	Analysis Type
			(F label = CHRG)	SOLID123	Harmonic
Thermal	3	TEMP	Heat Flow (F label = HEAT)	SOLID70 SOLID87 SOLID90	Steady-state Transient
Electric Current Conduction	4	VOLT	Electric Current (F label = AMPS)	SOLID231 SOLID232	Steady-state Harmonic Transient

INFIN111 material properties are shown in the following table. Nonzero material properties must be defined. Material properties are defined with the **MP**, **MPDATA** and **EMUNIT** commands.

Table 2 INFIN111 Material Properties

Analysis Category	KEYOPT(1)	Material Properties
Magnetic	0	MUZERO
Magnetic	1	MUZERO
Electrostatic	2	PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ
Thermal	3	KXX, KYY, KZZ, DENS, C
Electric Current Conduction	4	RSVX, RSVY, RSVZ, LSST, PERX, PERY, PERZ

INFIN111 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 3, "INFIN111 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 INFIN111 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P (if KEYOPT(2) = 0); Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B (if KEYOPT(2) = 1)	Y	Y
MAT	Material number	Y	Y

Name	Definition	O	R
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*4*
MUZERO	Magnetic permeability of free space	*1*	*1*
PERX, PERY, PERZ	Electric relative permittivity	*2*	*2*
KXX, KYY, KZZ	Thermal conductivity	*3*	*3*

1. If KEYOPT(1) = 0 or 1
2. If KEYOPT(1) = 2
3. If KEYOPT(1) = 3
4. Available only at centroid as a ***GET** item.

Table 4, "INFIN111 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "INFIN111 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 3, "INFIN111 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 4 INFIN111 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUZERO	NMISC	1
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
KXX	NMISC	1
KYY	NMISC	2
KZZ	NMISC	3

INFIN111 Assumptions and Restrictions

- Assumptions and restrictions listed for INFIN110 elements also apply to INFIN111 elements (see INFIN110 Assumptions and Restrictions).
- There are considerations in the application of INFIN111 that will lead to optimal performance in the analysis of your model. These consideration are covered in detail in the *Low-Frequency Electromagnetic Analysis Guide*.
- When used in a model with the higher-order elements SOLID87, SOLID90, SOLID98, SOLID122, SOLID123, SOLID231, and SOLID232, use the higher-order setting for INFIN111 (KEYOPT(2) = 1).

- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. KEYOPT(6) sets the electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

INFIN111 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

These restrictions apply unless the Emag option is enabled.

- This element does not have magnetic, electrostatic, or electric current conduction capability.
- The MAG, AX, AY, AZ, and VOLT degrees of freedom are not active. KEYOPT(1) defaults to 3 (TEMP) instead of 0 and cannot be changed.
- The material properties MUZERO, PERX, PERY, PERZ, RSVX, RSVY, RSVZ, and LSST are not allowed.

ANSYS Emag

- This element has only magnetic, electrostatic, and electric current conduction capability, and does not have thermal capability.
- TEMP is not allowed as a degree of freedom. KEYOPT(1) can only be set to 0, 1 or 2.
- The only allowable material properties are MUZERO, PERX, PERY, PERZ, RSVX, RSVY, RSVZ, and LSST.

INTER115

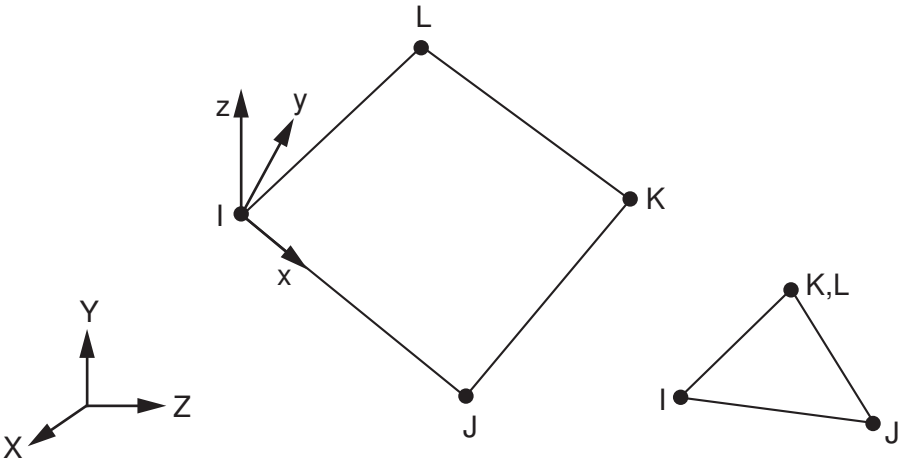
3-D Magnetic Interface

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

INTER115 Element Description

INTER115 is used to couple magnetic vector and scalar potentials in the same analysis. It is a 4-node interface element, capable of collapsing to a 3-node interface element, that is defined on the interface between vector and scalar potential finite element regions. The element has four degrees of freedom per node: AX, AY, AZ and MAG. The element does not have a thickness. It can be used with scalar elements SOLID5, SOLID96, SOLID98, and vector element SOLID97. All of these are 3-D magnetic elements which are used to perform linear, nonlinear, static and dynamic analyses, and coupled field analysis. See INTER115 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 INTER115 Geometry



INTER115 Input Data

The geometry, node locations, and the coordinate system for this element are shown in the *Figure 1, "INTER115 Geometry"*. The element is defined by four nodes and no material property is required. The element x-axis is oriented along the length of the element from node I toward node J.

A summary of the element input is given in *INTER115 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

INTER115 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
AX, AY, AZ, MAG

Real Constants
None

Material Properties
None

Surface Loads
None

Special Features
None

KEYOPTS
None

INTER115 Output Data

The interface element has no output of its own since it is used only to couple vector and scalar potential finite element regions.

INTER115 Assumptions and Restrictions

- The element should not be located at the interface of an air-iron boundary. Such a placement will lead to inaccurate coupling across the vector/scalar potential interface leading to a loss of accuracy in the solution. It is recommended that the interface between vector/scalar domains occur within a single homogenous material (for example, air).
- The normal component of the vector potential at the vector/scalar interface, where the INTER115 element is located, should be set to zero. By setting A x n = 0 at the interface, the Coulomb gauge condition is satisfied and the vector potential solution is assured to be unique. Node rotation [**NROTAT**] can be easily achieved for Cartesian, cylindrical, spherical, and toroidal boundaries from which the normal component can be set to zero.
- In the vector potential region, if a multiply-connected conductor exits, it may not be "cut" by a vector/scalar interface. For example, a closed loop conductor is multiply-connected. The air "hole" inside the conductor cannot contain a vector/scalar interface. In this case, enclose the entire conductor and "hole" region with vector potential elements, then encase the entire region with a scalar domain.
- The INTER115 element cannot lie on a free-surface; however, an element edge may exist at a free-surface.
- When using a scalar source primitive (SOURC36), it is recommended that a small cushion of air surround the primitive before interfacing to a vector potential domain. Having the primitive boundary located at the vector/scalar interface boundary can lead to solution inaccuracies.
- The scalar potential region of a problem using an INTER115 boundary is limited to the Reduced Scalar Potential (RSP) formulation [**MAGOPT**,0]. For accurate solutions, this region should be free from high permeability materials (that is, iron).
- Zero area elements are not allowed. This occurs most often if the elements are not numbered properly.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The 4 nodes defining the element should lie as close as possible to a flat plane; however, a moderate out-of-plane tolerance is permitted so that the element may have a somewhat warped shape. An excessively warped element will produce a warning message. In the case of warping errors, triangular elements should be used (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- Shell element warping tests are described in detail in tables of Applicability of Warping Tests and Warping Factor Limits in the *Theory Reference for ANSYS and ANSYS Workbench*.
- All units used in INTER115 must be expressed in the MKS system.

INTER115 Product Restrictions

There are no product-specific restrictions for this element.

FLUID116

Coupled Thermal-Fluid Pipe

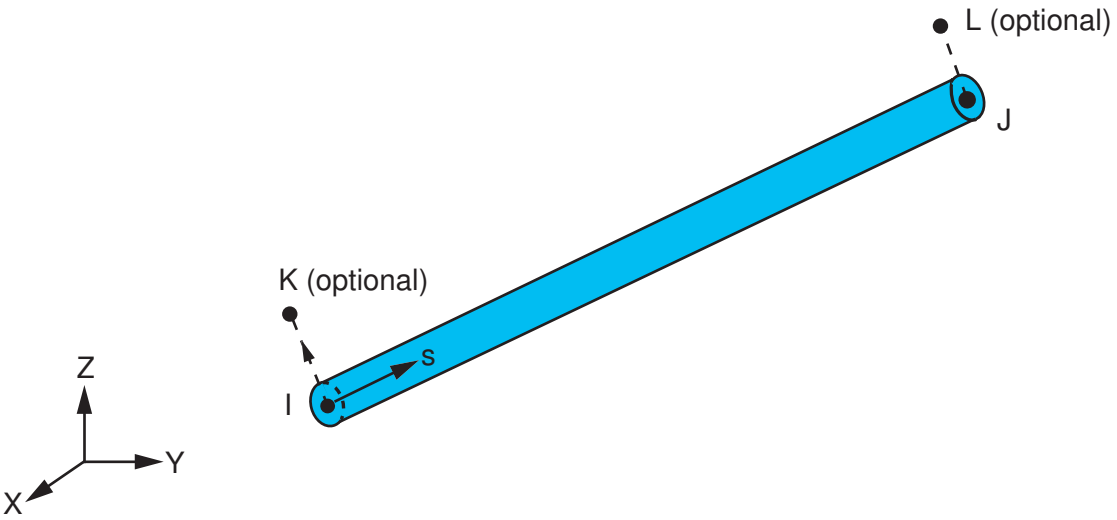
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

FLUID116 Element Description

FLUID116 is a 3-D element with the ability to conduct heat and transmit fluid between its two primary nodes. Heat flow is due to the conduction within the fluid and the mass transport of the fluid. Convection may be accounted for either with additional nodes and convection areas or with surface elements SURF151 and SURF152. In both cases, the film coefficient may be related to the fluid flow rate. The element may have two different types of degrees of freedom, temperature and/or pressure.

The thermal-flow element may be used in a steady-state or transient thermal analysis. If the model containing the thermal-flow element is also to be analyzed structurally, the element should be replaced by an equivalent (or null) structural element. See FLUID116 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID116 Geometry



FLUID116 Input Data

The geometry, node locations, and the coordinate system for this thermal-flow pipe element are shown in *Figure 1, "FLUID116 Geometry"*. The element is defined by two primary nodes, two additional nodes if convection is desired, several real constants (see *Table 1, " FLUID116 Element Real Constants"*), and the material properties. The length L of the element is determined from the two primary node locations.

The material properties can be input as numerical values or as tabular inputs evaluated as a function of pressure, temperature, velocity, time, and location. If temperature or pressure, you need to activate the appropriate pressure or temperature degrees of freedom. Tabular material properties are calculated before the first iteration (i.e., using initial values [**IC**]).

The fluid mass density ρ (Mass/Length^3) is input as property DENS or computed following the ideal gas law if the real constant R_{gas} is present. If KEYOPT(2) = 2, 3, or 4, the convection film coefficient h_f (Heat/Length^2*Time*Deg) is input by the options defined by KEYOPT(4). If KEYOPT(2) = 1, convection surfaces using FLUID116 velocities and other information are stored and can be used by SURF151 or SURF152 and optionally the user programmable

feature USRSURF116 in order to determine film coefficients and bulk temperatures as a function of velocities and other parameters. The input tables are explained in detail in *Table 2, "FLUID116 Empirical Data Table (Optional)"*. The thermal conductivity k_{xx} (Heat/Length*time*Deg) acts in the element longitudinal direction and is input as property KXX. The specific heat c_p (Heat/Mass*Deg or Heat*Length/Force*Time^2*Deg) is input as property C. The fluid viscosity μ is input as property VISC. In an axisymmetric analysis, such as for annular flow, the flow area, the convection areas, and all other input should be on a full 360° basis.

KEYOPT(2) = 3 and 4 are variations of KEYOPT(2) = 2 used to avoid an artificial reduction of the change in temperature in the last element next to an inlet or outlet with no specified temperature. If such an inlet or outlet is at node I, use KEYOPT(2) = 3 and if it is at node J, use KEYOPT(2) = 4. All elements of a run of pipe should use the same KEYOPT, not just the end one. For networks where the usage of KEYOPT(2) is not obvious and the detailed temperature distribution is important, use KEYOPT(2) = 2 with a relatively fine mesh (small elements). The effect of KEYOPT(2) = 3 and 4 could be alternatively achieved by adjusting the convection areas (Real Constants 7 and 8) but it is not as convenient.

The coefficient of friction (input as property MU) is the starting value of the Moody friction factor (f). The friction factor for the first iteration is always assumed to be MU. The smooth-pipe empirical correlations are a function of Reynolds number (Re) and depend on whether the flow is laminar or turbulent (Re>2500). If a friction table is supplied (**TB**,FCON), the friction factor is recomputed each substep from the table (using linear interpolation where necessary). The table is also explained in detail in *Table 2, "FLUID116 Empirical Data Table (Optional)"*.

The word PRES (or TEMP) should be input for the *Lab* variable on the **D** command and the pressure (or temperature) value input for the value. If a nodal heat (or fluid) flow rate is defined with the **F** command, input the word HEAT (or FLOW) for the *Lab* variable and input the flow rate for the value. If temperature is the only degree of freedom, (KEYOPT(1) = 1), input the known flow rate in units of mass/time with the **SFE**,,,HFLUX command instead of the **F** command. Fluid weight effects are activated by specifying a nonzero acceleration and/or rotation vector [**ACEL** and/or **OMEGA**].

When using the rotational speed and slip factor real constants (real constants 7-10 in *Table 1, " FLUID116 Element Real Constants"*), you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %`tabname`%). Also, if using table inputs for rotational speed, either both real constants 7 and 8 should have the same table name reference, or real constant 8 should be unspecified. Similarly, if using table inputs for slip factor, either both real constants 9 and 10 should have the same table name reference, or real constant 10 should be unspecified. Both rotational speed and the slip factor can vary with time and location.

If tabular real constants are used, then any node in a FLUID116 network must refer to a single table name. For correct results, at any node, the table names from different elements must all be the same and a table name cannot be used along with any numerical real constant from a different element.

See Steady-State Thermal Analysis in the *Thermal Analysis Guide* for more information on using table inputs.

Element loads are described in *Section 2.8: Node and Element Loads*. Element body loads may be input as heat generation rates at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

KEYOPT(8) is used for inputting flow losses (see *Table 1, " FLUID116 Element Real Constants"*). Momentum losses in pipes due to bends, elbows, joints, valves, etc., may be represented by a fictitious (equivalent) length of pipe L_a. This equivalent length may be input directly or calculated from an input constant K, the hydraulic diameter D, and the friction factor f.

A summary of the element input is given in *FLUID116 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

FLUID116 Input Summary

Nodes
I, J or I, J, K, L (see KEYOPT(2))

Degrees of Freedom

PRES, TEMP if KEYOPT(1) = 0
TEMP if KEYOPT(1) = 1
PRES if KEYOPT(1) = 2

Real Constants
See *Table 1, " FLUID116 Element Real Constants"*

Material Properties
KXX, C, DENS, MU, VISC, HF

Surface Loads
Imposed flow may be specified with the **SFE**,,,HFLUX command

Body Loads

Heat Generations --
HG(I), HG(J)

Special Features
Nonlinear

KEYOPT(1)
Pressure and temperature degrees of freedom:

0 --
PRES and TEMP degrees of freedom

1 --
TEMP degrees of freedom only

2 --
PRES degrees of freedom only

KEYOPT(2) (used only if KEYOPT(1) = 0 or 1)

0 --
2 nodes and no convection surface or convection information

1 --
2 nodes and convection information passed to SURF151/SURF152

2 --
4 nodes and convection surface logic included with this element, convection area shared between nodes I and J

3 --
4 nodes and convection surface logic included with this element, convection area only at node I

4 --
4 nodes and convection surface logic included with this element, convection area only at node J

KEYOPT(4) (used only if KEYOPT(2) = 2, 3, or 4)
Film coefficient (h_f) definition

0 --
Use **MP**,HF

1 --
Use real constants 9 thru 12 (see *Table 1, " FLUID116 Element Real Constants"*)

2 --
Use **TB**,HFLM for h_f as a function of temperature and average velocity

3 --
Use **TB**,HFLM for h_f as a function of temperature and Reynold's number

4 --
Use **TB**,HFLM for Nu as a function of temperature and Reynold's number ($h_f = K_{xx}$*Nu/diam)

5 --
Use call to User116Hf

KEYOPT(5) (used only if KEYOPT(4) = 0, 2, 3, 4, or 5)
Evaluation of film coefficient:

0 --
Average fluid temperature (TI + TJ)/2

1 --
Average wall temperature (TK +TL)/2

2 --
Average film temperature (TI + TJ + TK + TL)/4

3 --
Differential temperature (TI + TJ)/2 - (TK + TL)/2

KEYOPT(6) (used only if KEYOPT(1) = 0 or 2)
Fluid conductance coefficient definition:

0 --
Use conductance formula

1 --
Use real constant C

2 --
Use **TB**,FCON as a function of temperature and average velocity

3 --
Use **TB**,FCON as a function of temperature and Reynold's number

4 --
Use call to User116Cond

KEYOPT(7) (used only if KEYOPT(6) = 0)
Friction factor calculation:

0 --
Use smooth pipe empirical correlations

1 --
Use **MP**,MU

2 --
Use **TB**,FCON with friction factor being a function of temperature and average velocity

3 --
Use **TB**,FCON with friction factor being a function of temperature and Reynold's number

KEYOPT(8) (used only if KEYOPT(6) = 0)
Flow losses specified by input:

0 --
Use real constant L_a as the additional length

1 --
Use real constant K as loss coefficient

Table 1 FLUID116 Element Real Constants

(Given in the order required for input in the real constant table)			
No.	**Name**	**Definition**	**Units**
1	**D**	Hydraulic diameter.	Length
2	**A**	Flow cross-sectional area.	Length^2
3	**N_c**	Number of flow channels (defaults to 1). If greater than 1, real constants and element output are on a per channel basis.	
4-6		not currently used	
7	**$(A_n)_I$**	If KEYOPT(2) = 1, angular velocity associated with node I. If KEYOPT(2) = 2, 3, or 4, convection area between nodes I and K. Defaults to πDL/2 if KEYOPT(2) = 2, defaults to πDL if KEYOPT(2) = 3 where: L = element length	Length^2
8	**$(A_n)_J$**	If KEYOPT(2) = 1, angular velocity associated with node J. Defaults to value at node I. If KEYOPT(2) = 2, 3, or 4, convection area between nodes J and L. Defaults to πDL/2 if KEYOPT(2) = 2, defaults to πDL if KEYOPT(2) = 4	Length^2
9	**SLIPFAI**	If KEYOPT(2) = 1, slip factor at node I.	
10	**SLIPFAJ**	If KEYOPT(2) = 1, slip factor at node J. Defaults to value at node I.	
9-12	**N1, N2, N3, N4**	(Used if KEYOPT(4) = 1 and KEYOPT(2) = 2, 3, or 4) $Nu = N1 + N2\ Re^{N3}\ Pr^{N4}$ where: Re = Reynolds number (WD/ μA) Pr = Prandtl number (C_pμ/KXX) C_p = specific heat For example, the Dittus-Boelter correlation for full-developed turbulent flow in smooth pipes may be input with N1 = 0.0, N2 = 0.023, N3 = 0.8, and N4 = 0.4 (heating).	
13	**P_p**	Pump pressure.	Force / Length^2
14		Used to compute conductance coefficient C where: $W = C\sqrt{\Delta p}$ Δp = pressure drop	
	C_r	If KEYOPT(6) = 1, conductance coefficient is used to calculate flow. Hence, $C = C_r - \frac{\sqrt{\text{Weight}} * \text{Length}}{\text{Time}}$	

(Given in the order required for input in the real constant table)			
No.	**Name**	**Definition**	**Units**
	L_a	If KEYOPT(6) = 0, and KEYOPT(8) = 0, additional Length of pipe to account for flow losses (for example, valves, orifices, etc.) Hence, $C = A\sqrt{2\rho D/(F(L + L_a))}$ where: ρ = DENS F = friction coefficient	
	K	If KEYOPT(6) = 0 and KEYOPT(8) = 1, this real constant is the loss coefficient K. Hence, $C = A\sqrt{2\rho D/(FL + KD)}$	
15-18		not currently used	
19	**R_{gas}**	Gas constant in ideal gas law ($\rho = p/(R_{gas}T_{abs})$), where T_{abs} is the absolute temperature and p = average pressure. If zero, use ρ as specified by the DENS material property.	Length^2 / Deg * Time^2
20	**V_{DF}**	Viscous damping multiplier. Default 0.0	
21	**C_{ver}**	Units conversion factor for viscous damping. Default = 1.0 $Q_v = V_{DF}C_{ver}F\pi VISC(VELOC)^2L$ = viscous heating for element, with F = 8.0 for laminar and 0.21420 for turbulent flow.	

Note

Real constants 7 through 12 and 20 and 21 are used only if KEYOPT(1) = 0 or 1 and real constants 13 through 19 are used only if KEYOPT(1) = 0 or 2.

The data in *Table 2, "FLUID116 Empirical Data Table (Optional)"* is entered in the data table with the **TB** commands. The curves are initialized by using the **TB** command. The temperature for the first curve is input with the **TBTEMP** command, followed by **TBPT** commands for up to 100 points. Up to 20 temperature-dependent curves (NTEMP = 20 maximum on the **TB** command) may be defined in this manner. The constants (X, Y) entered on the **TBPT** command (two per command).

Table 2 FLUID116 Empirical Data Table (Optional)

Constant	Meaning
Film Coefficient The film coefficient table is initialized with the **TB**,HFLM command. The **TBPT** data are:	
X	Velocity (Length/Time)
Y	Film Coefficient (Heat/(time*area*temp) The velocity may be replaced with the Reynold's number, and the film coefficient may be replaced with the Nusselt number, depending on KEYOPT(4).
Fluid Conductance/Friction Factor The fluid conductance/friction factor is initialized with the **TB**,FCON command. The **TBPT** data are:	
X	Velocity (Length/Time)
Y	Corresponding friction factor value (Dimensionless)
The velocity may be replaced with the Reynold's number, and the friction factor may be replaced with the fluid conductance, depending on KEYOPT(6) and KEYOPT(7).	

FLUID116 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 3, "FLUID116 Element Output Definitions"*

The fluid flow rate is expressed in units of Mass/Time and is positive from node I to node J. In an axisymmetric analysis these flow rates and all other output are on a full 360° basis. The fluid flow rate and the heat flow rate at the nodes may be printed with the **OUTPR** command. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The following notation is used in *Table 3, "FLUID116 Element Output Definitions"*:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 FLUID116 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
XC, YC, ZC	Location where results are reported	Y	*4*
VELOC	Average velocity	Y	Y
RE	Reynolds number	Y	Y
FLOW RATE	Flow rate from node I to node J	Y	Y
HT COND RATE	Heat flow rate from node I to node J due to conduction	*1*	*1*
HT TRANSP RATE	Heat flow rate at node I due to mass transport	*1*	*1*
CONV AREAS (I, J)	Convection areas at nodes I and J	*3*	*3*
HFILM	Film coefficient	*3*	*3*
NUS	Nusselt number	*3*	*3*
PR	Prandtl number	*3*	*3*
HT CONV RATES (I, J)	Heat flow rates from nodes I to K and from nodes J to L due to convection	*3*	*3*
HGVD	Heat generation due to direct input and viscous damping	*1*	*1*
TEMP	Temperature	-	*1*
PUMP PR	Pump pressure	*2*	*2*
FRICTION	Friction factor	*2*	*2*
PRES	Pressure	-	*2*

1. If KEYOPT(1) = 0 or 1
2. If KEYOPT(1) = 0 or 2
3. If KEYOPT(2) = 2, 3, or 4
4. Available only at centroid as a ***GET** item.

Table 4, "FLUID116 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "FLUID116 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 3, "FLUID116 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 4 FLUID116 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
VELOC	NMISC	1	-	-	-	-
RE	NMISC	2	-	-	-	-
FLOW RATE	NMISC	3	-	-	-	-
HEAT COND RATE	NMISC	4	-	-	-	-
HEAT TRANSP RATE	NMISC	5	-	-	-	-
CONV AREA	NMISC	-	6	7	-	-
HFILM	NMISC	8	-	-	-	-
NUS	NMISC	9	-	-	-	-
PR	NMISC	10	-	-	-	-
HEAT CONV RATE	NMISC	-	11	12	-	-
HGVD	NMISC	-	13	14	-	-
TEMP	NMISC	-	15	16	17	18
PUMP PR	NMISC	19	-	-	-	-
FRICTION	NMISC	20	-	-	-	-
PRES	NMISC	-	21	22	-	-

FLUID116 Assumptions and Restrictions

- The element must not have a zero length, so nodes I and J must not be coincident.
- Nodes K and L may be located anywhere in space, even coincident with I and J, respectively.
- D must always be nonzero.
- A defaults to $\pi D^2/4.0$ and is assumed to remain constant for the element.
- Compressibility and flow inertia effects of the fluid are not included in the element formulation.
- If temperatures are degrees of freedom, the resulting unsymmetric matrix requires twice as much memory storage for the solution as other ANSYS elements.
- HF must be nonzero for the four node element.
- MU and DENS must be nonzero if a flow solution is desired and KEYOPT(6) is not zero.

- If the flow is specified at a node also having a specified pressure, the flow constraint is ignored.
- In general, flow is usually specified at the inlet, pressure at the outlet.
- For problems involving pressure specification on inlets and outlets, the solution may converge too soon (i.e., the PRES degree of freedom has converged but FLOW has not due to a loose convergence criterion). Be sure to check your results carefully. To force more iterations, you can tighten the convergence criteria (i.e., **CNVTOL**,flow,,1e-30), or you can specify a nonzero initial condition on pressure, which could be an average of the specified inlet and outlet pressures (i.e., **IC**,all,pres,pavg). You can use both options together; however, ANSYS recommends using a nonzero initial condition. Tightening the convergence requires you to estimate a suitable tolerance.
- More substeps are required for convergence as the flow approaches zero.
- See the **CNVTOL** command for convergence control.
- If pressure is a degree of freedom, the element is nonlinear and requires an iterative solution.

FLUID116 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

These restrictions apply when using this element with the ANSYS Professional product.

- The PRES degree of freedom (KEYOPT (1) = 0, 2) is not available with the ANSYS Professional product.

SOLID117

3-D 20-Node Magnetic Solid

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID117 Element Description

SOLID117 models 3-D magnetic fields. The element is defined by 20 nodes. It has 12 edge-flux DOFs (AZ), one at each midside node. The eight corner nodes carry the time-integrated electric potential DOF, VOLT (classical formulation) or the electric potential DOF, VOLT (solenoidal formulation).

SOLID117 is based on the edge-flux formulation, and applies to the low-frequency magnetic field analyses: magnetostatics, eddy currents (AC time harmonic and transient analyses). The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves for static and transient analyses. See SOLID117 in the *Theory Reference for ANSYS and ANSYS Workbench*, as well as 3-D Magnetostatics and Fundamentals of Edge-based Analysis, 3-D Harmonic Magnetic Analysis (Edge-Based), and 3-D Transient Magnetic Analysis (Edge-Based), in the *Low-Frequency Electromagnetic Analysis Guide*, for details about using this element's different formulations.

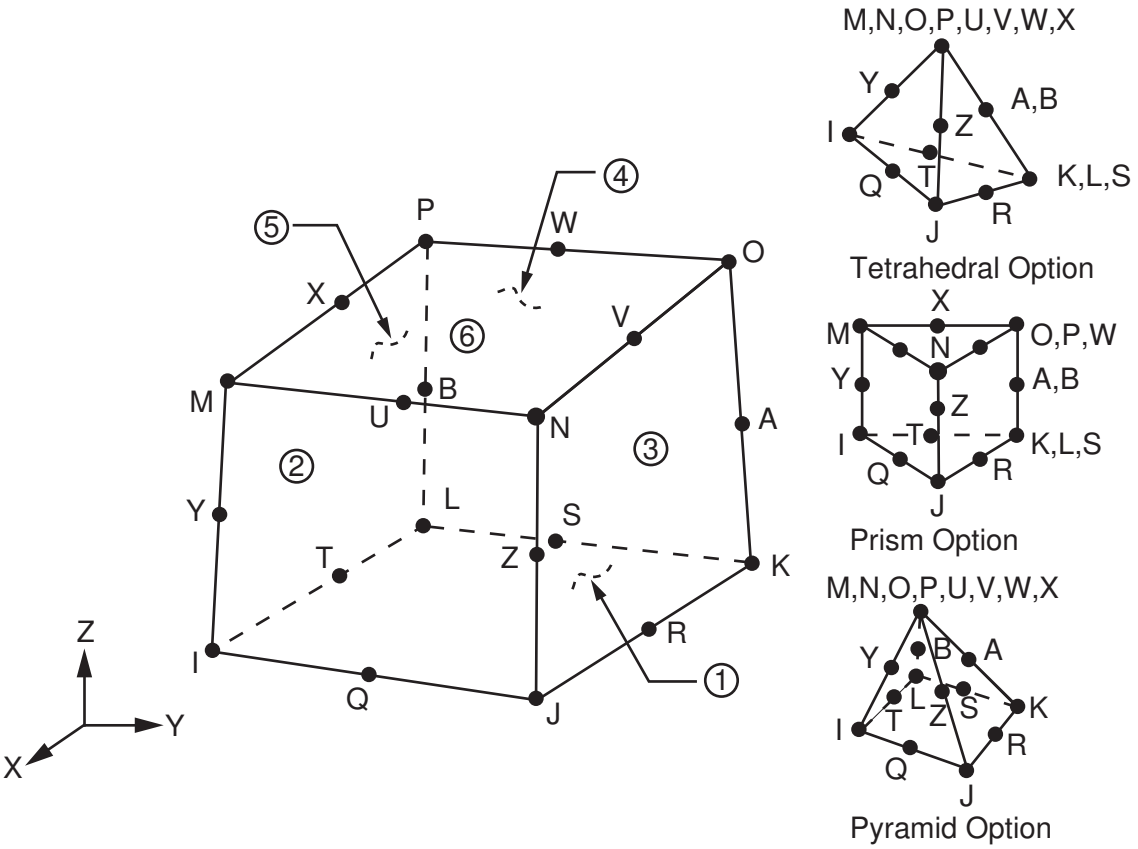
SOLID117 has two formulation options for non-eddy current regions: classical and solenoidal.

The classical formulation is used to model air, iron, or nonferrous materials, and permanent magnets. Current for the classical formulation can be defined directly as body loads using the **BFE**,,JS command, or with the SOURC36 element (recommended method for stranded conductors). Using the SOURC36 method has limited applicability to harmonic and transient analyses. For harmonic analyses, only real current is used (no imaginary). For transient analyses, because the current is applied as a real constant that cannot be changed during a load step, it has limited applicability.

Use the solenoidal formulation to model solid conductors without eddy current effects. The SOLID117 solenoidal formulation uses voltage-fed loading, current-fed loading, or circuit coupling capabilities with the CIRCU124, CIRCU125, and TRANS126 elements. The nonlinear symmetric solenoidal formulation is applicable to static and transient analyses. The linear unsymmetric solenoidal formulation is applicable to harmonic analysis. The electric scalar potential VOLT DOF is not time-integrated. For more information, see 3-D Circuit Coupled Solid Source Conductor in the *Coupled-Field Analysis Guide*.

Eddy currents in solid conductors use the edge element method with time-integrated electric potential VOLT. See SOLID117 in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on the theoretical formulation.

Figure 1 SOLID117 Geometry



SOLID117 Input Data

Figure 1, "SOLID117 Geometry" shows the geometry, node locations, and the coordinate system for this element. The element is defined by 20 nodes and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID117 Geometry"*. The positive orientation of an edge points from lower to higher corner nodes of the edge.

SOLID117 Real Constants

The real constants associated with SOLID117 apply when considering velocity effects of a conducting body (KEYOPT(2) = 1), and start in real constant location nine (one through eight are blank).

VELOX, VELOY, and VELOZ are the velocity components in the global Cartesian coordinate system X, Y, and Z direction. OMEGAX, OMEGAY, OMEGAZ describe the angular (rotational) velocity (Hz, cycles/sec) about the global Cartesian coordinate system X, Y, and Z-axes. The real constants XLOC, YLOC, ZLOC specify the pivot point location of the rotating body.

SOLID117 Units

Specify the type of units (MKS or user defined) using the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO (free-space permeability). The **EMUNIT** defaults are MKS units and MUZERO = $4\pi \times 10^{-7}$ Henries/meters.

Note

The minimum allowable element edge length for this element is 1.0e-6. Choose units accordingly if model dimensions are on the order of microns.

SOLID117 Material Properties

In addition to MUZERO, orthotropic relative permeability is available; specify it via the MURX, MURY, and MURZ material options.

Specify nonlinear magnetic B-H properties with the **TB** command. You can specify nonlinear orthotropic magnetic properties with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. For isotropic nonlinear behavior, you do not need to specify any relative permeability. You can specify only one B-H curve per material.

You can specify orthotropic resistivity through RSVX, RSVY, and RSVZ material property labels. MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The vector components MGXX, MGYY, and MGZZ determine the direction of polarization. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

SOLID117 Loads

You define nodal loads using the **D** and the **F** commands for solid conductors. With the **D** command, only the *Lab* = VOLT option is valid. Use the *VALUE* variable to define the time-integrated electric potential (classical formulation) or electric potential (solenoidal formulation). With the **F** command, the *Lab* variable corresponds to the force (Amps) and *VALUE* corresponds to the value (current) applied with respect to the VOLT DOF.

For stranded conductors, use the **BFE** command to prescribe source current density body loads (classical formulation) based on their value at the element's centroid location. Alternatively, use the **BFV** command to apply source current density body loads to volumes. The vector components of the current density are with respect to the element coordinate system (see *SOLID117 Assumptions and Restrictions* for solenoidal restriction). If you use KEYOPT(1) = 0, you can also define the current using SOURC36 elements. This option is only valid for stranded coils with no eddy current.

For edge-based analyses, the label AZ (when set to zero) applies the flux-parallel boundary condition. No prescription is required to set flux-normal, because it is the natural boundary condition. In the rare case when the AZ = 0 condition is not general enough for flux-parallel conditions, you can prescribe constraints using individual **D** commands.

When using loads from SOURC36 elements, use the MMF option on the **MAGOPT** command to specify how the SOURC36 load is treated. Use the distributed method (default) to apply a constant current density through the cross section of the current source. Use the filament method to concentrate the current in the center of the source.

The calculated field outside the current source region is essentially the same with either method. Inside the current source, however, the two methods provide different field distributions. This difference may lead to different inductance coefficients.

When the internal inductance (due to the energy of the field inside the conductor) is significant compared to the external inductance, the distributed method is recommended together with detailed mesh inside the source region. Typical applications are air coil inductors without iron.

When the internal inductance is negligible, the filament method is recommended with a course mesh inside the conductors. Typical applications are actuators and electric machines when the magnetic flux is conducted by iron and interaction takes place in the air gap. See the section Performing a Static Edge-Based Analysis in the *Low-Frequency Electromagnetic Analysis Guide* for more details on using the distributed and filament options.

You can use constraint equations to define cyclic symmetry.

See 3-D Magnetostatics and Fundamentals of Edge-based Analysis in the *Low-Frequency Electromagnetic Analysis Guide* for more information on loading in an edge-based analysis.

SOLID117 Flags

Section 2.8: Node and Element Loads describes element loads.

For static analyses, no flags are required. Any existing flags are ignored. Select the nodes and elements for which you want to summarize the electromagnetic force, and issue the **EMFT** or **FMAGSUM** command.

For harmonic or transient analyses, you can specify Maxwell force flags on the element faces indicated by the circled numbers in *Figure 1, "SOLID117 Geometry"* using the **SF** and **SFE** commands. To identify surfaces at which magnetic forces are to be calculated, use the MXWF label on the surface load commands. (No value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. You should apply the surface flag to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Use the **FMAGBC** command to automatically apply Maxwell surface flags to a named element component.

SOLID117 Field-Coupling

You can use the **LDREAD** command to read electromagnetic forces and Joule heating in a subsequent structural analysis with companion structural elements or heat transfer with companion thermal elements. When using the classical formulation, you can read element current densities from an electric current conduction analysis using the **LDREAD** command. In addition, you can specify the temperature (for material property evaluation only).

Note

Force coupling is supported only for first order brick elements such as SOLID45.

In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

SOLID117 Gauging

The ANSYS program gauges the problem domain automatically at solution time, using a Tree gauging technique. (See the description of the **GAUGE** command.) This produces additional constraints on nodes in the model by setting AZ to zero. The additional constraints are removed after solution. Thus, gauging is transparent to users.

The table below summarizes the element input. *Section 2.1: Element Input* provides a general description of element input.

SOLID117 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
See KEYOPT(1).

Real Constants

For KEYOPT(2) = 1:
(Blank), (Blank), (Blank), (Blank), (Blank), (Blank),
(Blank), (Blank), VELOX, VELOY, VELOZ, OMEGAX,
OMEGAY, OMEGAZ, XLOC, YLOC, ZLOC
See *Table 1, "SOLID117 Real Constants"* for a description of the real constants

Material Properties

MUZERO, MURX, MURY, MURZ, RSVX, RSVY,
RSVZ, MGXX, MGYY, MGZZ plus BH data table (see *Section 2.5: Data Tables - Implicit Analysis*)

Surface Loads

Maxwell Force Flags -- (harmonic and transient analyses only; ignored for static analyses)

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Source Current Density --
If KEYOPT(1) = 0: (See *SOLID117 Assumptions and Restrictions* for solenoidal restriction)

JSX(I), JSY(I), JSZ(I), PHASE(I), JSX(J), JSY(J),
JSZ(J), PHASE(J), JSX(K), JSY(K), JSZ(K), PHASE(K),
JSX(L), JSY(L), JSZ(L), PHASE(L), JSX(M), JSY(M),
JSZ(M), PHASE(M), JSX(N), JSY(N), JSZ(N), PHASE(N),
JSX(O), JSY(O), JSZ(O), PHASE(O), JSX(P), JSY(P),
JSZ(P), PHASE(P)

EF --
EFX, EFY, EFZ. See *SOLID117 Assumptions and Restrictions*.

Special Features
Requires an iterative solution if nonlinear material properties are defined

KEYOPT(1)
Element degree of freedom and formulation selection:

Classical Formulation

0 -- Stranded Conductors
AZ degrees of freedom

With KEYOPT(1) = 0, you can use either the classical formulation and apply loads manually using **BFE**,,JS, or you can apply source current loads using SOURC36 elements. See *Chapter 6: 3-D Magnetostatics and Fundamentals of Edge-Based Analysis* in the *Low-Frequency Electromagnetic Analysis Guide* for more information on using both of these options.

1 -- Solid Conductors (Eddy Current)
AZ, VOLT degrees of freedom (time-integrated VOLT); harmonic and transient analyses only

Solenoidal Formulation

5 -- Solid Conductors (DC Current)
AZ, VOLT degrees of freedom: nonlinear symmetric solenoidal formulation applicable to static and transient analyses.

6 -- Solid Conductors (DC Current)
AZ, VOLT degrees of freedom: linear unsymmetric solenoidal formulation applicable to harmonic analyses.

KEYOPT(2)
Element conventional velocity:

0 --
Velocity effects ignored

1 --
Conventional velocity formulation (not available if KEYOPT(1) = 0, 2, 3, or 4)

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

Table 1 SOLID117 Real Constants

No.	Name	Description
1, ..., 8	(Blank)	--
9, 10, 11	VELOX, VELOY, VELOZ	Velocity in X, Y, and Z-directions
12, 13, 14	OMEGAX, OMEGAY, OMEGAZ	Angular velocity about X, Y, and Z-axes
15, 16, 17	XLOC, YLOC, ZLOC	Pivot point X, Y, Z-locations

SOLID117 Solution Considerations

You can choose the analysis type (static, transient, or harmonic) using the **ANTYPE** command.

In a harmonic analysis, the output field quantities are peak values. The ANSYS program performs a complex solution and computes two sets of data: real and imaginary. The measurable field quantities can be computed as the real step with a cosine time change minus the imaginary step with a sine time change. You can set the frequency of the time change via the **HARFRQ** command. The measurable magnetic energy, the Joule heat, and average Lorentz forces can be computed as a sum of the calculated real and imaginary data. RMS time averaging is applied to Joule heat and average forces. Energy is computed to reflect peak values. The *Theory Reference for ANSYS and ANSYS Workbench* details complex formalism for harmonic analyses.

Use the **GAUGE** command to control automatic gauging of the problem domain. The default is Tree gauging, which removes constraints after the **SOLVE** or **MAGSOLV** command is issued.

To choose a solver, specify one on the **EQSLV** command. The sparse solver is recommended.

To define transient and nonlinear options, you can use the **MAGSOLV** command (which defines the options and solves the problem automatically) or you can issue the **CNVTOL**, **NEQIT**, and **NSUBST** commands. Use the **OUTPR** command to control printout and the **OUTRES** command to control database storage.

SOLID117 Output Data

The solution output associated with the element is in two forms:

- Nodal DOFs included in the overall nodal solution
- Additional element output as shown in *Table 2, "SOLID117 Element Output Definitions"* and *Table 3, "SOLID117 Miscellaneous Element Output"*

The element output directions are parallel to the element coordinate system. *Section 2.2: Solution Output* provides a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Table 2, "SOLID117 Element Output Definitions" uses the following notation:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SOLID117 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B	Y	Y
MAT	Material number	Y	Y
VOLU	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Input temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
MUX, MUY, MUZ	Magnetic secant permeability (B/H)	*1*	*1*
H:X, Y, Z, SUM	Magnetic field intensity components and vector magnitude	*1*	*1*
B:X, Y, Z, SUM	Magnetic flux density components and vector magnitude	*1*	*1*
JS:X, Y, Z	Source current density components in the global Cartesian coordinate system, valid for static analysis only	*1*	*1*
JT(X, Y, Z)	Total current density components in the global Cartesian coordinate system	*1*	*1*
JHEAT	Joule heat generation per unit volume	*1*	*1*
FJB(X, Y, Z)	Lorentz magnetic force components (harmonic and transient analyses only)	*1*	-
FMX(X, Y, Z)	Maxwell magnetic force components (harmonic and transient analyses only)	*1*	-
FVW(X, Y, Z)	Virtual work force components (harmonic and transient analyses only)	*1*	*1*
FMAG:X, Y, Z, SUM	Electromagnetic force	-	*1*
MRE:	Magnetic Reynold's Number	*3*	*3*

1. The solution is output if its value is not zero. The element solution is at the centroid.

Note

JT represents the total measurable current density that is induced in a conductor, including eddy current effects, and velocity effects if calculated. Components are also available: JS component from VOLT, JE component from A, JT = JS + JE. In a static analysis, JS represents the source current density.

For harmonic analysis, joule losses (JHEAT), forces (FJB(X, Y, Z), FMX(X, Y, Z), FVW(X, Y, Z)) represent time-average values. These values are stored in both the "Real" and "Imaginary" data sets. The macros **POWERH** and **FMAGSUM** can be used to retrieve this data.

2. Available only at centroid as a ***GET** item.
3. Available only with harmonic or transient analyses. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For harmonic analysis, joule losses (JHEAT) and forces (FJB(X, Y, Z), FMX(X, Y, Z), FVW(X, Y, Z)) represent time-average values. These values are stored in both the "Real" and "Imaginary" data sets. The macros **POWERH** and **FMAGSUM** can be used to retrieve this data.

Inductance values (EIND) obtained for KEYOPT(1) = 2, 3, or 4 are only valid under the following conditions: the problem is linear (constant permeability), there are no permanent magnets in the model, and only a single coil exists in the model.

Table 3 SOLID117 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	*1*	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 4, "SOLID117 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 4, "SOLID117 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 2, "SOLID117 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 4 SOLID117 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Source Current Density (static analysis), or time-varying component due to electric potential (VOLT)		
JSX	SMISC	1
JSY	SMISC	2
JSZ	SMISC	3
JSSUM	SMISC	4
Secant Permeability B/H		
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
Lorentz Forces/Virtual Work Force		
FVWX	NMISC	4 (*2*)
FVWY	NMISC	5 (*2*)
FVWZ	NMISC	6 (*2*)
FVWSUM	NMISC	7 (*2*)
Total (Measurable) Current Density		
JTX	NMISC	12
JTY	NMISC	13
JTZ	NMISC	14
JTSUM	NMISC	15
Differential Permeability dB/dH		
DMUX	NMISC	18
DMUY	NMISC	19
DMUZ	NMISC	20
VX (*1*)	NMISC	21
VY (*1*)	NMISC	22
VZ (*1*)	NMISC	23

1. VX, VY, and VZ are available only with harmonic and transient analyses.
2. For static analyses, these values are Lorentz forces. For all other analysis types, these values are virtual work forces. See Calculating Magnetic Force and Torque in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

SOLID117 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side.
- An error occurs frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID117 Geometry"* or may have the planes IJKL and MNOP interchanged.
- Midside nodes may not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

- The prism option is available for all standard mesh extrusion operations if the triangular source elements have straight edges. Issuing **MSHMID**,1 places the midside nodes so the triangular element edges are straight.
- The continuity equation must be satisfied for a proper electromagnetic analysis as explained in the *Theory Reference for ANSYS and ANSYS Workbench*. For this reason the source current density, JS, must be solenoidal (that is, $\nabla \cdot JS = 0$). You should verify that this condition is satisfied when prescribing the source current density load. If this condition is not satisfied SOLID117 can produce erroneous solutions without warning. Refer to Performing a Static Edge-based Analysis in the *Low-Frequency Electromagnetic Analysis Guide* for information on how to obtain solenoidal currents when the source current density is not constant. If you use a solenoidal formulation (KEYOPT(1) = 5 or 6), ANSYS will compute the current density. The solenoidal formulations are applicable to voltage and circuit coupled problems as well.
- You cannot use this element in a nonlinear harmonic analysis.
- When this element does not have the VOLT degree of freedom for a harmonic or transient analysis, it acts as a stranded conductor.
- Permanent magnets are not permitted in a harmonic analysis.
- The VOLT degree of freedom (KEYOPT(1) = 1) is required in all non-source regions with a specified nonzero resistivity. This allows eddy currents to be computed.
- Other elements in the model are restricted to the AX, AY, AZ, or VOLT degrees of freedom only. No other DOFs are allowed in the model when running an edge-based analysis using SOLID117.
- For specific recommendations and restrictions on current loading, see 3-D Magnetostatics and Fundamentals of Edge-Based Analysis and 3-D Nodal-Based Analyses (Static, Harmonic, and Transient) in the *Low-Frequency Electromagnetic Analysis Guide*.
- You cannot use this element with circuit element CIRCU124 if KEYOPT(1) = 0 or 1.
- For velocity effects (KEYOPT(2) = 1), note the following restrictions:
  - Velocity effects are valid only for the AZ, VOLT DOF option.
  - Velocity effects cannot be included in a static analysis. To simulate a static analysis, execute a harmonic analysis at a very low frequency and retrieve the "real" results for the solution.
  - Velocity effects are available only in a linear analysis.
  - Velocity effects are valid only for isotropic resistivity.
  - Solution accuracy may degrade if the element magnetic Reynolds number is much greater than 1.0. (See the discussion of magnetic fields in the *Low-Frequency Electromagnetic Analysis Guide*.)
- For harmonic and transient (time-varying) analyses the following restrictions apply:
  - You should use hexahedral elements in current carrying regions because hexahedral elements are more accurate than the degenerate shaped elements (tetrahedral and pyramid). You can expect comparable accuracy with all element shapes in noncurrent carrying regions.
  - Time-average Lorentz forces are calculated automatically for all current carrying elements. You cannot calculate Maxwell or virtual work forces.
  - You cannot use this element with other nodal-based electromagnetic elements (for example, SOLID5, SOLID96, SOLID97, SOLID98, SOURC36, INFIN111, INTER115).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- The solenoidal formulations do not model eddy current effects.

- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- You cannot use **NFORCE**, **PRRFOR**, or **PRRSOL** with SOLID117.
- You should avoid using pyramid shapes in critical regions.
- Warping should be minimized when using this element.
- You cannot use coupling with this element.

SOLID117 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

HF118

2-D High-Frequency Quadrilateral Solid

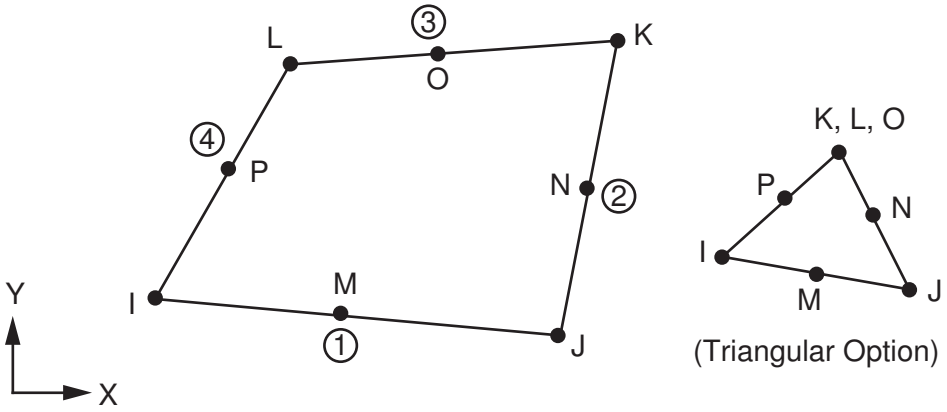
MP <> <> <> <> <> <> <> <> EH <> PP <>
Product Restrictions

HF118 Element Description

HF118 is a high-frequency element which models 2-D electromagnetic fields and waves governed by the full set of Maxwell's equations in linear media. It is based on a full-wave formulation of Maxwell's equations in terms of the time-harmonic electric field E (exponent jωt dependence assumed). See *Section 5.5: High-Frequency Electromagnetic Field Simulation* in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on Maxwell's equations and full-wave formulations, respectively.

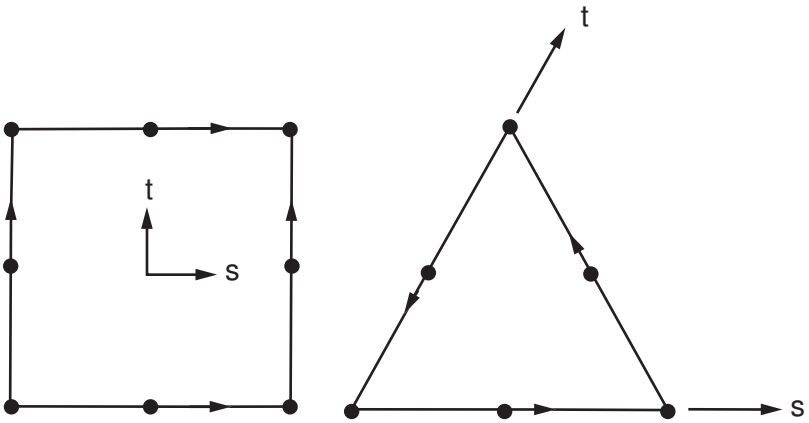
HF118 applies only to modal analyses. You can use it to compute dispersion characteristics of high-frequency transmission lines, including cutoff frequencies and propagating constants for multiple modes. It is a mixed node-scalar edge-vector element. Physically the AX DOF mean the projection of the electric field E on edges and faces. The AX DOF also represents the E_z component of the electric field E at the nodes.

Figure 1 HF118 Geometry



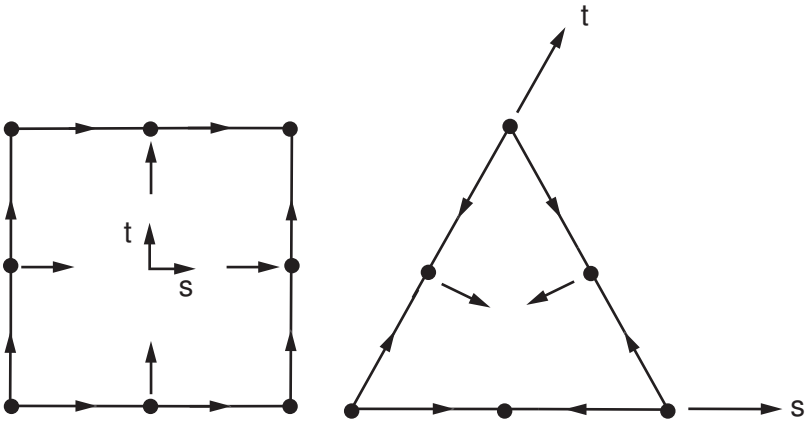
A first order or second order element option is available using KEYOPT(1). The first order quadrilateral and triangular elements have one AX DOF on each edge and at each corner node. The total number of DOFs is 8 for a first order quadrilateral element {1 (4 edges) + 1 (4 corner nodes)} and 6 for a first order triangular element {1 (3 edges) + 1 (3 corner nodes)}.

Figure 2 HF118 First Order Element



The second order quadrilateral element has two AX DOFs on each edge, four AX DOFs on the face for the tangential component of the electric field E, and one AX DOF at each corner and midside node. The total number of DOFs is 20 for a second order quadrilateral element {2 (4 edges) + 4 (1 face) + 1 (8 nodes)}. The second order triangular element has two AX DOFs on each edge, two AX DOFs on the face for the tangential component of the electric field E, and one AX DOF at each corner and midside node. The total number of DOFs is 14 for a second order triangular element {2 (3 edges) + 2 (1 face) + 1 (6 nodes)}.

Figure 3 HF118 Second Order Element



HF118 Input Data

Figure 1, "HF118 Geometry" shows the geometries, node locations, and the coordinate system for the element. The element supports two geometric shapes: a quadrilateral shape with a degeneracy to a triangular shape.

The only unit system supported for high-frequency analysis is the MKS unit, where the free-space permeability MUZERO = $4\pi \times 10^{-7}$ H/m and the free-space permittivity PER0 = 8.854×10^{-12} F/m (see the **EMUNIT** command).

HF118 requires two sets of material constants; that is, *relative* permeability and permittivity tensors (in the element coordinate system if any). To specify a diagonal *relative* permeability tensor use the MURX, MURY, and MURZ labels on the **MP** command. Use the **TB**,MUR and **TBDATA** commands to specify the terms of an anisotropic permeability matrix. To specify a diagonal *relative* permittivity tensor use the PERX, PERY, and PERZ labels on the **MP** command. Use the **TB**,DPER and **TBDATA** commands to specify the terms of an anisotropic permittivity matrix.

To define nodal constraints on geometric nodes, use the **D** command. With the **D** command, the *Lab* variable corresponds to the only degree of freedom AX and the *VALUE* corresponds to the AX value. AX is not the x component in the global Cartesian coordinate system. In most cases, the AX value is zero, which corresponds to a perfect electric conductor or "Electric Wall" (tangential component of $\bar{E} = 0$) condition. If both end nodes and the mid-node on an element edge are constrained, DOFs on the edge are also constrained. Similarly, if all edges on an element face are constrained, DOFs on the face are also constrained. If you leave the nodes on a surface unspecified, the boundary assumes a "Magnetic Wall" condition (tangential component of $\bar{H} = 0$).

To define constraints on lines, use the **DL** command. The *Lab* variable corresponds to the degree of freedom AX and the *Value1* corresponds to the AX value. Upon initiation of the solution calculations (**SOLVE**), the solid model DOF constraints transfer automatically to the finite element model.

HF118 Input Summary summarizes the element input. *Section 2.1: Element Input* in the *Elements Reference* provides a general description of element input.

HF118 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
AX

Real Constants
None

Material Properties

MUZERO, MURX, MURY, MURZ, EPZRO, PERX, PERY, PERZ, MUR, DPER

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Element polynomial order selection:

0,1 --
First order element

2 --
Second order element

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Centroid point printout

2 --
Nodal field printout

HF118 Solution Considerations

For modal analysis, choose a frequency shift point just below the anticipated eigenfrequency of interest (via the **MODOPT** command). Select an upper end frequency as well. Use the *Method* argument to choose the Block Lanczos solver. To visualize the electric and magnetic field modes, use the **MXPAND** command to expand the mode shapes.

HF118 Output Data

The solution output associated with this element is in two forms:

- Degrees of freedom (AX) included in the overall nodal solution
- Additional element output as shown in *Table 1, "HF118 Element Output Definitions"*

The element output direction is parallel to the element coordinate system (if any). *Section 2.2: Solution Output* in the *Elements Reference* provides a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 HF118 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, ..., B	Y	Y
MAT	Material number	Y	Y
VOLU	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
LOC	Output location	*1*	-
MURX, MURY, MURZ	Relative permeability	*1*	-
PERX, PERY, PERZ	Relative permittivity	*1*	-
EF:X, Y, Z	Electric field intensity E	*1*	Y
EF:SUM	Magnitude of E	*1*	-
H:X, Y, Z	Magnetic field intensity H	*1*	Y
H:SUM	Magnitude of H	*1*	-
JHEAT	Joule heat generation per unit volume (time-average value)	-	-
PX, PY, PZ	Pointing vector (time-average value)	-	Y
ALPHA	Real part of propagating constant	-	Y
BETA	Imaginary part of propagating constant	-	Y
PF	Poynting flow through cross-section	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** command item.

Table 2 HF118 Miscellaneous Element Output

Description	Names of Items Output	O	R
Centroid Point Solution	LOC, MUX, MUY, MUZ, PERX, PERY, PERZ, E, ESUM, H, HSUM	*1*	-
Nodal Electric and Magnetic Field Solutions	E, ESUM, H, HSUM	*2*	-

1. Output at each centroid point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "HF118 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "HF120 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "HF120 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 HF118 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	I
PX	NMISC	1
PY	NMISC	2
PZ	NMISC	3
ALPHA	NMISC	4
BETA	NMISC	5
PF	NMISC	6

HF118 Assumptions and Restrictions

- HF118 is only applicable to modal analyses.
- The element must not have a zero volume.
- The required material properties (MURX, MURY, MURZ, PERX, PERY, PERZ) must be input as relative values.
- Midside nodes must not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- The second order element is not available for the solution of propagating constant with fixed frequency.

HF118 Product Restrictions

There are no product-specific restrictions for this element.

HF119

3-D High-Frequency Tetrahedral Solid

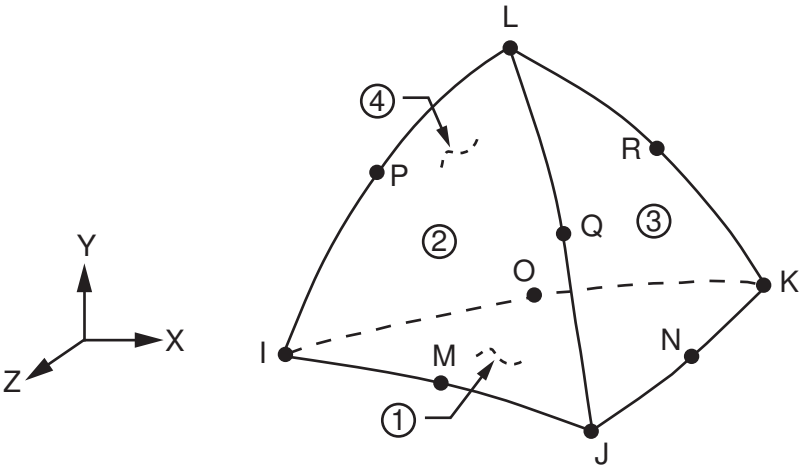
MP <> <> <> <> <> <> <> <> EH <> PP <>
Product Restrictions

HF119 Element Description

HF119 is a high-frequency tetrahedral element which models 3-D electromagnetic fields and waves governed by the full set of Maxwell's equations in linear media. It is based on a full-wave formulation of Maxwell's equations in terms of the time-harmonic electric field E (exponent $j\omega t$ dependence assumed). A companion brick element, HF120, has similar full-wave capability. See HF119 in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on Maxwell's equations and full-wave formulations, respectively.

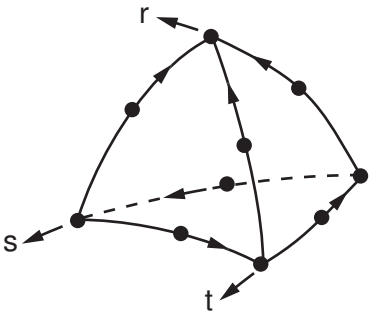
HF119 applies to the full-harmonic and modal analysis types, but not to the transient analysis type. It is defined by up to 10 geometric nodes with AX DOF on element edges and faces. The physical meaning of the AX DOF in this element is a projection of the electric field E on edges and faces.

Figure 1 HF119 Geometry



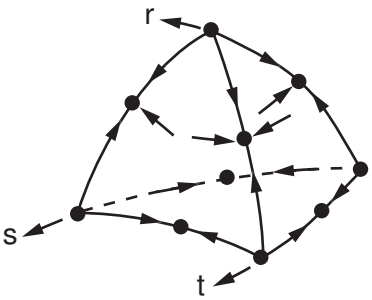
A first order or second order element option is available using KEYOPT(1). The first order element has one AX DOF on each edge for a total of 6 DOFs.

Figure 2 HF119 First Order Element



The second order element has two AX DOFs on each edge and face for a total of 20 DOFs {2(6 edges) + 2(4 faces)}.

Figure 3 HF119 Second Order Element



HF119 Input Data

Figure 1, "HF119 Geometry" shows the geometries, node locations, and the coordinate system for the element.

The only unit system supported for high-frequency analysis is the MKS unit, where the free-space permeability MUZERO = 4π x 10^{-7} H/m and the free-space permittivity PER0 = 8.854 x 10^{-12} F/m (see the **EMUNIT** command).

KEYOPT(4) provides options for the element formulation. KEYOPT(4) = 0 activates the normal full-wave formulation, which solves for the total field. KEYOPT(4) = 1 activates the perfectly matched layers (PML) formulation, which absorbs the field at the open boundary or at the port of a waveguide. KEYOPT(4) = 2 activates the scattering formulation, which is only required in the regions of a domain receiving a reflected wave from an imposed soft source magnetic field excitation (**BF**,,H option).

HF119 requires two sets of material constants; that is, *relative* permeability and permittivity tensors (in the element coordinate system if any). To specify a diagonal *relative* permeability tensor use the MURX, MURY, and MURZ labels on the **MP** command. Use the **TB**,MUR and **TBDATA** commands to specify the terms of an anisotropic permeability matrix. To specify a diagonal *relative* permittivity tensor use the PERX, PERY, and PERZ labels on the **MP** command. Use the **TB**,DPER and **TBDATA** commands to specify the terms of an anisotropic permittivity matrix.

You can specify an optional diagonal resistivity tensor (inverse of the conductivity tensor) using the RSVX, RSVY, and RSVZ labels on the **MP** command. To specify the terms of an anisotropic electric current conductivity tensor or anisotropic magnetic current conductivity tensor use **TBDATA** and **TB**,CNDE or **TB**,CNDM, respectively.

For an isotropic lossy material, you can define the lossy characteristics of the material by either the electric loss tangent (**MP**,LSST) or magnetic loss tangent (**MP**,LSSM). To calculate a specific absorption rate (SAR), you must input a mass density using the DENS label on the **MP** command.

To define nodal constraints on geometric nodes, use the **D** command. With the **D** command, the *Lab* variable corresponds to the only degree of freedom AX and the *VALUE* corresponds to the AX value. AX is not the x component in the global Cartesian coordinate system. In most cases, the AX value is zero, which corresponds to a perfect electric conductor (PEC) or "Electric Wall" (tangential component of $\bar{E} = 0$) condition. If both end nodes and the mid-node on an element edge are constrained, DOFs on the edge are also constrained. Similarly, if all edges on an element face are constrained, DOFs on the face are also constrained. The DOFs based on volume are not constrained. If you leave the nodes on a surface unspecified, the boundary assumes a perfect magnetic conductor (PMC) or "Magnetic Wall" condition (tangential component of $\bar{H} = 0$).

To define constraints on lines and areas, use the **DL** and **DA** commands, respectively. The *Lab* variable corresponds to the degree of freedom AX and the *Value1* corresponds to the AX value.

Section 2.8: Node and Element Loads describes element loads. You can specify an exterior waveguide port, surface impedance boundary conditions, infinite boundary surface flags, and Maxwell surface flags on the element faces

indicated by the circled numbers in *Figure 1, "HF119 Geometry"* using the **SF** and **SFE** commands or on the solid model using the **SFA** command. You can use the infinite boundary surface flag for a radiating open boundary in lieu of PML elements. You should use the Maxwell surface flag to determine an equivalent source surface for near and far field calculations performed in POST1.

You can apply current density, magnetic field, and electric field body loads to the finite element model using the **BF** and **BFE** commands or to the solid model using the **BFK**. **BFL**, **BFA**, and **BFV** commands. To specify a interior waveguide port, use the **BF** and **BFA** commands.

You can input the temperature (for material property evaluation only) body loads based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** and **TUNIF** commands. Element heat loss (JHEAT) represents the time-average Joule heat generation rate (W/m^3), and may be made available for a subsequent thermal analysis with companion elements (See the discussion of the **LDREAD** command).

Upon initiation of the solution calculations (**SOLVE**), the solid model loads and boundary conditions transfer automatically to the finite element model.

HF119 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
AX

Real Constants
None

Material Properties

MUZERO, MURX, MURY, MURZ, EPZRO, PERX, PERY, PERZ,
RSVX, RSVY, RSVZ, MUR, DPER, CNDE, CNDM,
LSST, LSSM, DENS

Surface Loads

Waveguide Port Surface Loads --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Impedance Surface Loads --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Infinite Boundary Surface Flags --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Maxwell Surface Flags for Equivalent Source Surface --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperature --
T(I), T(J), ..., T(R)

Current Density, Magnetic Field, Electric Field, and Waveguide Port --
JS, H, EF, PORT

Special Features
None

KEYOPT(1)
Used for element polynomial order selection:

0, 1 --
First order element

2 --
Second order element

KEYOPT(4)
Used for element type selection:

0 --
Normal element

1 --
Perfectly matched layers (PML) element

2 --
Scattering region element behind a soft source magnetic field excitation

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Centroid point printout

2 --
Nodal field printout

HF119 Solution Considerations

To choose the modal or full harmonic analysis type, issue the **ANTYPE** command.

In a harmonic analysis, the ANSYS program performs a complex solution and computes two sets of results data: real and imaginary. The measurable field quantities can be computed as the real step with a cosine time change and the imaginary set with a sine time change. You can set the frequency of the time change via the **HARFRQ** command. The measurable power terms and Joule losses are computed as rms (time-average) values and are stored with the real data set. You can choose a solver via the **EQSLV** command (the ICCG or sparse solvers are recommended).

For modal analysis, choose a frequency shift point just below the anticipated eigenfrequency of interest (via the **MODOPT** command). Select an upper end frequency as well. Use the *Method* argument to choose the Block Lanczos solver (the default). To visualize the electric and magnetic field modes, use the **MXPAND** command to expand the mode shapes.

HF119 Input Summary summarizes the element input. *Section 2.1: Element Input* of the *Elements Reference* gives a general description of element input.

HF119 Output Data

The solution output associated with this element is in two forms:

- Degrees of freedom (AX) included in the overall nodal solution
- Additional element output as shown in *Table 1, "HF119 Element Output Definitions"*

The element output direction is parallel to the element coordinate system (if any). *Section 2.2: Solution Output* in the *Elements Reference* provides a general description of solution output.See the *Basic Analysis Guide* for ways to view results.

Table 1, "HF119 Element Output Definitions" uses the following notation:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 HF119 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, ..., B	Y	Y
MAT	Material number	Y	Y
VOLU	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	Input temperatures T(I), T(J), ..., T(R)	Y	Y
LOC	Output location	*1*	-
MURX, MURY, MURZ	Relative permeability	*1*	-
PERX, PERY, PERZ	Relative permittivity	*1*	-
CNDX, CNDY, CNDZ	Conductivity	*1*	-
EF:X, Y, Z	Electric field intensity E	*1*	Y
EF:SUM	Magnitude of E	*1*	-
H:X, Y, Z	Magnetic field intensity H	*1*	Y
H:SUM	Magnitude of H	*1*	-
JC:X, Y, Z	Current density JC	*1*	Y
JC:SUM	Magnitude of JC	*1*	-
JHEAT	Joule heat generation per unit volume (time-average value)	-	-
PX, PY, PZ	Pointing vector (time-average value)	-	Y
PSCT	Reflected or transmitted power (time-average value)	*1*	-
PINC	Input power (time-average value)	-	-
VLOSS	Volumetric Joule losses (time-average value)	-	-
SFLOSS	Surface Joule losses (time-average value)	-	-
ENERGY	Stored energy (time-average value)	-	-
FACE1	1st element face number containing heat flux	-	*2*
HFLXAVG1	Heat flux across FACE1 caused by surface losses	-	*2*
FACE2	2nd element face number containing heat flux	-	*2*
HFLXAVG2	Heat flux across FACE2 caused by surface losses	-	*2*
FACE3	3rd element face number containing heat flux	-	*2*
HFLXAVG3	Heat flux across FACE3 caused by surface loses	-	*2*

Name	Definition	O	R
ETINCR	Real part of tangential incident electric field	-	*2*
ETINCI	Imaginary part of tangential incident electric field	-	*2*
ETOUTR	Real part of tangential outgoing electric field	-	*2*
ETOUTI	Imaginary part of tangential outgoing electric field	-	*2*
ETDOT	Dot product of waveguide eigen tangential electric field	-	*2*
SAR	Specific absorption rate	-	*2*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. The solution value is output only if calculated.
3. Available only at centroid as a ***GET** item.

Table 2 HF119 Miscellaneous Element Output

Description	Names of Items Output	O	R
Centroid Point Solution	LOC, MUX, MUY, MUZ, PERX, PERY, PERZ, E, ESUM, H, HSUM	*1*	-
Nodal Electric and Magnetic Field Solutions	E, ESUM, H, HSUM	*2*	-

1. Output at each centroid point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "HF119 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "HF119 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "HF119 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 HF119 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
PX	NMISC	1
PY	NMISC	2
PZ	NMISC	3
PSCT	NMISC	4
PINC	NMISC	5
ENERGY	NMISC	6
VLOSS	NMISC	7

Output Quantity Name	ETABLE Command Input	
	Item	E
SFLOSS	NMISC	8
FACE1	NMISC	9
HFLXAVG1	NMISC	10
FACE2	NMISC	11
HFLXAVG2	NMISC	12
FACE3	NMISC	13
HFLXAVG3	NMISC	14
ETINCR	NMISC	15
ETINCI	NMISC	16
ETOUTR	NMISC	17
ETOUTI	NMISC	18
ETDOT	NMISC	19
SAR	NMISC	20

HF119 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may be numbered either as shown in *Figure 1, "HF119 Geometry"* or may have the plane IJKL and MNOP interchanged.
- The required material properties (MURX, MURY, MURZ, PERX, PERY, PERZ) must be input as relative values.
- You cannot use the element in a transient analysis.
- Midside nodes must not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

HF119 Product Restrictions

There are no product-specific restrictions for this element.

HF120

3-D High-Frequency Brick Solid

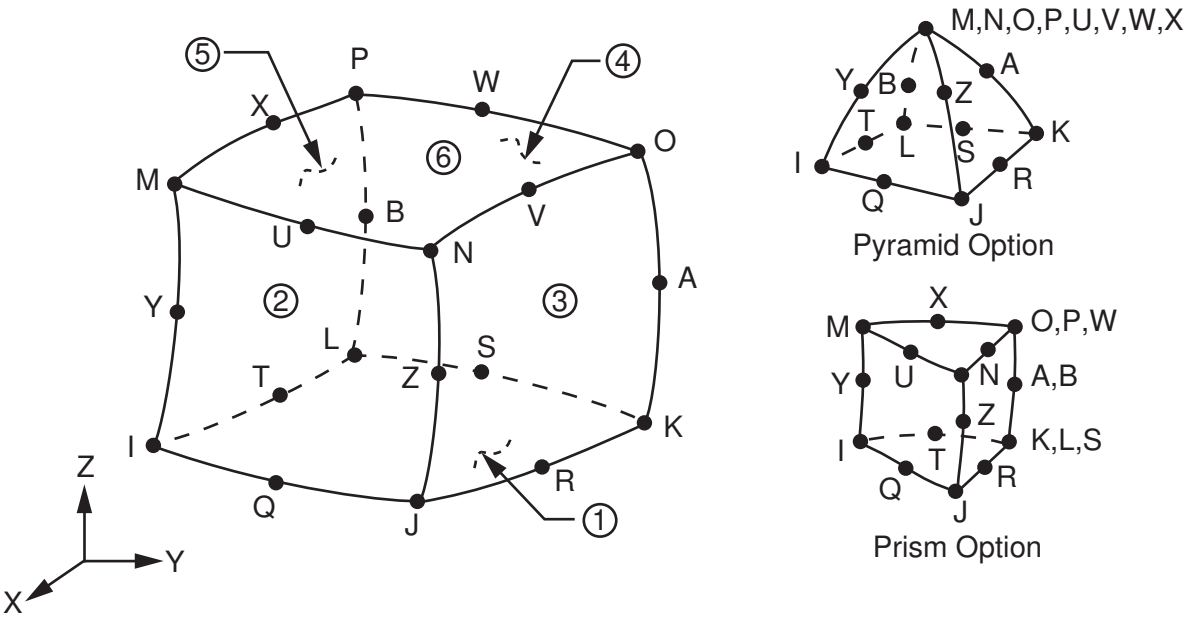
MP <> <> <> <> <> <> <> <> EH <> PP <>
Product Restrictions

HF120 Element Description

HF120 is a high-frequency brick element which models 3-D electromagnetic fields and waves governed by the full set of Maxwell's equations in linear media. It is based on a full-wave formulation of Maxwell's equations in terms of the time-harmonic electric field E (exponent jωt dependence assumed). A companion tetrahedral element, HF119, has similar full-wave capability. See HF120 in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on Maxwell's equations and full-wave formulations, respectively.

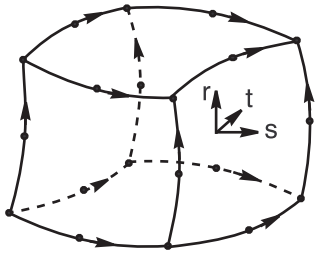
HF120 applies to the full-harmonic and modal analysis types, but not to the transient analysis type. It is defined by up to 20 geometric nodes with AX DOF on element edges and faces and inside the volume. The physical meaning of the AX DOF in this element is a projection of the electric field E on edges and faces, as well as normal components to the element faces.

Figure 1 HF120 Geometry



A first order or second order element option is available for the hexahedral and prism-shaped elements using KEYOPT(1). The pyramid-shaped element is only available as a first order element. The first order element has one AX DOF on each edge. The first order hexahedral element has a total of 12 AX DOFs.

Figure 2 HF120 First Order Hexahedral Element



The second order hexahedral element has two AX DOFs on each edge, four AX DOFs on each face, and six AX DOFs inside the volume for a total of 54 DOFs {2(12 edges) + 4(6 faces) + 6 (1 volume)}.

The first order prism and pyramid elements have a total of 9 AX and 8 AX DOFs, respectively. The second order prism element has a total of 42 DOFs {2(9 edges) + 4(5 faces) + 4(1 volume)}.

HF120 Input Data

Figure 1, "HF120 Geometry" shows the geometries, node locations, and the coordinate system for the element. The element supports three geometric shapes: a hexahedral (brick) shape defined by twenty geometric nodes with degeneracies to prism and pyramid shapes.

The only unit system supported for high-frequency analysis is the MKS unit, where the free-space permeability MUZERO = 4π x 10^{-7} H/m and the free-space permittivity PER0 = 8.854 x 10^{-12} F/m (see the **EMUNIT** command).

KEYOPT(4) provides options for the element formulation. KEYOPT(4) = 0 activates the normal full-wave formulation, which solves for the total field. KEYOPT(4) = 1 activates the perfectly matched layers (PML) formulation, which absorbs the field at the open boundary or at the port of a waveguide. KEYOPT(4) = 2 activates the scattering formulation, which is only required in the regions of a domain receiving a reflected wave from an imposed soft source magnetic field excitation (**BF**,,H option).

HF120 requires two sets of material constants; that is, *relative* permeability and permittivity tensors (in the element coordinate system if any). To specify a diagonal *relative* permeability tensor use the MURX, MURY, and MURZ labels on the **MP** command. Use the **TB**,MUR and **TBDATA** commands to specify the terms of an anisotropic permeability matrix. To specify a diagonal *relative* permittivity tensor use the PERX, PERY, and PERZ labels on the **MP** command. Use the **TB**,DPER and **TBDATA** commands to specify the terms of an anisotropic permittivity matrix.

You can specify an optional diagonal resistivity tensor (inverse of the conductivity tensor) using the RSVX, RSVY, and RSVZ labels on the **MP** command. To specify the terms of an anisotropic electric current conductivity tensor or anisotropic magnetic current conductivity tensor use **TBDATA** and **TB**,CNDE or **TB**,CNDM, respectively.

For an isotropic lossy material, you can define the lossy characteristics of the material by either the electric loss tangent (**MP**,LSST) or magnetic loss tangent (**MP**,LSSM). To calculate a specific absorption rate (SAR), you must input a mass density using the DENS label on the **MP** command.

To define nodal constraints on geometric nodes, use the **D** command. With the **D** command, the *Lab* variable corresponds to the only degree of freedom AX and the *VALUE* corresponds to the AX value. AX is not the x component in the global Cartesian coordinate system. In most cases, the AX value is zero, which corresponds to a perfect electric conductor (PEC) or "Electric Wall" (tangential component of $\bar{E} = 0$) condition. If both end nodes and the mid-node on an element edge are constrained, DOFs on the edge are also constrained. Similarly, if all edges on an element face are constrained, DOFs on the face are also constrained. The DOFs based on volume

are not constrained. If you leave the nodes on a surface unspecified, the boundary assumes a perfect magnetic conductor (PMC) or "Magnetic Wall" condition (tangential component of $H^{-} = 0$).

To define constraints on lines and areas, use the **DL** and **DA** commands, respectively. The *Lab* variable corresponds to the degree of freedom AX and the *Value1* corresponds to the AX value.

Section 2.8: Node and Element Loads describes element loads. You can specify an exterior waveguide port, surface impedance boundary conditions, infinite boundary surface flags, and Maxwell surface flags on the element faces indicated by the circled numbers in *Figure 1, "HF120 Geometry"* using the **SF** and **SFE** commands or on the solid model using the **SFA** command. You can use the infinite boundary surface flag for a radiating open boundary in lieu of PML elements. You should use the Maxwell surface flag to determine an equivalent source surface for near and far field calculations performed in POST1.

To define surface loads on areas of the model, use the **SFA** command.

You can apply current density, magnetic field, and electric field body loads to the finite element model using the **BF** and **BFE** commands or to the solid model using the **BFK**. **BFL**, **BFA**, and **BFV** commands. To specify a interior waveguide port, use the **BF** and **BFA** commands.

You can input the temperature (for material property evaluation only) body loads based on their value at the element's nodes or as a single element value (**BF** and **BFE** commands). In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** and **TUNIF** commands. Element heat loss (JHEAT) may be made available for a subsequent thermal analysis with companion elements. (See the description of the **LDREAD** command.)

Upon initiation of the solution calculations (**SOLVE**), the solid model loads and boundary conditions transfer automatically to the finite element model.

HF120 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

AX

Real Constants

None

Material Properties

MUZERO, MURX, MURY, MURZ, EPZRO, PERX, PERY, PERZ,
RSVX, RSVY, RSVZ, MUR, DPER, CNDE, CNDM,
LSST, LSSM, DENS

Surface Loads

Waveguide Port Surface Loads --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Impedance Surface Loads --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Infinite Boundary Surface Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Maxwell Surface Flags for Equivalent Source Surface --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), ..., T(B)

Current Density, Magnetic Field, Electric Field, and Waveguide Port --
JS, H, EF, PORT

Special Features
None

KEYOPT(1)
Element polynomial order selection:

0, 1 --
First order element

2 --
Second order element

Note

This option is only available for the hexahedral and prism-shaped elements. The pyramid-shaped element is only available as a first order element.

KEYOPT(4)
Element description options:

0 --
Normal element

1 --
Perfectly matched layers (PML) element

2 --
Scattering region element behind a soft source magnetic field excitation

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Centroid point printout

2 --
Nodal field printout

HF120 Solution Considerations

To choose the modal or full harmonic analysis type, issue the **ANTYPE** command.

In a harmonic analysis, the ANSYS program performs a complex solution and computes two sets of results data: real and imaginary. The measurable field quantities can be computed as the real step with a cosine time change and the imaginary set with a sine time change. You can set the frequency of the time change via the **HARFRQ** command. The measurable power terms and Joule losses are computed as rms (time-average) values and are stored with the real data set. You can choose a solver via the **EQSLV** command (the ICCG or sparse solvers are recommended).

For modal analysis, choose a frequency shift point just below the anticipated eigenfrequency of interest (via the **MODOPT** command). Select an upper end frequency as well. Use the *Method* argument to choose the Block Lanczos solver (the default). To visualize the electric and magnetic field modes, use the **MXPAND** command to expand the mode shapes.

HF120 Input Summary summarizes the element input. *Section 2.1: Element Input* in the *Elements Reference* provides a general description of element input.

HF120 Output Data

The solution output associated with this element is in two forms:

- Degrees of freedom (AX) included in the overall nodal solution
- Additional element output as shown in *Table 1, "HF120 Element Output Definitions"*

The element output direction is parallel to the element coordinate system (if any). *Section 2.2: Solution Output* in the *Elements Reference* provides a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Table 1, "HF120 Element Output Definitions" uses the following notation:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 HF120 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, ..., B	Y	Y
MAT	Material number	Y	Y
VOLU	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	Input temperatures T(I), T(J), ..., T(B)	Y	Y
LOC	Output location	*1*	-
MURX, MURY, MURZ	Relative permeability	*1*	-
PERX, PERY, PERZ	Relative permittivity	*1*	-
CNDX, CNDY, CNDZ	Conductivity	*1*	-
EF:X, Y, Z	Electric field intensity E	*1*	Y

Name	Definition	O	R
EF:SUM	Magnitude of E	*1*	-
H:X, Y, Z	Magnetic field intensity H	*1*	Y
H:SUM	Magnitude of H	*1*	-
JC:X, Y, Z	Current density JC	*1*	Y
JC:SUM	Magnitude of JC	*1*	-
JHEAT	Joule heat generation per unit volume (time-average value)	-	-
PX, PY, PZ	Pointing vector (time-average value)	-	Y
PSCT	Reflected or transmitted power (time-average value)	*1*	-
PINC	Input power (time-average value)	-	-
VLOSS	Volumetric Joule losses (time-average value)	-	-
SFLOSS	Surface Joule losses (time-average value)	-	-
ENERGY	Stored energy (time-average value)	-	-
FACE1	1st element face number containing heat flux	-	*2*
HFLXAVG1	Heat flux across FACE1 caused by surface losses	-	*2*
FACE2	2nd element face number containing heat flux	-	*2*
HFLXAVG2	Heat flux across FACE2 caused by surface losses	-	*2*
FACE3	3rd element face number containing heat flux	-	*2*
HFLXAVG3	Heat flux across FACE3 caused by surface losses	-	*2*
ETINCR	Real part of tangential incident electric field	-	*2*
ETINCI	Imaginary part of tangential incident electric field	-	*2*
ETOUTR	Real part of tangential outgoing electric field	-	*2*
ETOUTI	Imaginary part of tangential outgoing electric field	-	*2*
ETDOT	Dot product of waveguide eigen tangential electric field	-	*2*
SAR	Specific absorption rate	-	*2*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. The solution is output only if calculated.
3. Available only at centroid as a ***GET** item.

Table 2 HF120 Miscellaneous Element Output

Description	Names of Items Output	O	R
Centroid Point Solution	LOC, MUX, MUY, MUZ, PERX, PERY, PERZ, E, ESUM, H, HSUM	*1*	-
Nodal Electric and Magnetic Field Solutions	E, ESUM, H, HSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "HF120 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "HF120 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "HF120 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 HF120 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	I
PX	NMISC	1
PY	NMISC	2
PZ	NMISC	3
PSCT	NMISC	4
PINC	NMISC	5
ENERGY	NMISC	6
VLOSS	NMISC	7
SFLOSS	NMISC	8
FACE1	NMISC	9
HFLXAVG1	NMISC	10
FACE2	NMISC	11
HFLXAVG2	NMISC	12
FACE3	NMISC	13
HFLXAVG3	NMISC	14
ETINCR	NMISC	15
ETINCI	NMISC	16
ETOUTR	NMISC	17
ETOUTI	NMISC	18
ETDOT	NMISC	19
SAR	NMISC	20

HF120 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may be numbered either as shown in *Figure 1, "HF120 Geometry"* or may have the plane IJKL and MNOP interchanged.
- The required material properties (MURX, MURY, MURZ, PERX, PERY, PERZ) must be input as relative values.
- You cannot use the element in a transient analysis.
- Midside nodes must not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.

HF120 Product Restrictions

There are no product-specific restrictions for this element.

PLANE121

2-D 8-Node Electrostatic Solid

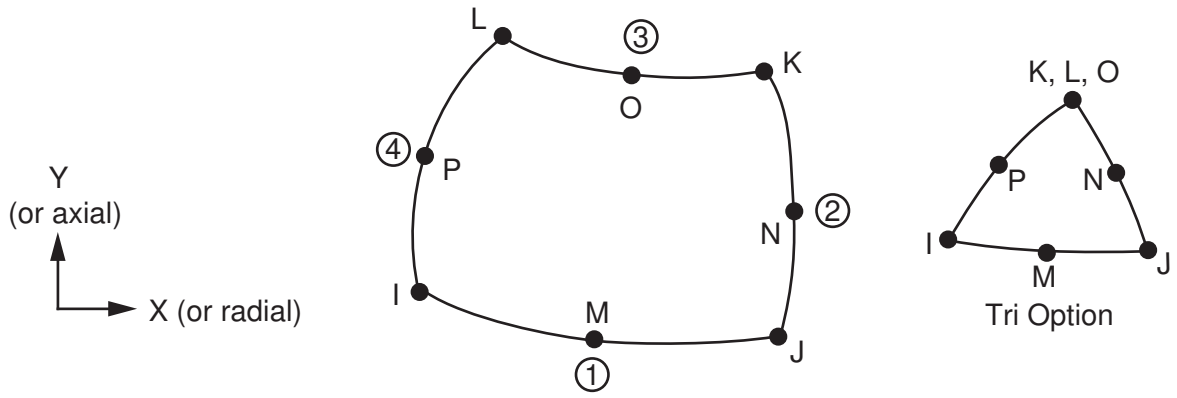
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

PLANE121 Element Description

PLANE121 is a 2-D, 8-node, charge-based electric element. The element has one degree of freedom, voltage, at each node. The 8-node elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation, and it is applicable to 2-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See PLANE121 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE121 Geometry



PLANE121 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE121 Geometry"*. The element is defined by eight nodes and orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Nodal loads are defined with the **D** (*Lab*= VOLT) and **F** (*Lab*= CHRG) commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers in *Figure 1, "PLANE121 Geometry"*. Charge densities may be input as element body loads at the nodes. If the node I charge density CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *PLANE121 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE121 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
PERX, PERY, LSST, RSVX, RSVY

Surface Loads

Surface charge densities --
CHRGS face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Volume charge densities --
CHRGD(I), CHRGD(J), CHRGD(K), CHRGD(L), CHRGD(M), CHRGD(N), CHRGD(O), CHRGD(P)

Special Features
Birth and death

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

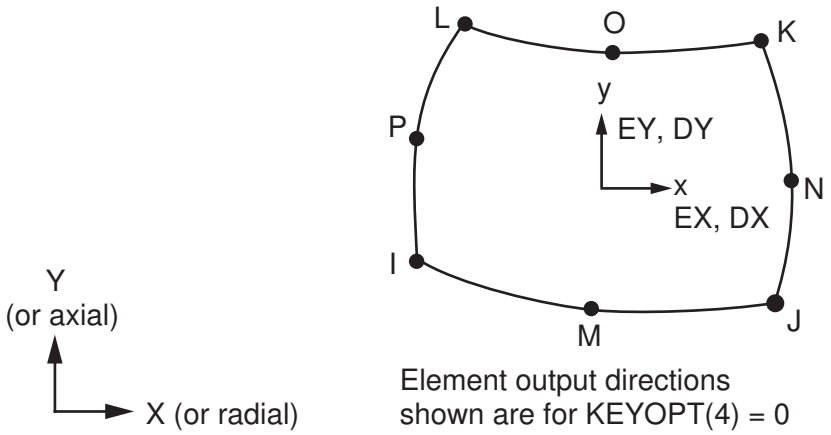
PLANE121 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE121 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE121 Output"*. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE121 Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE121 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y

Name	Definition	O	R
LOC	Output location (X, Y)	*1*	*1*
PERX, PERY	Electric relative permittivity	*1*	*1*
EF:X, Y	Electric field components	*1*	*1*
EF:SUM	Vector magnitude of EF	*1*	*1*
D:X, Y	Electric flux density components	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
JS:X, Y, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*5*] [*6*]	*1*	*1*
SENE:	Stored electric energy [*6*]	*1*	*1*
FMAG:X, Y	Electrostatic force [*7*]	-	*1*
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT.
4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.
7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 2 PLANE121 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	*1*	-
Nodal Solution	EF, EFSUM, D, DSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each node, if KEYOPT(5) = 2

Note

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively.

Table 3, "PLANE121 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "PLANE121 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE121 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 PLANE121 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1
PERX	NMISC	1
PERY	NMISC	2
JTX	NMISC	4
JTY	NMISC	5
JTSUM	NMISC	6

PLANE121 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 2, "PLANE121 Output"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

PLANE121 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

SOLID122

3-D 20-Node Electrostatic Solid

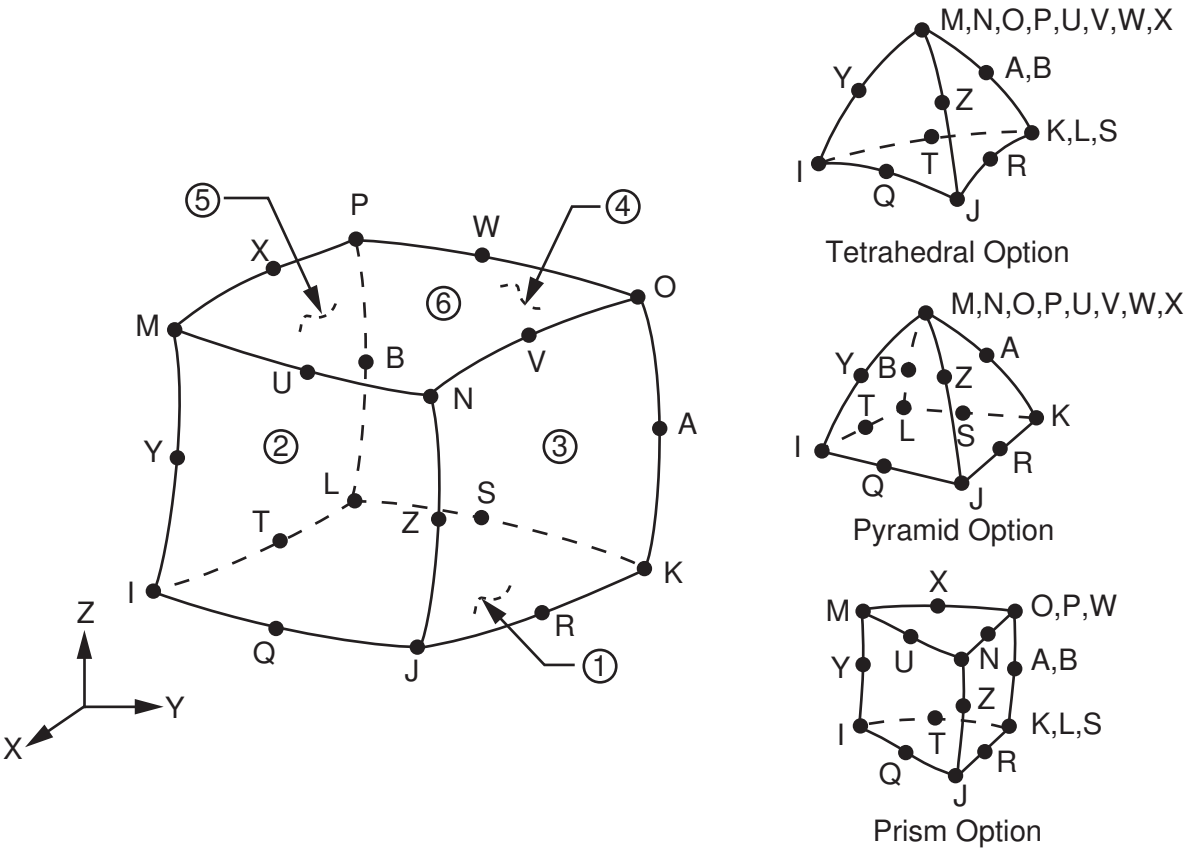
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID122 Element Description

SOLID122 is a 3-D, 20-node, charge-based electric element. The element has one degree of freedom, voltage, at each node. It can tolerate irregular shapes without much loss of accuracy. SOLID122 elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is applicable to 3-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See SOLID122 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID122 Geometry



SOLID122 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID122 Geometry"*. The element is defined by 20 node points and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A pyramid-shaped element and a tetrahedral-shaped element may also be formed as shown in *Figure 1, "SOLID122 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*. Nodal loads are defined with the **D** (*Lab*= VOLT) and **F** (*Lab*= CHRG) commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID122 Geometry"*. Charge density may be input as element body loads at the nodes. If the node I charge densities CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *SOLID122 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID122 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ

Surface Loads

Surface charge densities --

CHRGS face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Volume charge densities --

CHRGD(I), CHRGD(J), ..., CHRGD(Z), CHRGD(A), CHRGD(B)

Special Features
Birth and death

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

SOLID122 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID122 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID122 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
LOC	Output location (X, Y, Z)	*1*	*1*
PERX, PERY, PERZ	Electric relative permittivity	*1*	*1*
EF:X, Y, Z	Electric field components	*1*	*1*
EF:SUM	Vector magnitude of EF	*1*	*1*
D:X, Y, Z	Electric flux density components	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, Z, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*6*] [*5*]	*1*	*1*
SENE:	Stored electric energy [*5*]	*1*	*1*

Name	Definition	O	R
FMAG:X, Y, Z	Electrostatic force [*7*]	-	*1*
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD=JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].
4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.
7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 2 SOLID122 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	*1*	-
Nodal Solution	EF, EFSUM, D, DSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "SOLID122 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID122 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SOLID122 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 SOLID122 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
EFX	SMISC	2
EFY	SMISC	3
EFZ	SMISC	4
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID122 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID122 Geometry"* or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

SOLID122 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

SOLID123

3-D 10-Node Tetrahedral Electrostatic Solid

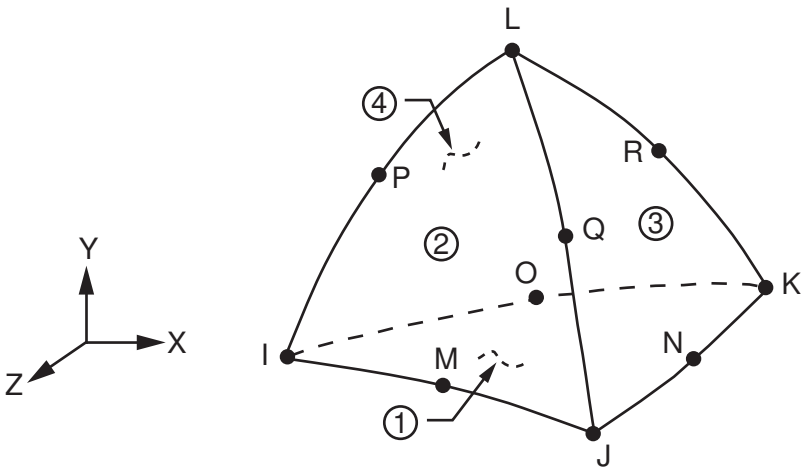
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID123 Element Description

SOLID123 is a 3-D, 10-node, charge-based electric element. It is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, voltage, at each node.

This element is applicable to 3-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See SOLID123 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID123 Geometry



SOLID123 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID123 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*. Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = CHRG) commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID123 Geometry"*. Charge densities may be input as element body loads at the nodes. If the node I charge density CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *SOLID123 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID123 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ

Surface Loads

Surface charge densities --
CHRGS face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Volume charge densities --

CHRGD(I), CHRGD(J), CHRGD(K), CHRGD(L), CHRGD(M),
CHRGD(N), CHRGD(O), CHRGD(P), CHRGD(Q), CHRGD(R)

Special Features
Birth and death

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

SOLID123 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID123 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* in the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID123 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
LOC	Output location (X, Y, Z)	*1*	*1*
PERX, PERY, PERZ	Electric relative permittivity	*1*	*1*
EF:X, Y, Z	Electric field components (X, Y, Z)	*1*	*1*
EF:SUM	Vector magnitude of EF	*1*	*1*
D:X, Y, Z	Electric flux density components	*1*	*1*
D:SUM	Vector magnitude of D	*1*	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, Z, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*6*] [*5*]	*1*	*1*
SENE:	Stored electric energy [*5*]	*1*	*1*
FMAG:X, Y, Z	Electrostatic force [*7*]	-	*1*
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current

density (JD) can be derived from JS and JT as JD=JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.
7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 2 SOLID123 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	*1*	-
Nodal Solution	EF, EFSUM, D, DSUM	*2*	-

1. Output at each integration point, if KEYOPT(5) = 1
2. Output at each corner node, if KEYOPT(5) = 2

Table 3, "SOLID123 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SOLID123 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID123 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 SOLID123 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1
EFX	SMISC	2
EFY	SMISC	3
EFZ	SMISC	4
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JTSUM	NMISC	8

SOLID123 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID123 Geometry"* or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

SOLID123 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag 3-D

- The birth and death special feature is not allowed.

CIRCU124

Electric Circuit

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

CIRCU124 Element Description

CIRCU124 is a general circuit element applicable to circuit simulation. The element may also interface with electromagnetic finite elements to simulate coupled electromagnetic-circuit field interaction. The element has up to 6 nodes to define the circuit component and up to three degrees of freedom per node to model the circuit response. For electromagnetic-circuit field coupling, the element may interface with PLANE53 and SOLID97, the 2-D and 3-D electromagnetic field elements. CIRCU124 is applicable to static, harmonic, and transient analyses.

CIRCU124 Input Data

The geometry, node definition, and degree of freedom options are shown in *Figure 1, "CIRCU124 Circuit Element Options"* (circuit components), *Figure 2, "CIRCU124 Circuit Source Options"* (circuit source options), and *Figure 3, "CIRCU124 Coupled Circuit Source Options"* (coupled circuit source options). The element is defined by *active* and *passive* circuit nodes. *Active* nodes are those connected to an overall electric circuit, and *passive* nodes are those used internally by the element and not connected to the circuit. For the coupled circuit source options, the passive nodes are actual nodes of a source conductor modeled in the electromagnetic field domain.

Element circuit components, sources, and coupled sources are defined by KEYOPT(1) settings and its corresponding real constants. Real constant input is dependent on the element circuit option used. A summary of the element input options is given in *CIRCU124 Input Summary*. Real constants numbers 15 and 16 are created by the GUI Circuit Builder (see the *Modeling and Meshing Guide*), and are not required input for analysis purposes. The element is characterized by up to three degrees of freedom:

- VOLT (voltage)
- CURR (current)
- EMF (potential drop)

Figure 1 CIRCU124 Circuit Element Options

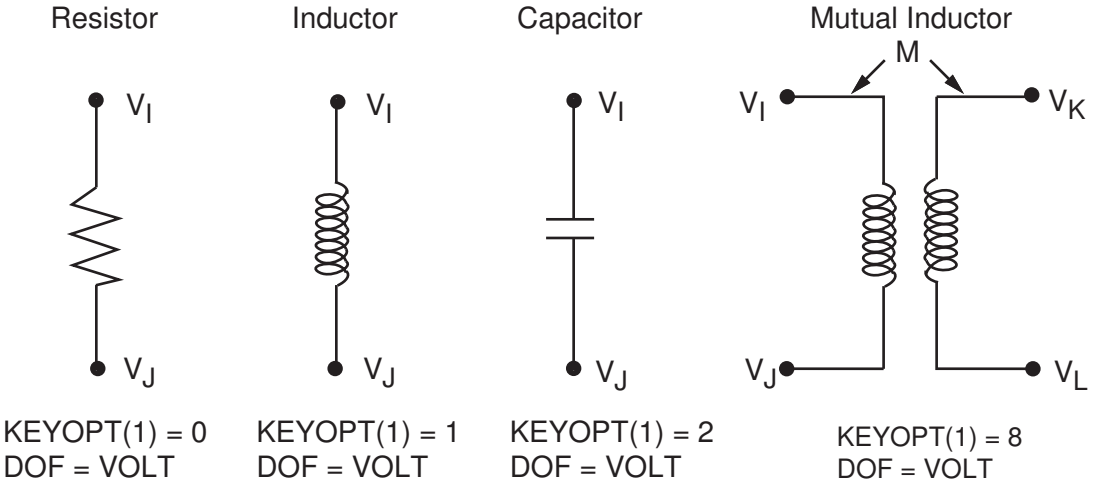
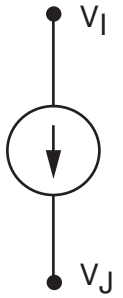


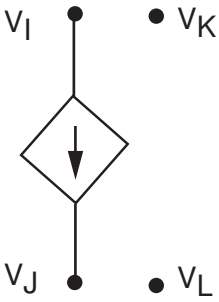
Figure 2 CIRCU124 Circuit Source Options

Independent Current Source



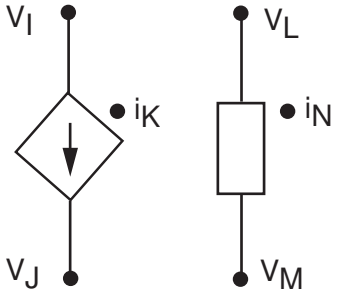
KEYOPT(1) = 3
DOF = VOLT

Voltage-controlled Current Source



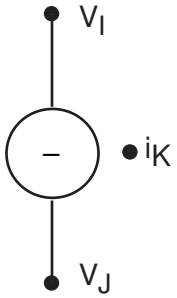
KEYOPT(1) = 9
DOF = VOLT

Current-controlled Current Source



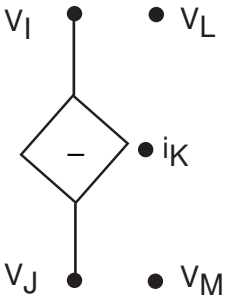
KEYOPT(1) = 12
DOF = VOLT (I,J,L,M),
CURR (K,N)

Independent Voltage Source



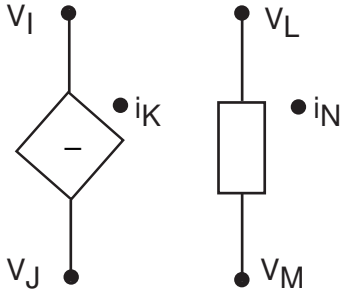
KEYOPT(1) = 4
DOF = VOLT (I,J),
CURR (K)

Voltage-controlled Voltage Source



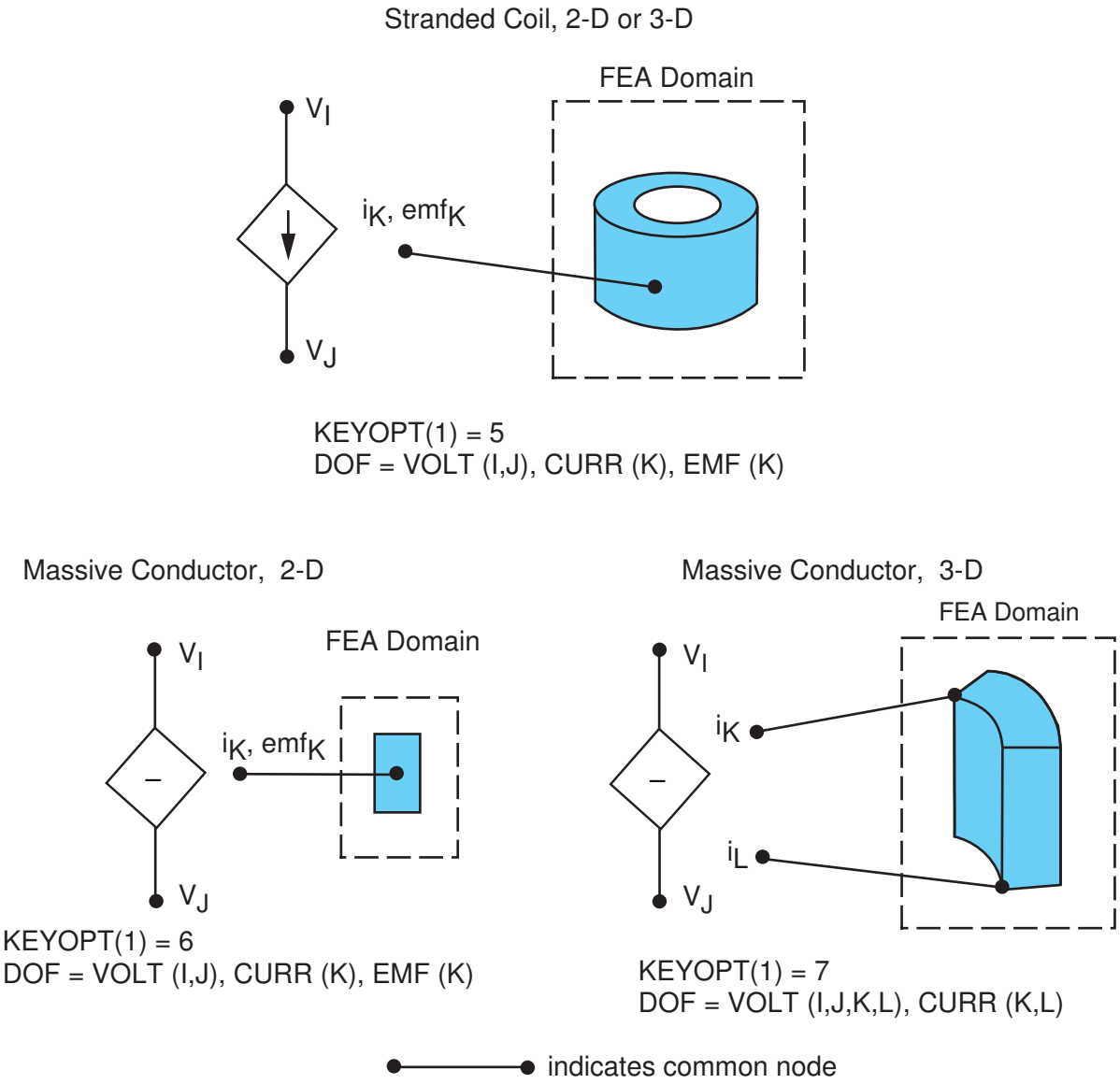
KEYOPT(1) = 10
DOF = VOLT (I,J,L,M)
CURR (K)

Current-controlled Voltage Source



KEYOPT(1) = 11
DOF = VOLT (I,J,L,M),
CURR (K,N)

Figure 3 CIRCU124 Coupled Circuit Source Options



Independent voltage and current sources (KEYOPT(1) = 3 or 4) may be excited by AC/DC, sinusoidal, pulse, exponential, or piecewise linear load functions as defined by KEYOPT(2); see *Figure 4, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.

The time-step size for a transient analysis is controlled by the **DELTIM** or **NSUBST** commands. The CIRCU124 element does not respond to automatic time stepping (**AUTOTS** command), but **AUTOTS** can be used as a mechanism for ramping the time step to its final value. For coupled electromagnetic-circuit problems, automatic time stepping may be used if controls are placed on degrees of freedom other than VOLT, CURR, or EMF, or loads associated with those degrees of freedom.

For problems using the CIRCU124 element with the EMF degree of freedom, the frontal solver is chosen by default. For problems using the CIRCU124 element with only the VOLT and/or CURR degrees of freedom, the sparse direct solver is chosen by default.

CIRCU124 Input Summary

Nodes
I, J, K, L, M, N

Degrees of Freedom
VOLT, CURR, EMF (see *Figure 1, "CIRCU124 Circuit Element Options"*)

Real Constants
Dependent on KEYOPT(1) and KEYOPT(2) settings. See *Table 1, "CIRCU124 Real Constants"* for details.

Material Properties
None

Surface Loads
None

Body Loads
See KEYOPT(2)

Special Features
Magnetic Field Coupling

KEYOPT(1)
Circuit component type:

0 --
Resistor

1 --
Inductor

2 --
Capacitor

3 --
Independent Current Source

4 --
Independent Voltage Source

5 --
Stranded Coil Current Source

6 --
2-D Massive Conductor Voltage Source

7 --
3-D Massive Conductor Voltage Source

8 --
Mutual Inductor

9 --
Voltage-Controlled Current Source

10 --
Voltage-Controlled Voltage Source

11 --
Current-Controlled Voltage Source

12 --
Current-Controlled Current Source

KEYOPT(2)

Body loads available if KEYOPT(1) = 3 or 4:

0 --
DC or AC Harmonic load

1 --
Sinusoidal load

2 --
Pulse load

3 --
Exponential load

4 --
Piecewise Linear load

Table 1 CIRCU124 Real Constants

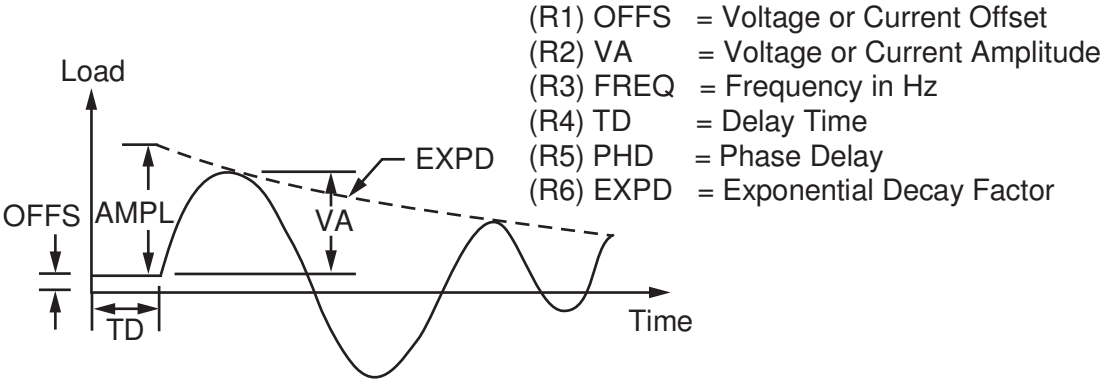
Circuit Option and Graphics Label	KEYOPT(1)	Real Constants
Resistor (R)	0	R1 = Resistance (RES)
Inductor (L)	1	R1 = Inductance (IND) R2 = Initial inductor current (ILO)
Capacitor (C)	2	R1 = Capacitance (CAP) R2 = Initial Capacitor Voltage (VCO)
Mutual Inductor (K)	8	R1 = Primary Inductance (IND1) R2 = Secondary Inductance (IND2) R3 = Coupling Coefficient (K)
Independent Current Source (I)	3	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0: see *Figure 4, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.
Voltage-Controlled Current Source (G)	9	R1 = Transconductance (GT)
Current-Controlled Current Source (F)	12	R1 = Current Gain (AI)
Independent Voltage Source (V)	4	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0: see *Figure 4, "Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources"*.
Voltage-Controlled Voltage Source (E)	10	R1 = Voltage Gain (AV)
Current-Controlled Voltage Source (H)	11	R1 = Transresistance (RT)
Stranded Coil Current Source (N)	5	R1 = Symmetry Factor (SCAL)

Circuit Option and Graphics Label	KEYOPT(1)	Real Constants
2-D Massive Conductor Voltage Source (M)	6	R1 = Symmetry Factor (SCAL)
3-D Massive Conductor Voltage Source (P)	7	R1 = Symmetry Factor (SCAL)

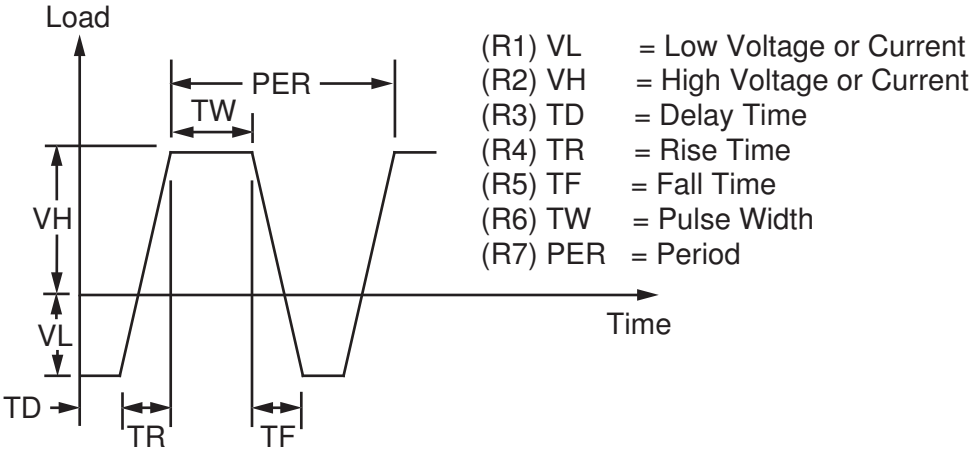
Note

For all above Circuit options, the GOFFST and ID real constants (numbers 15 and 16) are created by the Circuit Builder automatically:

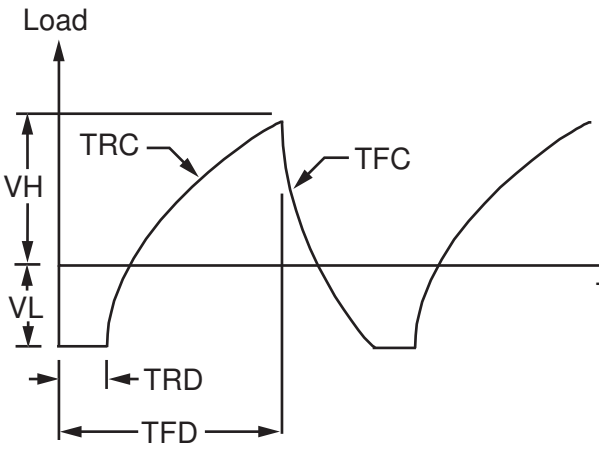
Figure 4 Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources



(a) Sinusoidal load, KEYOPT(2) = 1

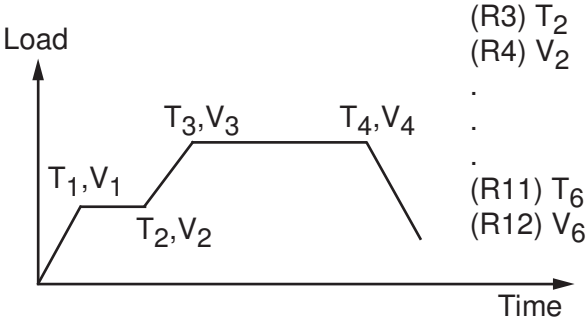


(b) Pulse load, KEYOPT(2) = 2



(R1) VL = Low Voltage or Current
(R2) VH = High Voltage or Current
(R3) TRD = Rise Delay Time
(R4) TRC = Rise Time-Constant
(R5) TFD = Fall Delay Time
(R6) TFC = Fall Time-Constant
(R7) PER = Period, defaults to TRD + 2 (TFD-TRD)

(c) Exponential load, KEYOPT(2) = 3



(R1) T_1 = Time-point 1
(R2) V_1 = Voltage or current at time-point 1
(R3) T_2 = Time-point 2
(R4) V_2 = Voltage or current at time-point 2
.
.
.
(R11) T_6 = Time-point 6
(R12) V_6 = Voltage or current at time-point 6

(d) Piecewise linear load, KEYOPT(2) = 4

CIRCU124 Output Data

The element output for this element is dependent on the circuit option selected. *Table 2, "CIRCU124 Element Output Definitions"* summarizes the element output data.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CIRCU124 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Resistor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
RES	Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y

Name	Definition	O	R
POWER	Power loss	Y	Y
For KEYOPT(1) = 1: Inductor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
IND	Inductance	Y	Y
IL0	Initial current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 2: Capacitor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
CAP	Capacitance	Y	Y
VC0	Initial voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 3: Independent Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
CURRENT SOURCE	Real or imaginary component of applied current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 4: Independent Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
VOLTAGE SOURCE	Real or imaginary component of applied voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 5: Stranded Coil Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 6: 2-D Massive Conductor Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y

Name	Definition	O	R
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 7: 3-D Massive Conductor Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K and L	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 8: 3-D Mutual Inductor (Transformer)			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
IND1	Primary inductance	Y	Y
IND2	Secondary inductance	Y	Y
INDM	Mutual inductance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current in I-J branch	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
CONTROL CURR	Current in K-L branch	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 9: Voltage Controlled Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
GT	Transconductance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current in I-J branch	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 10: Voltage Controlled Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M	Y	Y
AV	Voltage gain	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 11: Current Controlled Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
GT	Transresistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y

Name	Definition	O	R
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
CONTROL CURR	Current at node N	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 12: Current Controlled Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
AI	Current gain	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
CONTROL CURR	Current at node N	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y

Table 3, "CIRCU124 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CIRCU124 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CIRCU124 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 CIRCU124 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1
CURRENT	SMISC	2
CONTROL VOLT	SMISC	3
CONTROL CURR	SMISC	4
POWER	NMISC	1
SOURCE (real)	NMISC	2
SOURCE (imaginary)	NMISC	3

CIRCU124 Assumptions and Restrictions

- For static analyses, a capacitor circuit element is treated as an open-circuit and an inductor circuit element is treated as a short-circuit.
- Only MKS units are allowed (**EMUNIT** command).
- The resistor, inductor, capacitor, independent current source, and mutual inductor circuit options produce symmetric coefficient matrices while the remaining options produce unsymmetric matrices.

- The frontal (recommended) and sparse solvers are available for problems using the CIRCU124 element. Even if you choose a different solver, ANSYS uses the frontal or sparse solver (depending on the physics of your problem) when CIRCU124 elements are present.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

CIRCU124 Product Restrictions

There are no product-specific restrictions for this element.

CIRCU125

Diode

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

CIRCU125 Element Description

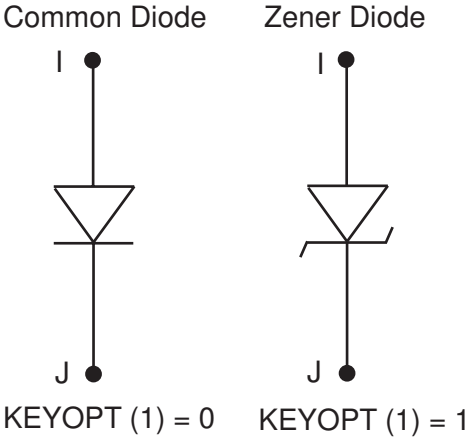
CIRCU125 is a diode element normally used in electric circuit analysis. The element may also interface with electromagnetic and mechanical finite elements to simulate fully coupled electromechanical analyses at the lumped parameter level. The element has 2 nodes to define the circuit component and one degree of freedom per node to model the circuit response. The element may interface with the electric circuit element CIRCU124, with the mechanical elements MASS21, COMBIN14, and COMBIN39, and with the electromechanical transducer element TRANS126. CIRCU125 is applicable to static analyses and transient analyses with restart.

CIRCU125 Input Data

The geometry, node definition, and degree of freedom options are shown in *Figure 1, "CIRCU125 Element Options"*.

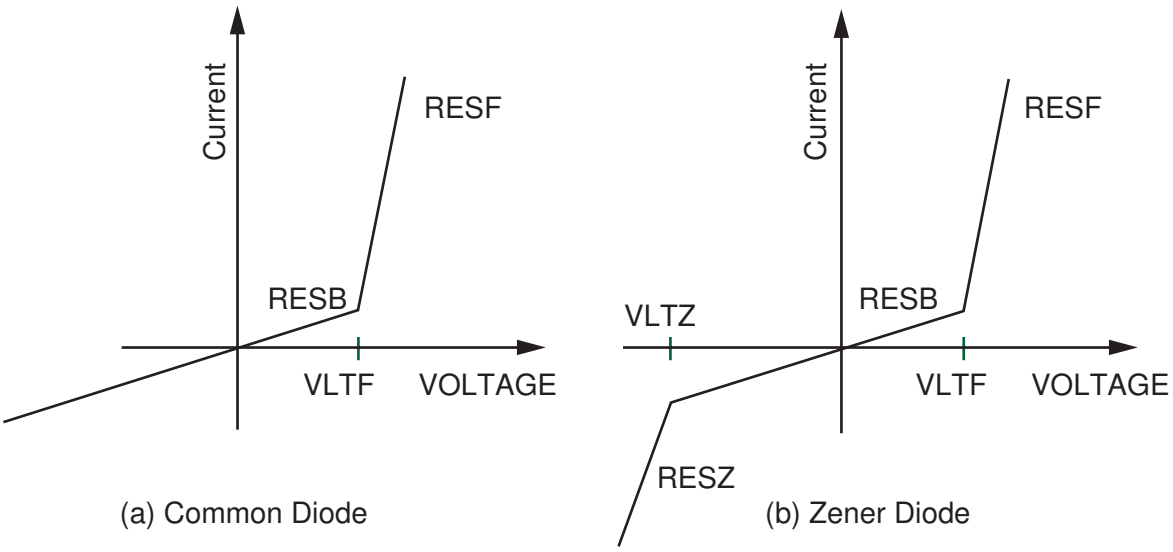
The diode element is defined by the KEYOPT(1) setting and its corresponding real constants. Real constant input is dependent on the diode option used. A summary of the element input options is given in *CIRCU125 Input Summary*. Real constants numbers 1 and 2 are created by the GUI Circuit Builder (see the *Modeling and Meshing Guide*), and are not required input for analysis purposes. The element is characterized by one degree of freedom, VOLT (voltage).

Figure 1 CIRCU125 Element Options



The I-U characteristics of the diodes are approximated by the piecewise linear functions shown in *Figure 2, "CIRCU125 I-U Characteristics"*. The characteristic of a common (non-Zener) diode consists of line segments corresponding to the closed and open states. The characteristic of a Zener diode consists of three segments corresponding to the Zener, closed, and open states. The diode characteristic can be ideal or lossy depending on the values of the real constants.

Figure 2 CIRCU125 I-U Characteristics



Legend
VLTF - Forward voltage
VLTZ - Zener voltage
RESF - Slope is forward resistance
RESB - Slope is blocking resistance
RESZ - Slope is Zener resistance

CIRCU125 Input Summary

Nodes
I, J

Degrees of Freedom
VOLT

Real Constants

Dependent on KEYOPT(1) settings.
For KEYOPT(1) = 0:
GOFFST, ID, (blank), RESF, VLTF, RESB,
(blank), (blank)
For KEYOPT(1) = 1:
GOFFST, ID, (blank), RESF, VLTF, RESB,
RESZ, VLTZ
See *Table 1, "CIRCU125 Real Constants"*.

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)

Select diode options:

0 --

Common Diode

1 --

Zener Diode

If you are using the Circuit Builder to construct your model, the real constants GOFFST and ID are provided automatically.

Note

The real constant default values may not be appropriate to analyze micro devices (i.e., devices with extremely small dimensions) in MKSA units.

Table 1 CIRCU125 Real Constants

Real Constant No.	Name	Description
Common Diode (D) (KEYOPT(1) = 0)		
1	GOFFST	Graphical offset
2	ID	Element identification number
3	--	(blank)
4	RESF	Forward resistance (if not entered, defaults to 1.0e-12 Ohm)
5	VLTF	Forward voltage (if not entered, defaults to 0.0e0 Volt)
6	RESB	Blocking resistance (if not entered, defaults to 1.0e+12 Ohm)
7, 8	--	(blank)
Zener Diode (Z) (KEYOPT(1) = 2) - use real constants 1 through 6 as above, then:		
7	RESZ	Zener resistance (if not entered, defaults to 1.0e+12 Ohm)
8	VLTZ	Zener voltage (if not entered, defaults to 1.0e-12 Volt)

CIRCU125 Solution Considerations

CIRCU125 is a highly nonlinear element. To obtain convergence, you may have to define convergence criteria, instead of using the default values. Use **CNVTOL**,VOLT,,0.001,2,1.0E-6 if you need to change the convergence criteria.

CIRCU125 Output Data

The element output for this element is dependent on the circuit option selected. *Table 2, "CIRCU125 Element Output Definitions"* summarizes the element output data.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CIRCU125 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Common Diode			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
REST	Tangent Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
STAT	Diode status	*1*	*1*
DYNRES	Dynamic resistance at operating point	Y	Y
AMPGEN	Norton equivalent current generator	Y	Y
For KEYOPT(1) = 1: Zener Diode			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
REST	Tangent resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
STAT	Diode status	*2*	*2*
DYNRES	Dynamic resistance at operating point	Y	Y
AMPGEN	Norton equivalent current generator	Y	Y

1. Common Diode Status Values

   1 - Forward, open

   2 - Reverse, blocked

2. Zener Diode Status Values

   1 - Forward, open

   2 - Reverse, blocked

   3 - Zener, breakdown

Table 3, "CIRCU125 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CIRCU125 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 2, "CIRCU125 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 CIRCU125 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1
CURRENT	SMISC	2
POWER	NMISC	1
blank	NMISC	2
blank	NMISC	3
DYNRES	NMISC	4
AMPGEN	NMISC	5
STAT	NMISC	6

CIRCU125 Assumptions and Restrictions

- If either the Zener voltage or Zener resistance is blank or very small, the Zener diode will be replaced with a common diode, and a warning will be issued.
- Only MKS units are allowed (**EMUNIT** command).
- If the Zener Voltage is entered as a positive number, the element will negate the value that is entered. If the Forward Voltage is entered as a negative number, the element will replace it with its absolute value. All resistance must be positive. Any negative resistance value is replaced by its absolute value.
- The element issues an error message if applied in harmonic analysis.
- This element does not work with the CIRCU94 piezoelectric element.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

CIRCU125 Product Restrictions

There are no product-specific restrictions for this element.

TRANS126

Electromechanical Transducer

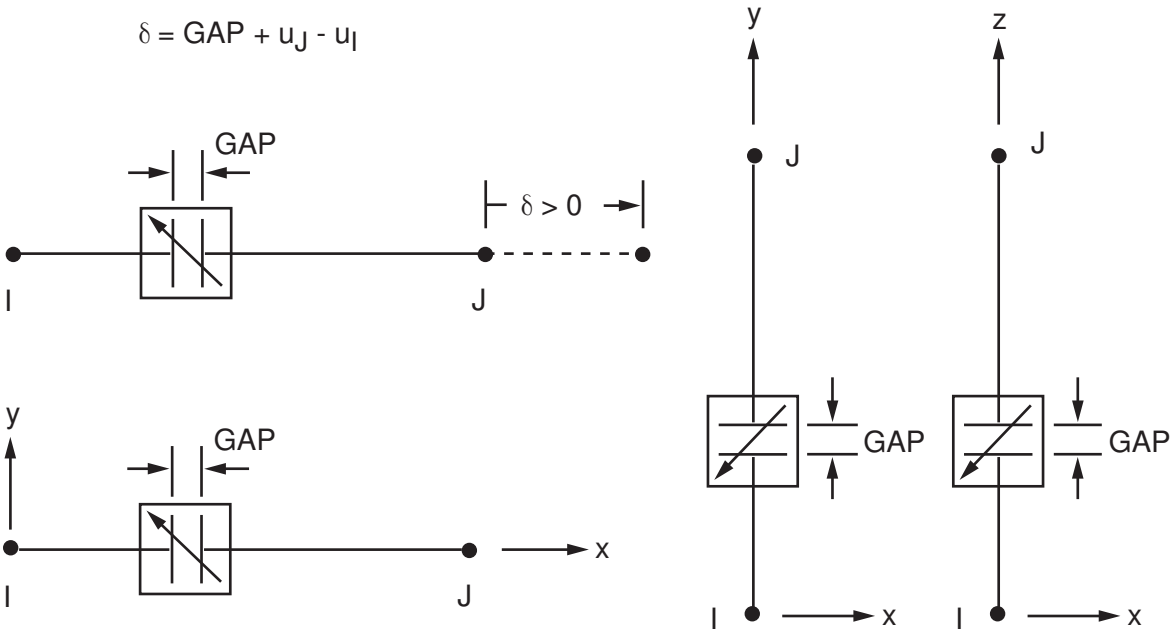
MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

TRANS126 Element Description

TRANS126 represents a transducer element that converts energy from an electrostatic domain into a structural domain (and vice versa), while also allowing for energy storage. The element fully couples the electromechanical domains and represents a reduced-order model suitable for use in structural finite element analysis as well as electromechanical circuit simulation. The element has up to two degrees of freedom at each node: translation in the nodal x, y, or z direction and electric potential (VOLT). The element is suitable for simulating the electromechanical response of micro-electromechanical devices (MEMS) such as electrostatic comb drives, capacitive transducers, and RF switches for example.

The characteristics of the element are derived from electrostatic field simulations of the electromechanical device using the electrostatic elements PLANE121, SOLID122, SOLID123, SOLID127, and SOLID128, as well as the **CMATRIX** macro. The TRANS126 element represents the capacitive response of the device to motion in one direction. Running a series of electrostatic simulations and extracting capacitance (**CMATRIX** command) as a function of stroke (or deflection) provides the necessary input for this element. The capacitance versus stroke represents a "reduced-order" characterization of the device suitable for simulation in this transducer element. Up to three characterizations (in X, Y, or Z) can be made from sets of electrostatic simulations to create three independent transducer elements to characterize a full translational response of the device. See TRANS126 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 TRANS126 Geometry



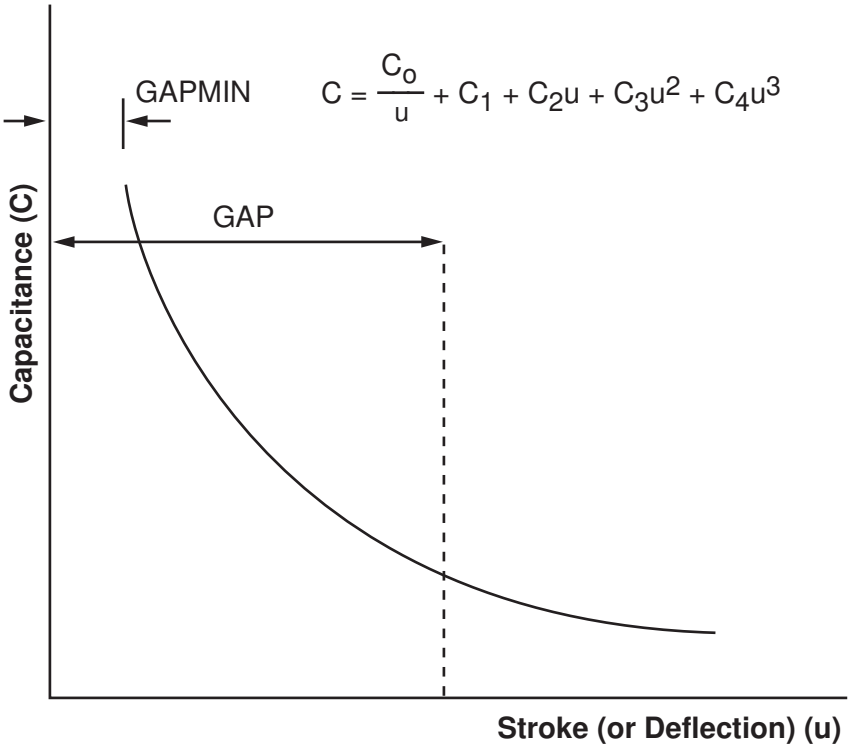
TRANS126 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "TRANS126 Geometry"*. Nodes I and J define the element. The nodes need not be coincident. The element may lie along any one of the three global Cartesian axes as shown in *Figure 1, "TRANS126 Geometry"*, or it may exist in any arbitrary

coordinate system as long as the nodes are rotated into the arbitrary coordinate system in such a manner that one of the axes lies along the element's I-J direction. Use the degree of freedom option (KEYOPT(2)) to select the appropriate structural displacement degree of freedom (corresponding to the element's I-J direction) and electric potential. Orientation of the element with respect to nodal displacements (node J relative to node I) is critical. Orient the element such that a positive movement of node J relative to node I produces a positive displacement (see *Figure 1, "TRANS126 Geometry"*). *Figure 4, "TRANS126 Valid/Invalid Orientations"* illustrates valid and invalid orientations of the element for a UX-VOLT degree of freedom set.

The capacitance vs. stroke data for the element is entered through the real constant table. Use KEYOPT(3) to select from two different methods of input. For KEYOPT(3) = 0, the real constant data (R7-R11) represent the coefficients of an equation (see *Figure 2, "TRANS126 Capacitance Relationship"*). Use as many terms as are required to represent the curve. For KEYOPT(3) = 1, the real constant data (R7-R46) represent discrete pairs of capacitance and stroke data. Up to 20 pairs of data may be input. The minimum required is 5 data point sets. A curve is fit to the discrete data sets represented by the equation shown in *Figure 2, "TRANS126 Capacitance Relationship"*.

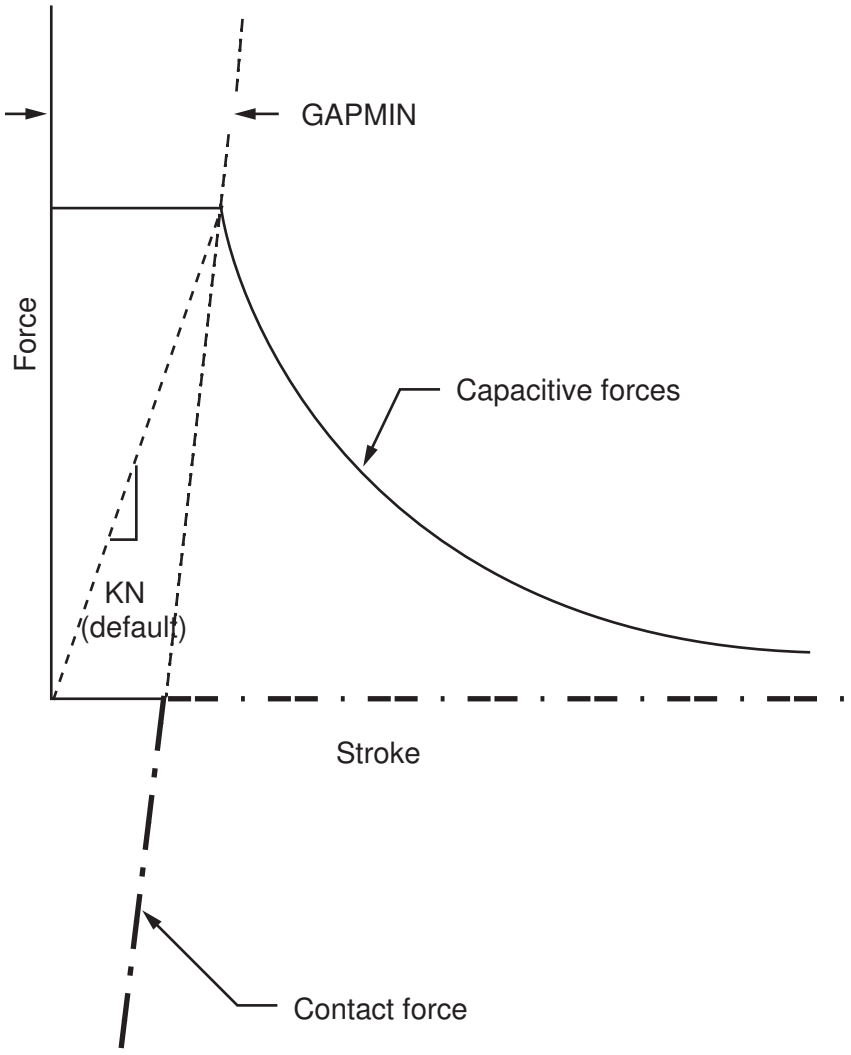
Figure 2 TRANS126 Capacitance Relationship



Capacitance vs. Stroke Representation

The initial gap distance GAP (R3) represents the initial distance between conducting walls of the electromechanical device (that is, plates of a parallel capacitor, beams of a comb drive, etc.). The initial gap value should fall within the range of the capacitance vs. stroke data as shown in *Figure 2, "TRANS126 Capacitance Relationship"*. The minimum gap distance GAPMIN (R4) represents the physical location where the gap is closed. If the gap closes to GAPMIN, the element behaves like a contact element with a normal stiffness KN represented by real constant R5. GAP and GAPMIN default to near-zero if not defined. *Figure 3, "TRANS126 Force Relationship"* illustrates the force vs. stroke for the transducer element. The curve highlights the capacitive force (which is compressive and acts to close the gap), and the contact force (which restrains the motion once the gap reaches GAPMIN). KN defaults to a stiffness represented by the slope from the capacitive force at GAPMIN to the origin as shown in *Figure 3, "TRANS126 Force Relationship"*.

Figure 3 TRANS126 Force Relationship



The element supports nodal voltage and displacements (**D**) as well as nodal current and force (**F**). Use **IC** to input an initial starting value of voltage or displacement for a transient analysis, or an initial guess for a static analysis. The element produces an unsymmetric matrix unless you prescribe both nodal voltages. Prescribing the nodal voltages and setting KEYOPT(4) = 1 produces a symmetric matrix which can yield more efficient solution run times. The KEYOPT(4) = 1 option is valid for any analysis except a transient analysis. The element supports static, prestressed harmonic, transient, and prestressed modal analysis. Prestress effects must be applied for both modal and harmonic analysis (for example, a prestress static analysis with an applied DC voltage, followed by a small-signal (AC voltage) harmonic analysis, or a prestress static analysis with an applied DC voltage followed by a modal analysis). The element is nonlinear for static and transient analysis and requires an iterative solution to converge. The element supports tension only.

The transducer element by nature has both stable and unstable solutions. If the system stiffness is negative, convergence problems can occur near unstable solutions. This typically occurs at small gap distances near GAPMIN. Use KEYOPT(6) = 1 to select the augmented stiffness method if you encounter convergence problems. In this method, the electrostatic stiffness is set to zero to guarantee a positive system stiffness. After convergence is reached, the electrostatic stiffness is automatically reestablished for postprocessing and subsequent analyses.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

TRANS126 Input Summary

Nodes
I, J

Degrees of Freedom
UX-VOLT, UY-VOLT, OR UZ-VOLT

Real Constants

If KEYOPT(3) = 0, then:
GOFFST, EID, GAP, GAPMIN, KN, (Blank),
C0, C1, C2, C3, C4

If KEYOPT(3) = 1, then:
GOFFST, EID, GAP, GAPMIN, KN, (Blank),
GAP1, CAP1, GAP2, CAP2, ..., GAP20, CAP20

See *Table 1, "TRANS126 Real Constants"* for details.

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Prestress

KEYOPT(2)
Select DOF set:

0,1 --
UX-VOLT

2 --
UY-VOLT

3 --
UZ-VOLT

KEYOPT(3)
Capacitance-Gap option:

0 --
Use capacitance-gap curve input coefficients: C0, C1, C2, C3, and C4

1 --
Use capacitance versus gap data points: GAP1, CAP1, GAP2, CAP2 ... GAP20, CAP20

KEYOPT(4)
DC voltage drop option:

0 --
DC voltage drop is unknown (produces unsymmetric matrix)

1 --
DC voltage drop is fully constrained (produces symmetric matrix)

KEYOPT(6)
Stiffness method:

0 --
Full stiffness method (default)

1 --
Augmented stiffness method

The first six real constants for this element are the same, whether you set KEYOPT(3) = 0 or 1. From number 7 on, the real constants differ between the two settings, as shown in the table below.

Table 1 TRANS126 Real Constants

Number	Name	Description
Basic Set		
1	GOFFST	Graphical offset
2	EID	ID number
3	GAP	Initial gap
4	GAPMIN	Minimal gap
5	KN	Gap Normal Stiffness
6	(blank)	unused
For KEYOPT(3) = 0; Capacitance (Cap) vs. gap (x) function: Cap = C0/x + C1 + C2*x + C3*x**2 + C4*x**3		
7	C0	Equation constant C0
8	C1	Equation constant C1
9	C2	Equation constant C2
10	C3	Equation constant C3
11	C4	Equation constant C4
For KEYOPT(3) = 1 (Capacitance-gap curve data)		
7	GAP1	Gap 1
8	CAP1	Capacitance 1
9, ..., 46	GAP2, CAP2, ..., GAP20, CAP20	Gap2 and Capacitance 2 through Gap 20 and Capacitance 20

TRANS126 Output Data

The solution output associated with the element is shown in *Table 2, "TRANS126 Element Output Definitions"*.

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* in the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

If this element is used in a harmonic analysis, all variables will be stored in two-column arrays as complex variables. The first column will be titled real component and the second column will be titled imaginary component. If the variable is not complex, the same value will be stored in both columns.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 TRANS126 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
EFORCE	Electrostatic Force	Y	Y
ESTIFF	Electrostatic stiffness (dEFORCE/dU)	Y	Y
CONDUCT	Motion conductance (dCap/dU) (RELVEL)	Y	Y
DVDT	Time rate of change of Voltage (dVOLT/dt)	Y	Y
RELDISP	Relative displacement node I to node J	Y	Y
RELVEL	Relative velocity node I to node J	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
CAP	Capacitance	Y	Y
MECHPOWER	Mechanical power, (force x velocity)	Y	Y
ELECPOWER	Electrical power, (voltage drop x current)	Y	Y
CENERGY	Electrostatic energy stored in capacitor	Y	Y
GAP	Actual gap, U_J - U_I + GAP (nominal) (real constant input)	Y	Y
KUU	Coupled system stiffness, dF/dU	Y	Y
KUV	Coupled system stiffness, dF/dV	Y	Y
KVU	Coupled system stiffness, dI/dU	Y	Y
KVV	Coupled system stiffness, dI/dV	Y	Y
DUU	Coupled system damping, dF/dVEL	Y	Y
DUV	Coupled system damping, dF/dVRATE	Y	Y
DVU	Coupled system damping, dI/dVEL	Y	Y
DVV	Coupled system damping, dI/dVRATE	Y	Y
DISPR, DISPI	Real and imaginary components of displacement	*1*	*1*
FORCR, FORCI	Real and imaginary components of electrostatic force	*1*	*1*
VOLTR, VOLTI	Real and imaginary components of voltage drop	*1*	*1*
CURRR, CURRI	Real and imaginary components of current	*1*	*1*

1. The item is only available for prestress harmonic analysis.

Table 3, "TRANS126 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "TRANS126 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 3, "TRANS126 Item and Sequence Numbers"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 TRANS126 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MECHPOWER	SMISC	1
ELECPOWER	SMISC	2
CENERGY	SMISC	3
GAP	NMISC	1
RELVEL	NMISC	2
EFORCE	NMISC	3
VOLTAGE	NMISC	4
DVDT	NMISC	5
CURRENT	NMISC	6
CAP	NMISC	7
ESTIFF	NMISC	8
UCT	NMISC	9
KUU	NMISC	10
KUV	NMISC	11
KVU	NMISC	12
KVV	NMISC	13
DUU	NMISC	14
DUV	NMISC	15
DVU	NMISC	16
DVV	NMISC	17
DISPR	NMISC	18
DISPI	NMISC	19
FORCR	NMISC	20
FORCI	NMISC	21
VOLTR	NMISC	22
VOLTI	NMISC	23
CURRR	NMISC	24
CURRI	NMISC	25

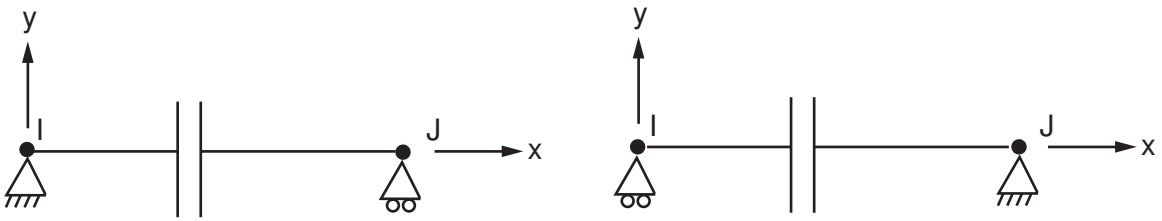
TRANS126 Assumptions and Restrictions

- The transducer element must be aligned such that the element I-J direction points along the active structural degree of freedom in the nodal coordinate system. In addition, a positive movement in the nodal coordinate system of node J relative to node I should act to open the gap (Stroke = GAP + U_j - U_i). *Figure 4, "TRANS126 Valid/Invalid Orientations"* illustrates valid and invalid orientations of the element for a UX-VOLT degree of freedom set.

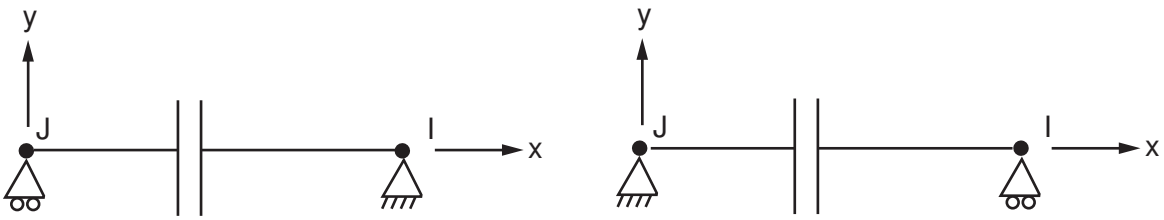
- Nodes I and J may be coincident since the orientation is defined by the relative motion of node J to node I. No moment effects due to noncoincident nodes are included. That is, if the nodes are offset from a line perpendicular to the element axis, moment equilibrium may not be satisfied.
- Unreasonable high stiffness (KN) values should be avoided. The rate of convergence decreases as the stiffness increases.
- The element may not be deactivated with **EKILL**.
- Harmonic and modal analyses are valid only for small-signal analyses after a static prestress calculation.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- A minimum of two load steps must be used to obtain valid electrostatic force calculations.

Figure 4 TRANS126 Valid/Invalid Orientations

Valid orientations for UX,VOLT DOF set:



Invalid orientations for UX,VOLT DOF set:



TRANS126 Product Restrictions

The TRANS126 element is only available in the ANSYS Multiphysics, ANSYS ED, and ANSYS PrepPost products.

SOLID127

3-D Tetrahedral Electrostatic Solid p-Element

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

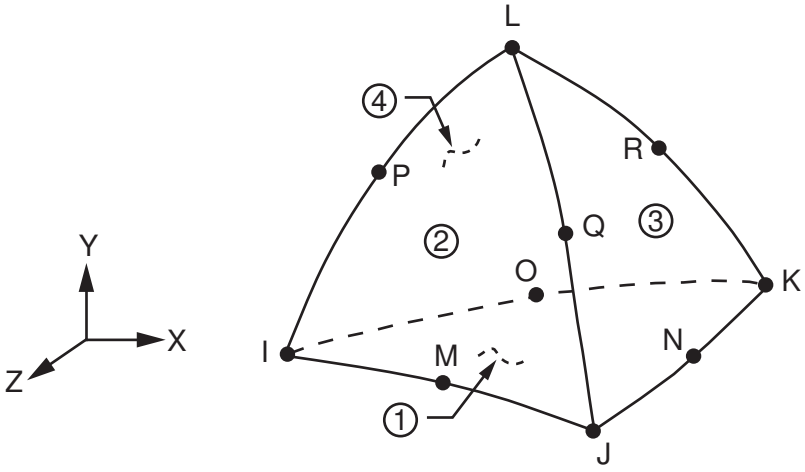
SOLID127 Element Description

SOLID127 is a tetrahedron-shaped p-element that supports a polynomial with a maximum order of eight.

SOLID127 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, voltage, at each node.

The element is applicable to a 3-D, electrostatic field analysis. See SOLID127 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID127 Geometry



SOLID127 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID127 Geometry"*. Midside nodes may not be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface charge densities (CHRGS) or Maxwell surface flags (MXWF) may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID127 Geometry"*. Charge densities may be input as element body loads at the nodes. If the node I charge density CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

A summary of the element input is given in *SOLID127 Input Summary*

SOLID127 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
PERX, PERY, PERZ

Surface Loads

Surface charge densities --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Maxwell surface loads --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Charge densities --

CHRGD(I), CHRGD(J), CHRGD(K), CHRGD(L), CHRGD(M), CHRGD(N), CHRGD(O), CHRGD(P), CHRGD(Q), CHRGD(R)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default)

N --
Maximum possible p-level ($2 \leq N \leq 8$)

SOLID127 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID127 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* in the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID127 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
EF:X, Y, Z	Electric field components (X, Y, Z)	-	Y
EF:SUM	Vector magnitude of EF	-	Y
D:X, Y, Z	Electric flux density components	-	Y
D:SUM	Vector magnitude of D	-	Y
FMAG:X, Y, Z	Maxwell tensor force components	-	*1*
ENERGY	Stored Electric Energy	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.

Table 2, "SOLID127 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SOLID127 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SOLID127 Item and Sequence Numbers"*

Item
: predetermined Item label for **ETABLE**

E
: sequence number for single-valued or constant element data

Table 2 SOLID127 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-Level	NMISC	1

SOLID127 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID127 Geometry"* or in an opposite fashion.
- Nodal charges should only be applied to corner nodes.
- An applied nodal voltage may only vary linearly along an edge or face.
- Constraint equations (CE) can relate a set of corner nodes only, or as set of mid-nodes only.

SOLID127 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

SOLID128

3-D Brick Electrostatic Solid p-Element

MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

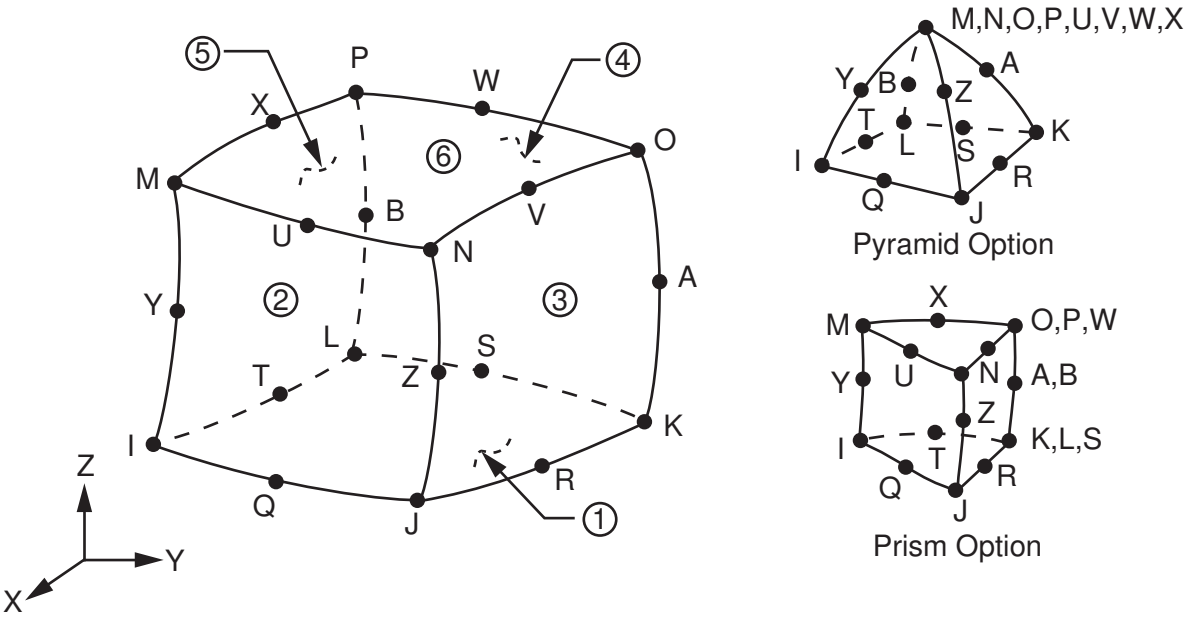
SOLID128 Element Description

SOLID128 is a brick p-element that supports a polynomial with a maximum order of eight.

SOLID128 is a 3-D 20-node solid element. The element has one degree of freedom, voltage, at each node. It can tolerate irregular shapes without much loss of accuracy. SOLID128 elements have compatible voltage shapes and are well suited to model curved boundaries.

The 20-node electrostatic p-element is applicable to a 3-D, electrostatic field analysis. Various printout options are also available. See SOLID128 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID128 Geometry



SOLID128 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID128 Geometry"*. Midside nodes may not be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface charge densities or Maxwell surface flags may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SOLID128 Geometry"*. Charge densities may be input as element body loads at the nodes. If the node I charge density

CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

SOLID128 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
PERX, PERY, PERZ

Surface Loads

Surface charge densities --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Maxwell surface loads --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Charge densities --
CHRGD(I), CHRGD(J), ... , CHRGD(Z), CHRGD(A), CHRGD(B)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default)

N --
Maximum possible p-level ($2 \leq N \leq 8$)

KEYOPT(7)
Store electrostatic forces for coupling with elements:

0 --
Midside-node (higher-order) structural elements

1 --
Non-midside-node structural elements

SOLID128 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID128 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* of the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID128 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B	-	Y
MAT	Material number	-	Y
VOLU	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
EF:X, Y, Z	Electric field components (X, Y, Z)	-	Y
EF:SUM	Vector magnitude of EF	-	Y
D:X, Y, Z	Electric flux density components	-	Y
D:SUM	Vector magnitude of D	-	Y
FMAG:X, Y, Z	Maxwell tensor force components	-	*1*
ENERGY	Stored Electric Energy	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.

Table 2, "SOLID128 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID128 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SOLID128 Item and Sequence Numbers"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 SOLID128 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-Level	NMISC	1

SOLID128 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID128 Geometry"* or in an opposite fashion.
- Nodal charges should only be applied to corner nodes.
- An applied nodal voltage may only vary linearly along an edge or face.
- Constraint equations (CE) can relate a set of corner nodes only, or as set of mid-nodes only.

SOLID128 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

FLUID129

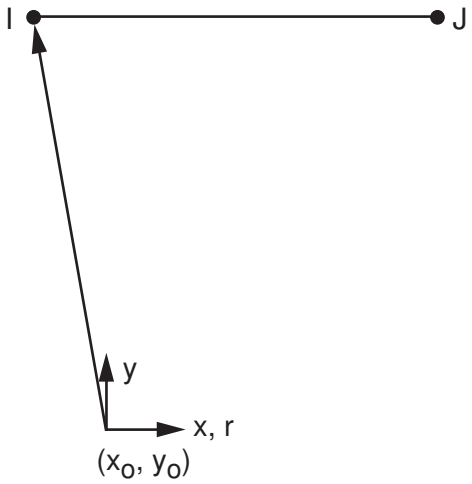
2-D Infinite Acoustic

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID129 Element Description

FLUID129 has been developed as a companion element to FLUID29. It is intended to be used as an envelope to a model made of FLUID29 finite elements. It simulates the absorbing effects of a fluid domain that extends to infinity beyond the boundary of FLUID29 finite element domain. FLUID129 realizes a second-order absorbing boundary condition so that an outgoing pressure wave reaching the boundary of the model is "absorbed" with minimal reflections back into the fluid domain. The element can be used to model the boundary of 2-D (planar or axisymmetric) fluid regions and as such, it is a line element; it has two nodes with one pressure degree of freedom per node. FLUID129 may be used in transient, harmonic, and modal analyses. Typical applications include structural acoustics, noise control, underwater acoustics, etc. See FLUID129 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID129 Geometry



FLUID129 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID129 Geometry"*. The element is defined by two nodes (I, J), the material properties and the real constants (defined in *FLUID129 Input Summary*). The element must be circular with radius RAD and center located at or near the center of the structure. The radius RAD should be supplied through the real constants.

The element is characterized by a pair of symmetric stiffness and damping matrices.

In a typical meshing procedure, you should mesh the interior fluid domain that is bounded by a circular boundary with FLUID29 elements, select the nodes on the circular boundary, select the type associated with the FLUID129 and then issue the **ESURF** command. The latter will automatically add the FLUID129 elements on the boundary of the finite domain.

FLUID129 Input Summary

Nodes
I, J

Degrees of Freedom
PRES

Real Constants

RAD - Radius
X_0 - Center of enclosing circle, X value
Y_0 - Center of enclosing circle, Y value

Material Properties
SONC - velocity of sound

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(3)
Element behavior:

0 --
Planar

1 --
Axisymmetric

FLUID129 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID129 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID129 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
LINE:	Length	Y	Y
XC, YC	Location where results are reported	Y	*1*

Name	Definition	O	R
SONC	Speed of sound	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "FLUID129 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID129 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID129 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID129 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SONC	NMISC	1

FLUID129 Assumptions and Restrictions

- FLUID129 must lie on a boundary circular in shape and should completely enclose the domain meshed with FLUID29 elements.
- The radius RAD of the circular boundary of the finite domain should be specified as a real constant. If the coordinates (X_0, Y_0) of the center of the circle are not supplied through the real constant input, the center will be assumed to be at the origin. The center of the circle should be as close to the center of the model as possible.
- It is recommended that the enclosing circular boundary is placed at a distance of at least 0.2*lambda from the boundary of any structure that may be submerged in the fluid, where lambda = c/f is the dominant wavelength of the pressure waves; c is the speed of sound (SONC) in the fluid, and f is the dominant frequency of the pressure wave. For example, in the case of a submerged circular cylindrical shell of diameter D, the radius of the enclosing boundary, RAD, should be at least (D/2) + 0.2*lambda.
- FLUID129 uses an extra DOF, labeled XTR1, that is not available to the user. This DOF is solely for ANSYS' internal use, although it may appear in DOF listings or in program messages.
- The only applicable modal analysis method is the Damped method.

FLUID129 Product Restrictions

There are no product-specific restrictions for this element.

FLUID130

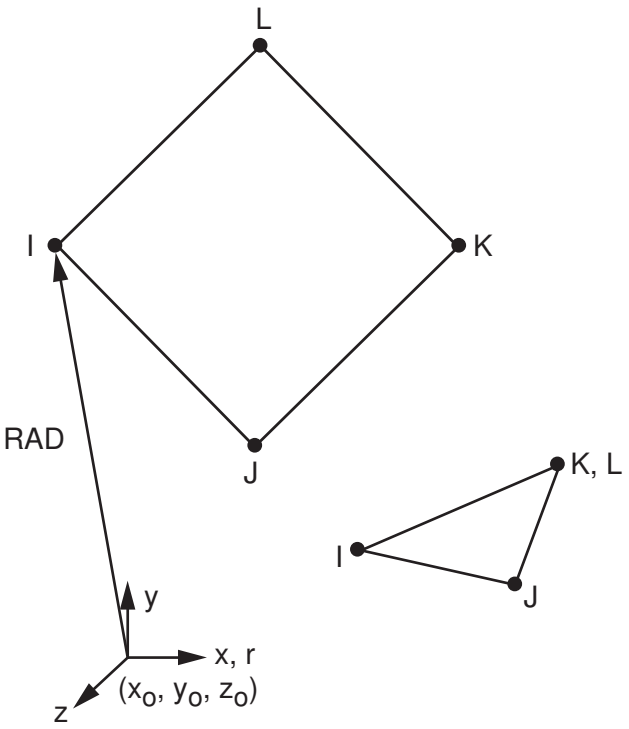
3-D Infinite Acoustic

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID130 Element Description

FLUID130 has been developed as a companion element to FLUID30. It is intended to be used as an envelope to a model made of FLUID30 finite elements. It simulates the absorbing effects of a fluid domain that extends to infinity beyond the boundary of the finite element domain that is made of FLUID30 elements. FLUID130 realizes a second-order absorbing boundary condition so that an outgoing pressure wave reaching the boundary of the model is "absorbed" with minimal reflections back into the fluid domain. The element can be used to model the boundary of 3-D fluid regions and as such, it is a plane surface element; it has four nodes with one pressure degrees of freedom per node. FLUID130 may be used in transient, harmonic, and modal analyses. Typical applications include structural acoustics, noise control, underwater acoustics, etc. See FLUID130 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 FLUID130 Geometry



FLUID130 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FLUID130 Geometry"*. The element is defined by four nodes (I, J, K, L), the material property SONC (speed of sound) and the real constants shown in *FLUID130 Input Summary*. A triangular element may be formed by defining duplicate K and L node numbers. The element must be at the spherical boundary of an acoustic fluid domain, meshed using FLUID30 elements, with radius RAD and center located at or near the center of the structure. The radius RAD should be supplied through the real constants.

The element is characterized by a symmetric stiffness and a damping matrix.

In a typical meshing procedure the user should mesh the interior fluid domain that is bounded by a spherical boundary with FLUID30 elements, select the nodes on the spherical boundary, select the type associated with the FLUID130 and then issue the **ESURF** command. The latter will automatically add the FLUID130 elements on the boundary of the finite domain.

FLUID130 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
PRES

Real Constants

RAD - Radius
X_0 - Center of enclosing circle, X value
Y_0 - Center of enclosing circle, Y value
Z_0 - Center of enclosing circle, Z value

Material Properties
SONC

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

FLUID130 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID130 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID130 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA:	AREA	Y	Y
XC, YC	Location where results are reported	Y	*1*
SONC	Speed of sound	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2, "FLUID130 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID130 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID130 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID130 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SONC	NMISC	1

FLUID130 Assumptions and Restrictions

- FLUID130 must lie on a boundary spherical in shape and should completely enclose the domain meshed with FLUID30 elements.
- The radius RAD of the spherical boundary of the finite domain should be specified as a real constant. If the coordinates (X_0, Y_0, Z_0) of the center of the sphere are not supplied through the real constant input, the center will be assumed to be at the origin of the global coordinate system. The center of the sphere should be as close to the center of the model as possible.
- It is recommended that the enclosing spherical boundary is placed at a distance of at least 0.2*lambda from the boundary of any structure that may be submerged in the fluid, where lambda = c/f is the dominant wavelength of the pressure waves. c is the speed of sound (SONC) in the fluid and f is the dominant frequency of the pressure wave. For example, in the case of a submerged spherical shell of diameter D, the radius of the enclosing boundary, RAD, should be at least (D/2) + 0.2*lambda.
- FLUID130 uses extra DOFs, labeled XTR1 and XTR2, that are not available to the user. These DOFs are solely for ANSYS' internal use, although they may appear in DOF listings or in program messages.
- The only applicable modal analysis method is the Damped method.

FLUID130 Product Restrictions

There are no product-specific restrictions for this element.

SHELL131

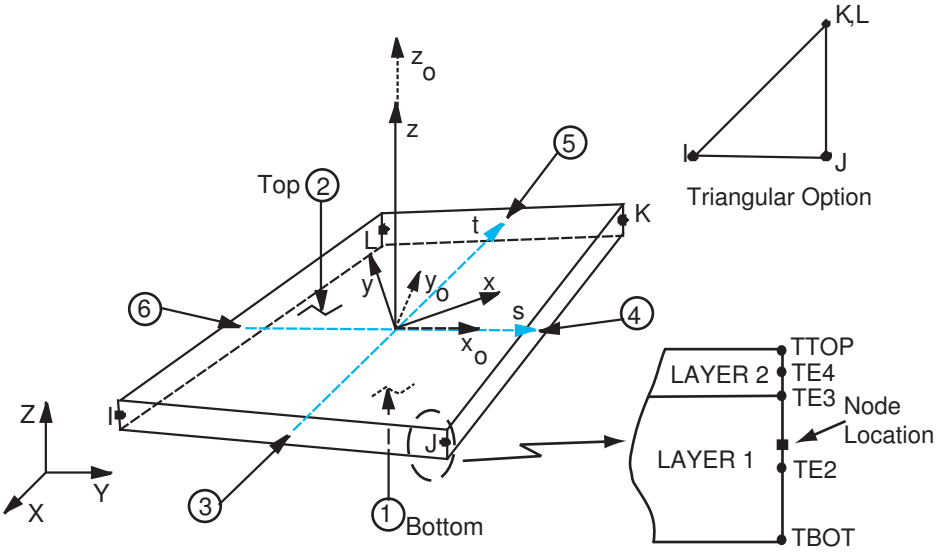
4-Node Layered Thermal Shell

MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL131 Element Description

SHELL131 is a 3-D layered shell element having in-plane and thru-thickness thermal conduction capability. The element has four nodes with up to 32 temperature degrees of freedom at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis. SHELL131 generates temperatures that can be passed to structural shell elements in order to model thermal bending. See SHELL131 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing the conducting shell element is to be analyzed structurally, the element should be replaced by an equivalent structural element such as SHELL43, SHELL63, SHELL181, or SHELL281.

Figure 1 SHELL131 Geometry



x_o = element x-axis if ESYS is not supplied.

x = element x-axis if ESYS is supplied.

SHELL131 Input Data

The geometry, node locations, and coordinates systems for this element are shown in *Figure 1, "SHELL131 Geometry"*. The element is defined by four nodes, one thickness per layer, a material angle for each layer, and the material properties. If the material is uniform and the analysis has no transient effects, only one layer is needed with a linear temperature variation through the thickness.

The cross-sectional properties are input using the **SECTYPE**,,SHELL and **SECDATA** commands. These properties are the thickness, material number, and orientation of each layer. Tapered thicknesses may be input using the **SECFUNCTION** command. The number of integration points from the **SECDATA** command is not used; rather it is determined for all layers with KEYOPT(3). In the GUI, the ShellTool provides a convenient way to define section data for this element (see Shell Analysis and Cross Sections in the *Structural Analysis Guide*). Real constants are not used for this element.

Generally, the quadratic variation in temperature through each layer (KEYOPT(3) = 0) is used for transient analysis or for strongly temperature dependent materials, and the linear variation in temperature through each layer (KEYOPT(3) = 1) is used for steady state analysis with materials that are either not temperature dependent or weakly temperature dependent. Layers may be used to model the physical changes of properties through the thickness or the effect of a thru-thickness transient in greater detail.

KEYOPT(4) duplicates the number of layers input on the **SECDATA** commands. If KEYOPT(4) is 0 or blank, the program will query each element during definition in PREP7 as to which section information is being used, and then reassign the element to a different type. More element types are created as needed. The result can be seen using **ETLIST** and **ELIST** after all elements are defined. To ensure that the program can do this redefinition, the user is required to define the section information before the element is defined.

If KEYOPT(6) (also referred to as the paint option) is used, TBOT is replaced with TEMP, allowing the element to be directly attached to an underlying solid to avoid the use of constraint equations. When this option is used, surface loads cannot be applied to face 1.

As this is a thermal shell element, the direction of the element z-axis and the presence of the **SECOFFSET** command have no effect on the solution. However, to get correct plots when using the **/ESHAPE** command:

- The element z-axis should be defined with the same care as for a structural shell element.
- If KEYOPT(6) = 1 (the paint option) is set, **SECOFFSET**,BOT should be input.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation (using the RDSF surface load label) may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SHELL131 Geometry"*. Edge convection and flux loads are input on a per unit length basis. Radiation is not available on the edges. You can also generate film coefficients and bulk temperatures using the surface effect element SURF152. SURF152 can also be used with FLUID116.

Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1). Nodal values are averaged over the entire element.

A summary of the element input is given in *SHELL131 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL131 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

Quadratic:

If KEYOPT(3) = 0
If KEYOPT(4) = 0 or 1: TBOT, TE2, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TE3, TE4, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TE4, TE5, TE6, TTOP
Etc.
If KEYOPT(4) = 15: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TTOP

Linear:

If KEYOPT(3) = 1
If KEYOPT(4) = 0 or 1: TBOT, TTOP

If KEYOPT(4) = 2: TBOT, TE2, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TTOP
Etc.
If KEYOPT(4) = 31: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TE31, TTOP

Constant:

If KEYOPT(3) = 2: TEMP (one layer only, essentially the same as SHELL57)

Real Constants

None

Material Properties

KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Heat Fluxes --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Radiation --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)

Body Loads

Heat Generations --
HG(1), HG(2), HG(3), ..., HG(KEYOPT(4))

Special Features

Birth and death

KEYOPT(2)

Film coefficient evaluation (if any):

0 --
Evaluate at an average film temperature, (TS+TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS-TB|

KEYOPT(3)

Temperature variation through layer:

0 --
Quadratic temperature variation thru-layer (maximum number of layers = 15)

1 --
Linear temperature variation thru-layer (maximum number of layers = 31)

2 --
No temperature variation thru-layer (number of layers = 1)

KEYOPT(4)
Number of layers (input a value to match **SECDATA** commands, or leave blank to default). Maximum number of layers allowed depends on KEYOPT(3) setting (see above).

KEYOPT(6)
Application:

0 --
Thermal shell application

1 --
Paint application

SHELL131 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output shown in *Table 1, "SHELL131 Element Output Definitions"*

Output temperatures may be read by structural shell elements using the **LDREAD**,TEMP command.

If the structural shell element uses *only one* temperature through the thickness, such as SHELL41, only TEMP can be used.

If the structural shell element uses *two* temperatures through the thickness, such as for SHELL43, SHELL63, SHELL181 (with only one layer), or SHELL281 (with only one layer), only TBOT and TTOP are used and any internal temperatures such as TE2 are ignored.

If the structural shell element uses *more than two* temperatures through the thickness, such as for SHELL181 (with multiple layers), all temperatures are transferred over. In this case, the corner nodes of each SHELL131 element must have identical temperature degrees of freedom.

The number of temperature points at a node generated in the thermal shell must match the number of temperature points at a node needed by the structural shell. For example, a two-layer SHELL181 element using the same material and thickness for both layers can get its temperatures from a SHELL131 element using either two layers with KEYOPT(3) = 1 (linear variation) or one layer with KEYOPT(3) = 0 (quadratic variation). Temperatures passed from this element to the stress analysis via **LDREAD**,TEMP can be viewed using **BFELIST**, as opposed to the usual **BFLIST**.

Heat flowing out of the element is considered to be positive. Heat flows are labeled HBOT, HE2, . . . HTOP, similar to the temperature labels. Gradient and flux information is provided at the midthickness of each layer. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

To see the temperature distribution thru the thickness for this element as well as all other thermal elements, use **/GRAPHICS**,POWER and **/ESHAPE**,1 followed by **PLNSO**L,TEMP.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL131 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Element material number (from **MAT** command)	Y	Y
AREA	Area of element	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations: HG(1), HG(2), HG(3), . . .	Y	-
TG:X, Y, Z	Thermal gradient components at integration points	Y	Y
TF:X, Y, Z	Thermal flux components at integration points	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area (same as element area)	*1*	*1*
NODES	Face nodes (same as element nodes)	*1*	*1*
HFILM	Face film coefficient	*1*	*1*
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HEAT FLUX	Heat flux at each node of the face	*1*	-

1. If a surface load is input.
2. Available only at the centroid as a ***GET** item.

Table 4, "SHELL41 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL131 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "SHELL131 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

Table 2 SHELL131 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Face 1 (Bot)	Face 2 (Top)
AREA	NMISC	1	7
HFAVG	NMISC	2	8
TAVG	NMISC	3	9
TBAVG	NMISC	4	10
HEAT RATE	NMISC	5	11
HFLXAVG	NMISC	6	12

SHELL131 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the element is not numbered properly.
- Zero thickness layers are not allowed.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- The cut boundary interpolation command (**CBDOF**) does not work with this element.
- When using thermal contact, the TEMP degree of freedom must be present (KEYOPT(3) = 2 or KEYOPT(6) = 1).
- There should not be a large variation in the ratio of through-thickness conductivity (KZZ) to layer thickness for all layers within the element. If the highest and lowest values for this ratio differ by a large factor (for example, 1e5), then the results for the element may be unreliable.
- No check is made to ensure either that the number of layers between adjacent elements match or that the effective location of a degree of freedom (for example, TE7 from a 10 layer element) between elements sharing the same node is the same to a tolerance. If this is a concern, study the area using the **/ESHAPE** command. For cases where the layering intentionally changes, such as at a joint or at the runout of a tapered layer, use constraint equations (**CE** family of commands) with or without double nodes to connect the two sides.
- The program removes all imposed degrees of freedom and nodal loads (i.e., internally issues **DDELE**,all,all and **FDELE**,all,all commands) when elements that use TTOP, TBOT, etc. as degrees of freedom:
  - are defined or redefined using the **ET** or **KEYOPT** commands.
  - are changed (or deleted) using the **ET**, **ETCHG**, or **ETDELE** commands to an element type that does not use these degrees of freedom.

    If your model contained SHELL131 elements with **D** and **F** loads, and you deleted these elements via **ETDELE**, the **D** and **F** loads will automatically be deleted and reapplied to the new DOF list. You do, however, need to check other loads and verify if they need to be deleted and reapplied.

SHELL131 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS ED

- Section definitions are not allowed if more than one material is referenced.

SHELL132

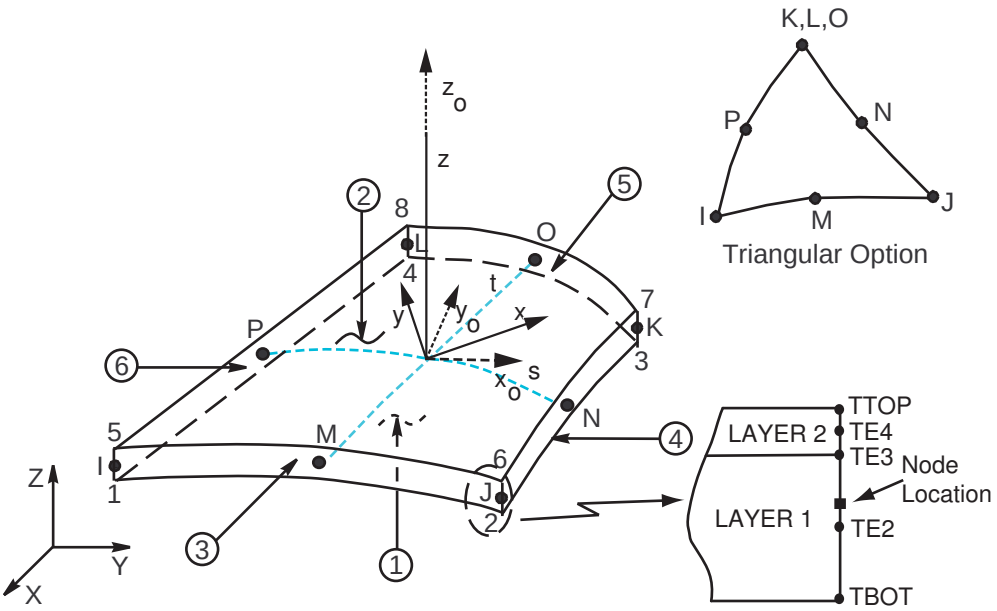
8-Node Layered Thermal Shell

MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL132 Element Description

SHELL132 is a 3-D layered shell element having in-plane and thru-thickness thermal conduction capability. The element has eight nodes with up to 32 temperature degrees of freedom at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis. SHELL132 generates temperatures that can be passed to structural shell elements in order to model thermal bending. See SHELL132 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If the model containing the conducting shell element is to be analyzed structurally, the element should be replaced by an equivalent structural element such as SHELL91, SHELL93, or SHELL99.

Figure 1 SHELL132 Geometry



x_o = element x-axis if ESYS is not supplied.

x = element x-axis if ESYS is supplied.

SHELL132 Input Data

The geometry, node locations, and coordinates systems for this element are shown in *Figure 1, "SHELL132 Geometry"*. The element is defined by four/eight nodes, one thickness per layer, a material angle for each layer, and the material properties. If the material is uniform and the analysis has no transient effects, only one layer is needed with a linear temperature variation through the thickness.

The cross-sectional properties are input using the **SECTYPE**,,SHELL and **SECDATA** commands. These properties are the thickness, material number, and orientation of each layer. Tapered thicknesses may be input using the **SECFUNCTION** command. The number of integration points from the **SECDATA** command is not used; rather it is determined for all layers with KEYOPT(3). In the GUI, the ShellTool provides a convenient way to define section data for this element (see Shell Analysis and Cross Sections in the *Structural Analysis Guide*). Real constants are not used for this element.

Generally, the quadratic variation in temperature through each layer (KEYOPT(3) = 0) is used for transient analysis or for strongly temperature dependent materials, and the linear variation in temperature through each layer (KEYOPT(3) = 1) is used for steady state analysis with materials that are either not temperature dependent or weakly temperature dependent. Layers may be used to model the physical changes of properties through the thickness or the effect of a thru-thickness transient in greater detail.

KEYOPT(4) duplicates the number of layers input on the **SECDATA** commands. If KEYOPT(4) is 0 or blank, the program will query each element during definition in PREP7 as to which section information is being used, and then reassign the element to a different type. More element types are created as needed. The result can be seen using **ETLIST** and **ELIST** after all elements are defined. To ensure that the program can do this redefinition, the user is required to define the section information before the element is defined.

If KEYOPT(6) (also referred to as the paint option) is used, TBOT is replaced with TEMP, allowing the element to be directly attached to an underlying solid to avoid the use of constraint equations. When this option is used, surface loads cannot be applied to face 1.

As this is a thermal shell element, the direction of the element z-axis and the presence of the **SECOFFSET** command have no effect on the solution. However, to get correct plots when using the **/ESHAPE** command:

- The element z-axis should be defined with the same care as for a structural shell element.
- If KEYOPT(6) = 1 (the paint option) is set, **SECOFFSET**,BOT should be input.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation (using the RDSF surface load label) may be input as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SHELL132 Geometry"*. Edge convection and flux loads are input on a per unit length basis. Radiation is not available on the edges. You can also generate film coefficients and bulk temperatures using the surface effect element SURF152. SURF152 can also be used with FLUID116.

Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1). Nodal values are averaged over the entire element.

A summary of the element input is given in *SHELL132 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL132 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

Quadratic:

If KEYOPT(3) = 0
If KEYOPT(4) = 0 or 1: TBOT, TE2, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TE3, TE4, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TE4, TE5, TE6, TTOP
Etc.
If KEYOPT(4) = 15: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TTOP

Linear:

If KEYOPT(3) = 1
If KEYOPT(4) = 0 or 1: TBOT, TTOP

If KEYOPT(4) = 2: TBOT, TE2, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TTOP
Etc.
If KEYOPT(4) = 31: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TE31, TTOP

Constant:

If KEYOPT(3) = 2: TEMP (one layer only, essentially the same as SHELL57)

Real Constants

None

Material Properties

KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Heat Fluxes --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Radiation --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)

Body Loads

Heat Generations --
HG(1), HG(2), HG(3), ..., HG(KEYOPT(4))

Special Features

Birth and death

KEYOPT(2)

Film coefficient evaluation (if any):

0 --
Evaluate at an average film temperature, (TS+TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS-TB|

KEYOPT(3)

Temperature variation through layer:

0 --
Quadratic temperature variation through layer (maximum number of layers = 15)

1 --
Linear temperature variation through layer (maximum number of layers = 31)

2 --
No temperature variation through layer (number of layers = 1)

KEYOPT(4)
Number of layers (input a value to match **SECDATA** commands, or leave blank to default). Maximum number of layers allowed depends on KEYOPT(3) setting (see above).

KEYOPT(6)
Application:

0 --
Thermal shell application

1 --
Paint application

SHELL132 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL132 Element Output Definitions"*.

Output nodal temperatures may be read by structural shell elements using the **LDREAD**,TEMP capability. If the structural shell element uses two temperatures thru the thickness such as for SHELL93 and SHELL99, only TBOT and TTOP are used and any internal temperatures such as TE2 are ignored. If the structural shell element uses more than two temperatures through the thickness such as for SHELL91, all temperatures are transferred over. In this case, the corner nodes of each SHELL132 element must have identical temperature degrees of freedom. Also, the number of temperature points at a node generated in the thermal shell must match the number of temperature points at a node needed by the structural shell. For example, a two layer SHELL91 element using the same material and thickness for both layers can get its temperatures from a SHELL132 element using either two layers with KEYOPT(3) = 1 (linear variation) or one layer with KEYOPT(3) = 0 (quadratic variation). Temperatures passed from this element to the stress analysis via **LDREAD**,TEMP can be viewed using **BFELIST**, as opposed to the usual **BFLIST**.

Heat flowing out of the element is considered to be positive. Heat flows are labeled HBOT, HE2, . . . HTOP, similar to the temperature labels. Gradient and flux information is provided at the midthickness of each layer. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

To see the temperature distribution thru the thickness for this element as well as all other thermal elements, use **/GRAPHICS**,POWER and **/ESHAPE**,1 followed by **PLNSO**L,TEMP.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL132 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Element material number (from **MAT** command)	Y	Y
AREA	Area of element	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations: HG(1), HG(2), HG(3), . . .	Y	-
TG:X, Y, Z	Thermal gradient components at integration points	Y	Y
TF:X, Y, Z	Thermal flux components at integration points	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area (same as element area)	*1*	*1*
NODES	Face nodes (same as element nodes)	*1*	*1*
HFILM	Face film coefficient	*1*	*1*
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HFAVG	Average film coefficient of the face	-	*1*
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	*1*
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	*1*	-
HEAT FLUX	Heat flux at each node of the face	*1*	-

1. If a surface load is input.
2. Available only at the centroid as a *GET item.

Table 4, "SHELL41 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL131 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL132 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

Table 2 SHELL131 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Face 1 (Bot)	Face 2 (Top)
AREA	NMISC	1	7
HFAVG	NMISC	2	8
TAVG	NMISC	3	9
TBAVG	NMISC	4	10
HEAT RATE	NMISC	5	11

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Face 1 (Bot)	Face 2 (Top)
HFLXAVG	NMISC	6	12

SHELL132 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the element is not numbered properly.
- Zero thickness layers are not allowed.
- A triangular element may be formed by defining duplicate K, L, and O node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*.
- Midside nodes may not be dropped.
- The cut boundary interpolation command (**CBDOF**) does not work with this element.
- When using thermal contact, the TEMP degree of freedom must be present (KEYOPT(3) = 2 or KEYOPT(6) = 1).
- There should not be a large variation in the ratio of through-thickness conductivity (KZZ) to layer thickness for all layers within the element. If the highest and lowest values for this ratio differ by a large factor (for example, 1e5), then the results for the element may be unreliable.
- No check is made to ensure either that the number of layers between adjacent elements match or that the effective location of a degree of freedom (for example, TE7 from a 10 layer element) between elements sharing the same node is the same to a tolerance. If this is a concern, study the area using the **/ESHAPE** command. For cases where the layering intentionally changes, such as at a joint or at the runout of a tapered layer, use constraint equations (**CE** family of commands) with or without double nodes to connect the two sides.
- This element may not be used with **THOPT**,QUASI if convection or radiation surfaces are present.
- This element may not be used with the **/EFACET** command for PowerGraphics displays.
- The program removes all imposed degrees of freedom and nodal loads (i.e., internally issues **DDELE**,all,all and **FDELE**,all,all commands) when elements that use TTOP, TBOT, etc. as degrees of freedom:
  - are defined or redefined using the **ET** or **KEYOPT** commands.
  - are changed (or deleted) using the **ET**, **ETCHG**, or **ETDELE** commands to an element type that does not use these degrees of freedom.

    If your model contained SHELL132 elements with **D** and **F** loads, and you deleted these elements via **ETDELE**, the **D** and **F** loads will automatically be deleted and reapplied to the new DOF list. You do, however, need to check other loads and verify if they need to be deleted and reapplied.

SHELL132 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS ED

- Section definitions are not allowed if more than one material is referenced.

FLUID136

3-D Squeeze Film Fluid Element

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID136 Element Description

FLUID136 models viscous fluid flow behavior in small gaps between fixed surfaces and structures moving perpendicular to the fixed surfaces. FLUID136 can be used to determine the stiffening and damping effects that the fluid exerts on the moving structure. The element behavior is based on the Reynolds squeeze film theory and the theory of rarefied gases. As such, it is limited to structures with lateral dimensions much greater than the gap size. In addition, the pressure change must be small relative to the ambient pressure, and any viscous heating is neglected. FLUID136 is particularly applicable to modeling squeeze-film effects in microstructures. However, it can also model thin-film fluid behavior in macrostructures.

If the velocity of the moving surface is known, FLUID136 can directly determine the fluid response. The velocity normal to the element surface is specified as a body force. If the velocity of the moving surface is not known, FLUID136 can determine the fluid response from the eigenmodes of the structure using the Modal Projection Method.

FLUID136 is applicable to static, harmonic, and transient analyses. A static analysis is used to determine the damping effects for low operating frequencies where fluid stiffening effects are negligible. A harmonic analysis is used to determine the fluid stiffening and damping effects for high operating frequencies where fluid stiffening effects are not negligible. A transient analysis is used to determine the fluid stiffening and damping effects for non-harmonic loadings. The Modal Projection Method can also be used to extract frequency-dependent damping ratios for use with the **MDAMP** and **DMPRAT** commands; and Alpha and Beta damping parameters for use with the **ALPHAD** and **BETAD** commands.

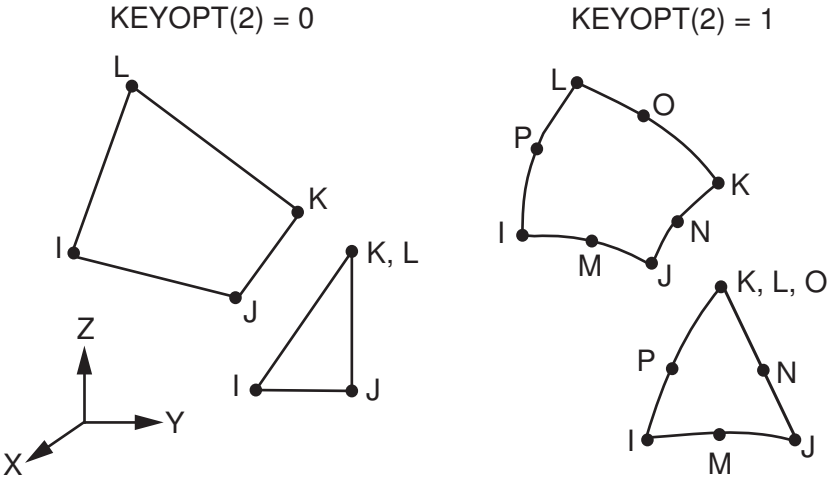
FLUID136 can be used to model three different flow regimes: continuum theory, high Knudsen number, and high Knudsen number with accommodation factors.

See FLUID136 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

FLUID136 Input Data

The element is defined by four corner nodes with an option to include mid-side nodes (KEYOPT(2) = 1). The element should be oriented such that the element normal is pointing toward the fluid domain. If solid elements are used for the structural domain, the fluid element normal vector is automatically computed. If necessary, the fluid element normal vector can be flipped using **ENSYM**.

Figure 1 FLUID136 Geometry



KEYOPT (1) specifies the flow regime. The Knudsen number can be calculated from the mean free fluid path at a reference pressure, the operating pressure, and the gap.

K_n = (MFP*PREF) / (PAMB*GAP

For continuum theory to be valid (KEYOPT(1) = 0), the Knudsen number should be less than 0.01. If the Knudsen number is greater than 0.01 (KEYOPT(1) = 1 or 2), the dynamic viscosity is adjusted to account for the slip flow boundary. See *Section 16.2.3: Flow Regime Considerations* in the *Fluids Analysis Guide* for a complete discussion of flow regimes and calculation of the Knudsen number.

The type of reflection of the gas molecules at the wall interface is specified using accommodation factors. Squeeze film models assume diffuse reflection of the gas molecules at the wall interface (accommodation factor = 1). This assumption is valid for most metals, but is less accurate for micromachined surfaces, particularly those fabricated from silicon. Materials, such as silicon, cause specular reflection. Typical accommodation factors for silicon are between 0.80 and 0.90.

The fluid environment is defined by a set of real constants:

GAP specifies the local gap separation, PAMB specifies the ambient (i.e., surrounding) pressure, ACF1 and ACF2 specify the accommodation factors surface1 and surface 2, PREF specifies the reference pressure for the mean free fluid path, and MFP specifies the mean free fluid path at reference pressure PREF.

For continuum theory (KEYOPT(1) = 1), GAP and PAMB must be specified.

For high Knudsen numbers (KEYOPT(1) = 1), GAP, PAMB, PREF, and MFP must be specified. PREF and MFP are used to adjust the dynamic viscosity. ACF1 and ACF2 are assumed to be 1.

For high Knudsen numbers with accommodation factors (KEYOPT(1) = 2), GAP, PAMB, PREF, MFP, ACF1, and ACF2 must be specified. Different accommodation factors may be specified for each surface.

For small deflections, GAP is assumed to be constant. For large deflections, GAP can be updated using **SETFGAP**.

The fluid velocity normal to the surface may be specified using nodal or element loading with the FLUE body load label on the **BF** or **BFE** commands. If FLUID136 is used in conjunction with the Modal Projection Method, the fluid velocities are obtained from the modal displacements and applied using the **DMPEXT** command.

FLUID136 Input Summary

Nodes
I, J, K, L (KEYOPT(2) = 0)

I, J, K, L, M, N, O, P (KEYOPT(2) = 1)

Degrees of Freedom
PRES

Real Constants
GAP, (blank), (blank), PAMB, ACF1, ACF2

PREF, MFP

Material Properties
VISC - dynamic viscosity

Surface Loads
None

Body Loads
FLUE (velocity)

Special Features
None

KEYOPT(1)
Continuous flow options

0 --
Continuum theory

1 --
High Knudsen numbers (greater than 0.01)

2 --
High Knudsen numbers and accommodation factors

KEYOPT(2)
Element geometry

0 --
Four node element

1 --
Eight node element

FLUID136 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID136 Element Output Definitions"*

A general description of solution output is given in *Table 1, "FLUID136 Element Output Definitions"*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID136 Element Output Definitions

Name	Definition	O	R
PRES	Pressure change with regard to ambient temperature		Y
PG (X, Y, Z)	Mid-surface fluid velocity	Y	Y
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA:	Area	Y	Y
FLUE	Velocity (normal to surface)	Y	Y

Table 2, "FLUID136 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID136 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "FLUID136 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 FLUID136 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
GAP	NMISC	2

FLUID136 Assumptions and Restrictions

- Knudsen numbers larger than 880 are not supported.
- The gas flow is assumed to be isothermal.
- The pressure change must be small compared to ambient pressure.
- Displacement amplitudes must be small compared to the film thickness.
- The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**). See *Section 7.8: Squeeze Film Theory* in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on the governing equations.

FLUID136 Product Restrictions

There are no product-specific restrictions for this element.

FLUID138

3-D Viscous Fluid Link Element

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

FLUID138 Element Description

FLUID138 models the viscous fluid flow behavior through short channels (i.e., holes) in microstructures moving perpendicular to fixed surfaces. FLUID138 can be used in conjunction with FLUID136 elements to determine the stiffening and damping effects that the fluid exerts on the moving perforated microstructure.

FLUID138 assumes isothermal flow at low Reynolds numbers. The channel length must be small relative to the acoustic wave length, and the pressure change must be small relative to the ambient pressure. FLUID138 accounts for gas rarefaction effects and fringe effects due to the short channel length.

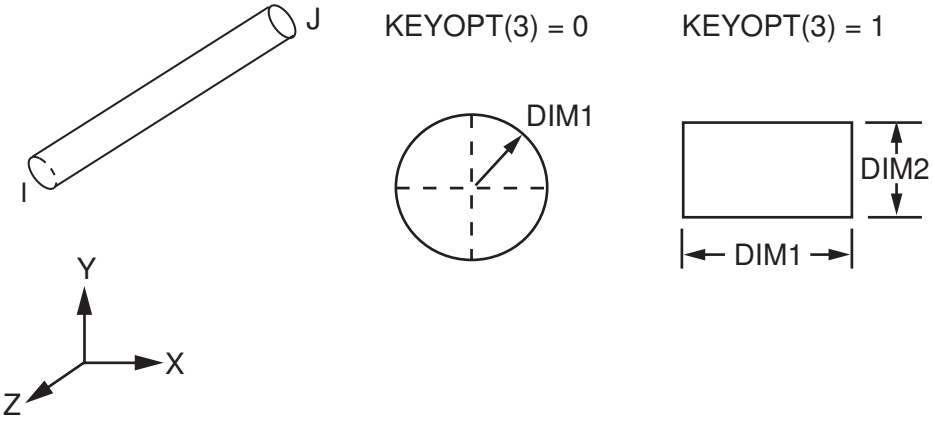
As with FLUID136, FLUID138 is applicable to static, harmonic, and transient analyses. FLUID138 can be used to model two different flow regimes: continuum theory and high Knudsen number.

In contrast to FLUID116, this element is more accurate for channels of rectangular cross section, allows channel dimensions to be small compared to the mean free path, allows modeling of evacuated systems, and considers fringe effects at the inlet and outlet. These effects can considerably increase the damping force in the case of short channel length. See *Section 14.138: FLUID138 - 3-D Viscous Fluid Link Element* in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

FLUID138 Input Data

The element is defined by two nodes. The I node is located at the center of the cross-section of the hole region on the same plane as the nodes used to model the squeeze film fluid region (FLUID136 elements). The J node is located at the opposite face of the structure through the channel depth.

Figure 1 FLUID138 Geometry



KEYOPT(1) specifies the flow regime. The Knudsen number can be calculated from the mean free fluid path at a reference pressure, the operating pressure, and the lateral dimensions.

$$K_n = (MFP*PREF) / (PAMB*DIM)$$

For rectangular channels, DIM is the smallest lateral dimension. For circular channels, DIM is the radius.

For continuum theory to be valid (KEYOPT(1) = 0), the Knudsen number should be less than 0.01. If the Knudsen number is greater than 0.01 (KEYOPT(1) = 1 or 2), the dynamic viscosity is adjusted to account for the slip flow boundary.

The fluid environment is defined by a set of real constants:

For rectangular channels, DIM1 and DIM2 specify the lateral dimensions of the channel. For circular channels, DIM1 specifies the radius of the channel and DIM2 is not used, PAMB specifies the ambient (i.e., surrounding) pressure, PREF specifies the reference pressure for the mean free fluid path, and MFP specifies the mean free fluid path at reference pressure PREF.

For continuum theory (KEYOPT(1) = 1), DIM1, DIM2 (if rectangular channel), and PAMB must be specified.

For high Knudsen numbers (KEYOPT(1) = 1), DIM1, DIM2 (if rectangular channel), PAMB, PREF and MFP must be specified. PREF and MFP are used to adjust the dynamic viscosity.

FLUID138 does not support any loadings. To preserve the pressure drop through the hole, the PRES degree of freedom for the nodes of the FLUID136 elements at the periphery of the hole must be coupled to the PRES degree of freedom for node I of the FLUID138 element representing the hole, and the pressure degree of freedom for node J must be set to the surrounding ambient pressure.

FLUID138 Input Summary

Nodes
I, J

Degrees of Freedom
PRES

Real Constants
DIM1, DIM2, (blank), PAMB, (blank), (blank),

PREF, MFP

Material Properties
VISC - dynamic viscosity

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Continuous flow options

0 --
Continuum theory

1 --
High Knudsen numbers

KEYOPT(3)
Cross section definition

0 --
Circular cross section

1 --
Rectangular cross section

FLUID138 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID129 Element Output Definitions"*

A general description of solution output is given in *Table 1, "FLUID136 Element Output Definitions"*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID138 Element Output Definitions

Name	Definition	O	R
PRES	Pressure change with regard to ambient pressure		Y
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOL	Volume	Y	Y
FLUE	Fluences	Y	Y
LENGTH	Channel Length	Y	Y
AREA	Area	Y	Y
PRES (I, J)	P1 at node I, P2 at node J	Y	Y
FLOW	Flow rate	Y	
VELOCITY	Average velocity	Y	

Table 2, "FLUID138 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID129 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID129 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID138 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
Effective length	NMISC	2
Fluid resistance	NMISC	3
Cross sectional area	NMISC	4

FLUID138 Assumptions and Restrictions

- Knudsen numbers larger than 880 are not supported.
- The gas flow is assumed to be isothermal.
- The pressure change must be small compared to ambient pressure.
- The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**). See *Section 7.8: Squeeze Film Theory* in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on the governing equations.

FLUID138 Product Restrictions

There are no product-specific restrictions for this element.

FLUID139

3-D Slide Film Fluid Element

MP ME <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

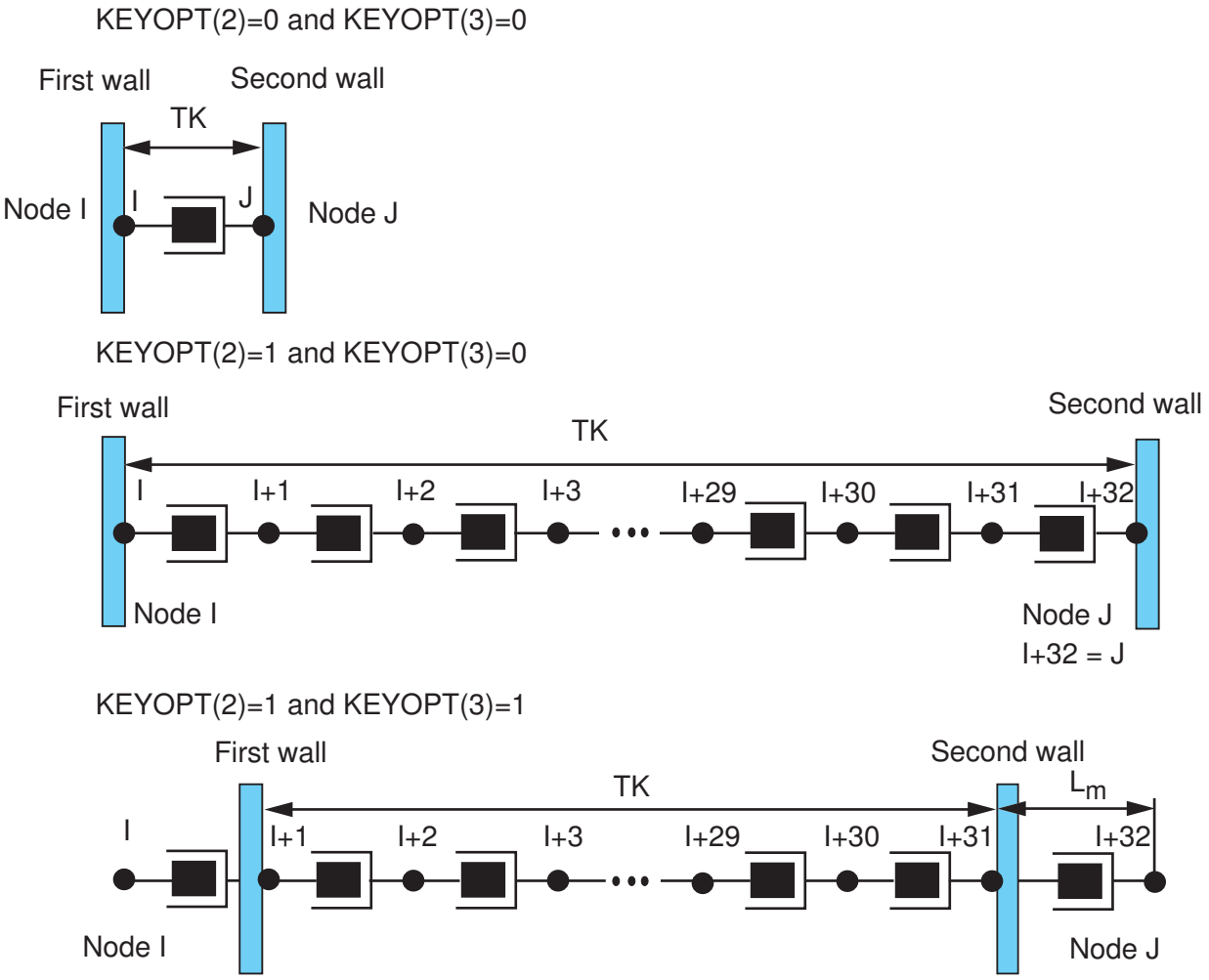
FLUID139 Element Description

FLUID139 is a uniaxial element which models the fluid behavior between a sliding surface and a fixed wall. The viscous flow between surfaces is represented by a series connection of mass-damper elements whereby each node corresponds to a local fluid layer. The element has applications for modeling the fluid damping effects in microsystems such as comb drive fingers, large horizontally moving plates in seismic devices, etc. The element can be used in conjunction with other elements to model complete structural-fluid damping interaction, or stand-alone to add damping effects in a lumped sense to a structure. For low frequency applications, Couette flow assumptions is used. At higher frequencies where inertial effects become important, Stokes flow theory is used. First and second order slip flow models can be activated for systems which operate at high Knudsen numbers. The element is applicable to large deflection cases where the surface area exposed to a fixed wall changes with displacement (such as in comb fingers). See FLUID139 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

FLUID139 Input Data

The element is defined by two nodes. The I node is connected to the first "wall" and the J (or I+32) node is attached to the second "wall". Either wall may be constrained from moving, or both walls may move with respect to one another.

Figure 1 FLUID139 Geometry



The 2-node option (KEYOPT(2) = 0) is recommended for systems which operate at frequencies below the cut-off frequency.

The 32-node option (KEYOPT(2) = 1) is necessary for Stokes flow models where only a small fluid layer at the wall is accelerated due to fluid inertia. For the 32 node option (KEYOPT(2) = 1), the first node is node I, and the 32nd node is node J. The intermediate node numbers (2-31) must be defined, but their location may be arbitrary. The geometric location of node I and J is not important as their separation distance is computed from the real constant gap separation GAP.

The fluid environment is defined by the real constants.

Gap is the local gap separation (i.e., width of fluid domain). AREA is the surface area. DADU is the change in the overlap area with respect to the surface displacement. That is, DADU is the first derivative of AREA with respect to displacement. If the surface area is constant, then DADU is the width of the overlap surface. PAMB is the ambient (i.e., surrounding) pressure. PREF is the reference pressure for which the mean free path of the fluid is defined. MFP is the mean free path of the fluid at PREF.

FLUID139 can be used to model continuous flow or slip flow boundary conditions. If the Knudsen number is less than 0.01, then continuous flow boundary conditions are valid. If the Knudsen number is greater than 0.01, but not near 1, then first order slip flow boundary conditions are valid. If the Knudsen number is near 1, then extended slip flow boundary conditions are valid. KEYOPT(3) is used to specify fluid blow boundary conditions. KEYOPT(3) = 0 specifies continuous flow. KEYOPT(3) = 1 specifies first order slip flow boundary conditions. KEYOPT(3) = 2

specifies extended slip flow boundary conditions. See *Section 16.2.3: Flow Regime Considerations* in the *Fluids Analysis Guide* for a complete discussion of flow regimes and calculation of the Knudsen number.

FLUID139 can be loaded by nodal displacements at the interface nodes using the **D** command or by nodal forces using the **F** command. A combination of FLUID139 and structural elements allows a simultaneous fluid-structure domain simulation.

FLUID139 Input Summary

Nodes
I, J (KEYOPT(2) = 0)

I, J, node 32 (KEYOPT(2) = 1)

Degrees of Freedom
UX, UY, UZ (Depending on KEYOPT(1))

Real Constants
GAP, AREA, DADU, PAMB, (blank), (blank)

PREF, MFP

Material Properties
DENS - density

VISC - dynamic viscosity

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Operating Directions

0,1 --
x-direction (UX DOF)

2 --
y-direction (UY DOF)

3 --
z-direction (UZ DOF)

KEYOPT(2)
Flow model

0 --
2-node element (Couette flow)

1 --
32-node element (Stokes flow)

KEYOPT(3)
Continuous flow options

0 --
Continuum theory

1 --
First order slip flow

2 --
Extended slip flow theory

FLUID139 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "FLUID129 Element Output Definitions"*

A general description of solution output is given in *Table 1, "FLUID136 Element Output Definitions"*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 FLUID139 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOL	Volume	Y	Y
GAP	Gap separation	Y	Y
AREA	Area	Y	Y
PRES (I, J)	P1 at node I, P2 at node J	Y	Y

Table 2, "FLUID138 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "FLUID129 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "FLUID129 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 FLUID139 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
GAP	NMISC	2
AREA	NMISC	3

FLUID139 Assumptions and Restrictions

The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**). See *Section 7.9: Slide Film Theory* in the *Theory Reference for ANSYS and ANSYS Workbench* for more information on the governing equations.

FLUID139 Product Restrictions

There are no product-specific restrictions for this element.

FLUID141

2-D Fluid-Thermal

MP <> <> <> <> <> <> FL <> <> <> PP <>
Product Restrictions

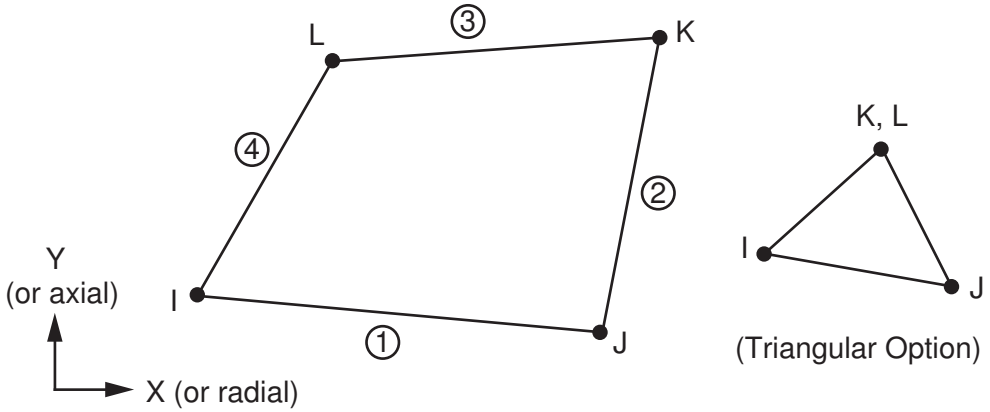
FLUID141 Element Description

You can use FLUID141 to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. Use this FLOTRAN CFD element to solve for flow and temperature distributions within a region, as opposed to elements that model a network of one-dimensional regions hooked together (such as FLUID116). You can also use FLUID141 in a fluid-solid interaction analysis. See FLUID141 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

For the FLOTRAN CFD elements, the velocities are obtained from the conservation of momentum principle, and the pressure is obtained from the conservation of mass principle. (The temperature, if required, is obtained from the law of conservation of energy.) A segregated sequential solver algorithm is used; that is, the matrix system derived from the finite element discretization of the governing equation for each degree of freedom is solved separately. The flow problem is nonlinear and the governing equations are coupled together. The sequential solution of all the governing equations, combined with the update of any temperature- or pressure-dependent properties, constitutes a *global iteration*. The number of global iterations required to achieve a converged solution may vary considerably, depending on the size and stability of the problem. Transport equations are solved for the mass fractions of up to six species.

You may solve the system of equations in a constant angular velocity rotating coordinate system. The degrees of freedom are velocities, pressure, and temperature. Two turbulence quantities, the turbulent kinetic energy and the turbulent kinetic energy dissipation rate, are calculated if you invoke an optional *turbulence* model. For axisymmetric models, you can calculate an optional *swirl* - velocity VZ normal to the plane. You also can specify swirl at the inlet or a boundary (moving wall).

Figure 1 FLUID141 Geometry



FLUID141 Input Data

Figure 1, "FLUID141 Geometry" shows the geometry, node locations, and the coordinate system for this element. The element is defined by three nodes (triangle) or four nodes (quadrilateral) and by isotropic material properties. The coordinate system is selected according to the value of KEYOPT(3), and may be either Cartesian, axisymmetric, or polar.

Section 2.8: Node and Element Loads describes element loads. For a fluid-solid interaction analysis, you can apply a fluid-solid interaction flag using the SF family of commands (**SF**, **SFA**, **SFE**, or **SFL**) and the FSIN surface load label. You must also apply the same interface number to the solid interface where load transfer takes place. See Sequentially Coupled Physics Analysis in the *Coupled-Field Analysis Guide* for more information on the use of the fluid-solid interaction flag.

The *Fluids Analysis Guide* includes a discussion of which ANSYS commands are unavailable or inappropriate for FLUID141.

FLUID141 Fluid Elements

If the material number [**MAT**] of a FLUID141 element is 1, it is assumed to be a fluid element. Its properties - density, viscosity, thermal conductivity and specific heat - are defined with a series of **FLDATA** commands. You can analyze only one fluid, and it must be in a single phase. Thermal conductivity and specific heat are relevant (and necessary) only if the problem is thermal in nature. The properties can be a function of temperature through relationships specified by the **FLDATA7**,PROT command or through a property database (the file `floprp.ans`). In addition, the density may vary with pressure (per the ideal gas law) if the fluid is specified to be air or a gas.

Six turbulence models are available. You can activate turbulence modeling with the **FLDATA1**,SOLU,TURB,T command. The Standard k-ε Model and the Zero Equation Turbulence Model are available along with four extensions of the Standard k-ε Model. See the *Theory Reference for ANSYS and ANSYS Workbench* and the *Fluids Analysis Guide* for more information on the models.

KEYOPT(1) activates *multiple species transport*, which allows you to track the transport of up to six different fluids (species) in the main fluid. KEYOPT(4) allows you to use displacement DOFs to specify motion of boundaries when using the Arbitrary Lagrangian-Eulerian (ALE) formulation.

Real constants, shown in *Table 1, "FLUID141 Real Constants"*, are required only if a *distributed resistance* (*FLUID141 Distributed Resistance*), a *fan model* (*FLUID141 Fan Model*), a wall roughness (*FLUID141 Wall Roughness*), or an ALE formulation is to be included.

FLUID141 Distributed Resistance

A *distributed resistance* provides a convenient way to approximate the effect of porous media (such as a filter) or other such flow domain features without actually modeling the geometry of those features. It is an artificially imposed, unrecoverable loss associated with geometry not explicitly modeled. Any fluid element with a distributed resistance will have a real constant set number [**REAL**] greater than 1 assigned to it.

The resistance to flow, modeled as a distributed resistance, may be due to one or a combination of these factors: a localized head loss (K), a friction factor (f), or a permeability (C). The total pressure gradient is the sum of these three terms, as shown below for the X direction.

$$\left(\frac{\partial p}{\partial x}\right)_{\text{resistance}} = \{-K\rho V_x|V| + \frac{f}{D_h}\rho Vx|V| + C\mu V_x\}$$

where:

ρ = is the density (mass/length^3)
μ = is the viscosity (mass/(length*time))
RE = is the local value of the Reynolds Number (calculated by the program): RE = (ρ V D_h) / μ
f = is a friction coefficient (calculated by the program): f = a RE^{-b}
C = is the FLOTRAN permeability (1/length2). FLOTRAN permeability is the inverse of the intrinsic or physical permeability.

If large gradients exist in the velocity field within a distributed resistance region, you should deactivate the turbulence model by setting ENKE to 0 and ENDS to 1.0 in this region.

Non-Newtonian viscosity models also are available for this element. Currently, ANSYS provides a Power Law model, a Bingham model, and a Carreau model.

In addition, ANSYS provides a user-definable subroutine to compute viscosity. The *Theory Reference for ANSYS and ANSYS Workbench* and the *Fluids Analysis Guide* describe these models and how to use them. The subroutine, called UserVisLaw, is documented in the *Guide to ANSYS User Programmable Features*.

FLUID141 Fan Model

The *fan model* provides a convenient way to approximate the effect of a fan or pump in the flow domain. It is an artificially imposed momentum source that provides momentum source terms associated with a fan or a pump not explicitly modeled.

The pressure rise associated with a fan model is given by the pressure gradient times the flow length through the elements with the fan model real constants. For a one-directional fan model, (real constant TYPE = 4), three coefficients are input. The pressure gradient can be treated as a quadratic function of velocity, as shown below for the X direction.

$$\frac{\partial p}{\partial x}_{fan} = C_1 + C_2 \left| V_x \right| + C_3 V_x^2$$

V is the fluid velocity and C_1, C_2, and C_3 are the coefficients specified as real constants. For an arbitrary direction fan model (real constant TYPE = 5), the three coefficients are the components of the actual coefficients along a coordinate direction. See also the *Fluids Analysis Guide*.

FLUID141 Wall Roughness

The FLOTRAN default condition is smooth walls. For information on applying roughness values, see Flow Boundary Conditions in the *Fluids Analysis Guide*.

FLUID141 Non-Fluid Elements

If the material number [**MAT**] of the element is greater than 1, it is assumed to be a non-fluid element. Only the energy equation is solved in the non-fluid elements. You can define up to 100 different non-fluid materials. To specify density, specific heat, and thermal conductivity for the non-fluid elements, use the **MP** command. Temperature variation of the non-fluid properties is permitted, and you specify it via the **MP** or **MPDATA** commands. Orthotropic variation also is permitted, with the restriction that the spatial variation is always with respect to the global coordinate system. Note that element real constants have no meaning for non-fluid FLUID141 elements.

FLUID141 Input Summary summarizes the element input. *Section 2.1: Element Input* gives a general description of element input. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

FLUID141 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
VX, VY, VZ, PRES, TEMP, ENKE, ENDS

Real Constants
See *Table 1, "FLUID141 Real Constants"*

Material Properties

Non-fluid: KXX, KYY, C, DENS
Fluid: Density, viscosity, thermal conductivity, specific heat (use **FLDATA** commands) or **MPTEMP** and **MPDATA**.

Surface Loads
HFLUX, CONV, RAD, RDSF, FSIN

Body Loads
HGEN, FORC

Special Features

Nonlinear
Six turbulence models
Incompressible or compressible algorithm
Transient or steady state algorithm
Rotating or stationary coordinate system
Algebraic solvers particular to FLOTRAN
Optional distributed resistance and fan models
Multiple species transport

KEYOPT(1)
Number of species:

0 --
Species transport is not activated.

2 - 6 --
Number of species transport equations to be solved.

KEYOPT(3)
Element coordinate system:

0 --
Cartesian coordinates (default)

1 --
Axisymmetric about Y-axis

2 --
Axisymmetric about X-axis

3 --
Polar Coordinates

KEYOPT(4)
Support mesh displacement DOFs:

0 --
Do not include displacement DOFs.

1 --
Include displacement DOFs (UX and UY).

Table 1 FLUID141 Real Constants

No.	Name	Definition and Type no.	Units
1	TYPE	Type of distributed resistance or fan model:	
		1 = Distributed resistance: isotropic	-
		2 = Distributed resistance: one-directional	-
		3 = Distributed resistance: direction-dependent	-
		4 = Fan model: aligned with a coordinate axis	-
		5 = Fan model: arbitrary direction	-
2	(Blank)	1, 2, 3 - Not used	-
	DIR	4 - Fan orientation: 1 = X, 2 = Y, 3 = Z	-
	(Blank)	5 - Not Used	-
3	K	1, 2 - Dimensionless head loss / length	1/L
	K_x	3 - Head loss in X direction	1/L
	C_1	4 - Constant term	M/L^2t^2
	C_{1x}	5 - Vector component of C_1 in X direction	M/L^2t^2
4	C	1, 2 - Permeability	$1/L^2$
	C_x	3 - Permeability in X direction	$1/L^2$
	C_2	4 - Linear coefficient	M/L^3t
	C_{2x}	5 - Vector component of C_2 in X direction	M/L^3t
5	D_h	1, 2 - Hydraulic diameter	L
	D_{hx}	3 - Hydraulic diameter in X direction	L
	C_3	4 - Quadratic coefficient	M/L^4
	C_{3x}	5 - Vector component of C_3 in X direction	M/L^4
6	a	1, 2 - Coefficient of Reynolds number, used in friction factor calculations	-
	a_x	3 - Coefficient a in X direction	-
	(Blank)	4, 5 - Not Used	-
7	b	1, 2 - Exponent of Reynolds number, used in friction factor calculations	-
	b_x	3 - Exponent b in X direction	-
	(Blank)	4, 5 - Not Used	-
8	(Blank)	1 - Not Used	-
	FLDIR	2 - Flow direction: 1 = X, 2 = Y, 3 = Z	-
	K_y	3 - Head loss in Y direction	1/L
	(Blank)	4 - Not Used	-
	C_{1y}	5 - Vector component of C_1 in Y direction	M/L^2t^2
9	(Blank)	1, 2 - Not Used	-
	C_y	3 - Permeability in Y direction	$1/L^2$
	(Blank)	4 - Not Used	-
	C_{2y}	5 - Vector component of C_2 in Y direction	M/L^3t
10	(Blank)	1, 2 - Not Used	-
	D_{hy}	3 - Hydraulic diameter in Y direction	L
	(Blank)	4 - Not Used	-
	C_{3y}	5 - Vector component of C_3 in Y direction	M/L^4

No.	Name	Definition and Type no.	Units
11	(Blank)	1, 2 - Not Used	-
	a_y	3 - Coefficient of Reynolds number in Y direction	-
	(Blank)	4, 5 - Not Used	-
12	(Blank)	1, 2 - Not Used	-
	b_y	3 - Exponent of Reynolds number in Y direction	-
	(Blank)	4, 5 - Not Used	-
13	(Blank)	1, 2 - Not Used	-
	K_z	3 - Head loss in Z (swirl) direction	1/L
	(Blank)	4 - Not Used	-
	C_{1z}	5 - Vector component of C_1 in Z (swirl) direction	M/L^2t^2
14	(Blank)	1, 2 - Not Used	-
	C_z	3 - Permeability in Z (swirl) direction	$1/L^2$
	(Blank)	4 - Not Used	-
	C_{2z}	5 - Vector component of C_2 in Z (swirl) direction	M/L^3t
15	(Blank)	1, 2 - Not Used	-
	D_{hz}	3 - Hydraulic diameter in Z (swirl) direction	L
	(Blank)	4 - Not Used	-
	C_{3z}	5 - Vector component of C_3 in Z (swirl) direction	M/L^4
16	(Blank)	1, 2 - Not Used	-
	a_z	3 - Coefficient of Reynolds number in Z (swirl) direction	-
	(Blank)	4, 5 - Not Used	-
17	(Blank)	1, 2 - Not Used	-
	b_z	3 - Exponent of Reynolds number in Z (swirl) direction	-
	(Blank)	4, 5 - Not Used	-
18	BDTOL	Element birth/death tolerance	L
19	MMFAC	Mesh morphing multiplier	-
20	K_s	Local uniform wall roughness	L
21	CK_s	An empirical dimensionless factor between 0.5 and 1.0 that specifies the degree of nonuniformity of the surface.	-

FLUID141 Output Data

The solution output associated with the element takes the form of nodal quantities. Additional intermediate properties and derived quantities supplement the degrees of freedom. See the *Basic Analysis Guide* for ways to view results.

Table 2, "FLUID141 Element Output Definitions" describes quantities that are output on a nodal basis. Some quantities are not output if the relevant options are not activated. Once an option is used, the relevant DOF quantities are always stored. For example, if a temperature field has been obtained and upon restart the energy equation is no longer to be solved, the temperatures are stored anyway. You control the storage of derived properties such as effective viscosity by issuing the **FLDATA5**,OUTP command.

The `Jobname.PFL` file provides additional output. This file contains periodic tabulations of the maximum, minimum, and average values of the velocities, pressure, temperature, turbulence quantities, and properties. The file also records the convergence monitoring parameters calculated at every global iteration. The `Jobname.PFL` file also tabulates the mass flow at all the inlets and outlets and the heat transfer information at all the boundaries.

A wall results file (Jobname.RSW) contains information associated with the boundary faces of wall elements. Average pressure, temperature, shear stress, Y-plus values and wall heat fluxes are stored, along with vectors denoting the normal direction from the surface (Normal Vector) and the direction of the velocity immediately adjacent to the wall (Tangent Vector).

An optional residual file (Jobname.RDF) shows how well the current solution satisfies the implied matrix equations for each DOF.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The R column indicates the availability of the items in the results file.

A Y in the R column indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 FLUID141 Element Output Definitions

Name	Definition	R
UX	Displacement in the X direction (Cartesian coordinates); Displacement along axis of symmetry (Axisymmetric about X); Displacement in the radial direction (Axisymmetric about Y)	*10*
UY	Displacement in the Y direction (Cartesian coordinates); Displacement along radial direction (Axisymmetric about X); Displacement along the axis of symmetry (Axisymmetric about Y)	*10*
VX:	Velocity in the X direction (Cartesian coordinates); Velocity in the radial direction (Polar coordinates); Velocity along axis of symmetry (Axisymmetric about X); Velocity in the radial direction (Axisymmetric about Y)	Y
VY:	Velocity in the Y direction (Cartesian coordinates); Velocity in the tangential direction (Polar coordinates); Velocity in the radial direction (Axisymmetric about X); Velocity along the axis of symmetry (Axisymmetric about Y)	Y
VZ:	Velocity in the swirl direction (Axisymmetric problems)	*8*
PRES:	Relative Pressure	Y
ENKE:	Turbulent kinetic energy	*2*
ENDS:	Turbulence dissipation rate	*2*
TEMP:	Temperature	*1*
DENS:	Nodal fluid density	*8*
VISC:	Nodal fluid viscosity	*8*
COND:	Nodal fluid thermal conductivity	*8*
SPHT:	Nodal fluid specific heat	*8*
EVIS:	Effective viscosity (includes effects of turbulence)	*8*
ECON:	Effective thermal conductivity (includes the effects of turbulence)	*2*
CMUV:	Turbulent viscosity coefficient	*2*
TTOT:	Stagnation (Total) Temperature (Only relevant to compressible analyses)	*7*
HFLU:	Heat Flux at external surface nodes (per unit area)	*1*
HFLM:	Heat Transfer (film) coefficient at external surface nodes	*1*
RDFL:	Radiation Heat Flux	*1*
STRM:	Stream Function (2-D)	Y
MACH:	Mach Number (must be requested if incompressible)	*6*

Name	Definition	R
PTOT:	Stagnation (Total) Pressure	Y
PCOE:	Pressure Coefficient	*3*
YPLU:	Y+ a turbulent law of the wall parameter	*3*
TAUW:	Shear Stress at the wall	*3*
SP0*N*:	Mass fraction of species *N*, where *N* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*N*.	*4*
LMD*N*:	Laminar mass diffusion coefficient for species *N*, where *N* = 1 to 6. (Not relevant unless species defined.)	*3*
EMD*N*:	Effective mass diffusion coefficient for species *N*, where *N* = 1 to 6. (Not relevant unless species defined.)	*2*

1. Available if thermal is on.
2. Available if turbulence is on.
3. Must be requested.
4. Available if species defined.
5. Available if property is variable.
6. Available if compressible.
7. Available if compressible and thermal.
8. Available if swirl is turned on.
9. For solid material elements in FLOTRAN, when nodes are connected only to solid nodes, the column for the density (DENS) label within the `Jobname.RFL` results file, stores the product of the solid material's density and its specific heat.
10. Available if KEYOPT(4) = 1.

FLUID141 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- You must define the connectivity of an element with the nodes in counterclockwise order.
- The element must lie in the X-Y plane.
- When triangles are formed by duplicating the third node, the FLOTRAN element will ignore the duplicate node and treat nodes I, J, and K.
- Only linear elements are supported.
- You cannot use FLUID141 with any other ANSYS elements.
- Not all ANSYS commands are relevant to the use of FLUID141. The *Fluids Analysis Guide* documents these command usage restrictions.
- FLOTRAN CFD analyses are highly nonlinear.
- In some cases, convergence is difficult to achieve and requires the use of stability and relaxation parameters.
- Highly turbulent cases may benefit from preconditioning (the initialization of the flow field with a laminar analysis), particularly if a coarse finite element mesh is being used.
- You must determine if use of the turbulence and/or compressible option is warranted. The turbulence option requires a fine mesh near the walls and a fine mesh is recommended near any regions where shock

waves occur. If the larger gradients occur in regions with the coarsest mesh, rerun the problems with adjusted meshes.

- Surface-to-surface radiation (RDSF) is not supported for compressible flow thermal analysis and R-θ and R-θ-Z coordinate systems.
- The FLOTRAN element must be in counterclockwise order for a 2-D FSI analysis (for *Figure 1, "FLUID141 Geometry"*, I, J, K, L order) and it must be in positive volume order for a 3-D FSI analysis (for *Figure 1, "FLUID142 Geometry"*, I, J, K, L, M, N, O order). If the element order is not proper, you will need to recreate the mesh to reverse it.

The following assumptions have been made in the formulation:

- The nodal coordinate system and the global coordinate system must remain the same.
- The problem domain and the finite element mesh may not change during an analysis.
- The fluid is a single phase fluid.
- Non-fluid thermal conductivities can vary with temperature. Orthotropic variation of non-fluid thermal conductivity also is supported. For more information, see **MP**, **MPDATA**, and related commands in the *Commands Reference*.
- Free surfaces are not permitted.
- The equation of state of gases is the ideal gas law. This is the case regardless of whether the incompressible or compressible algorithm is invoked. The ideal gas law is not valid at Mach numbers above 5.
- In the incompressible option, work done on the fluid by pressure forces, viscous dissipation, and kinetic energy terms are neglected in the energy equation. The incompressible energy equation is a thermal transport equation.
- In the compressible adiabatic case, the stagnation (total) temperature is assumed constant and the static temperature is calculated from it by subtracting a kinetic energy term.
- Load case operations are not permitted with the FLOTRAN elements.

FLUID141 Product Restrictions

There are no product-specific restrictions for this element.

FLUID142

3-D Fluid-Thermal

MP <> <> <> <> <> <> FL <> <> <> PP <>
Product Restrictions

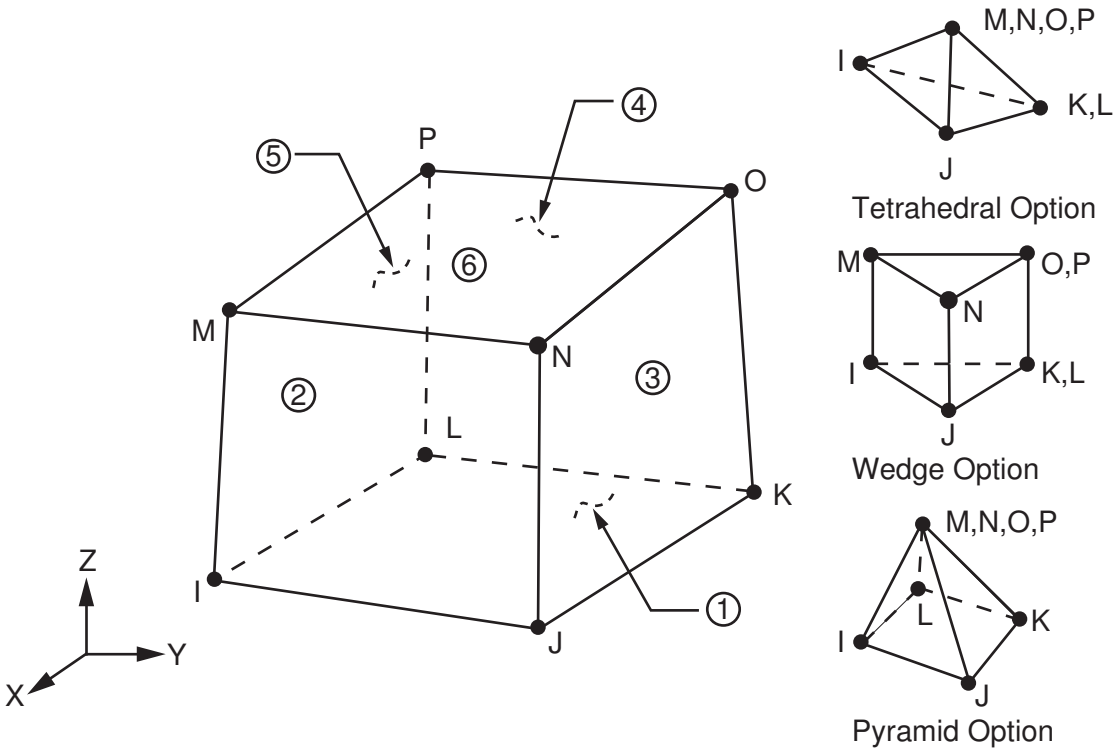
FLUID142 Element Description

You can use FLUID142 to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. Use this FLOTRAN CFD element to solve for flow and temperature distributions within a region, as opposed to elements that model a network of one-dimensional regions hooked together (such as FLUID116). You can also use FLUID142 in a fluid-solid interaction analysis. See FLUID142 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

For the FLOTRAN CFD elements, the velocities are obtained from the conservation of momentum principle, and the pressure is obtained from the conservation of mass principle. (The temperature, if required, is obtained from the law of conservation of energy.) A segregated sequential solver algorithm is used; that is, the matrix system derived from the finite element discretization of the governing equation for each degree of freedom is solved separately. The flow problem is nonlinear and the governing equations are coupled together. The sequential solution of all the governing equations, combined with the update of any temperature- or pressure-dependent properties, constitutes a *global iteration*. The number of global iterations required to achieve a converged solution may vary considerably, depending on the size and stability of the problem. Transport equations are solved for the mass fractions of up to six species.

You can solve the system of equations in a constant angular velocity rotating coordinate system. The degrees of freedom are velocities, pressure, and temperature. Two turbulence quantities, the turbulent kinetic energy and the turbulent kinetic energy dissipation rate, are calculated if you invoke an optional *turbulence* model.

Figure 1 FLUID142 Geometry



FLUID142 Input Data

Figure 1, "FLUID142 Geometry" shows the geometry, node locations, and the coordinate system for this element. The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "FLUID142 Geometry"*. The coordinate system, selected according to the value of KEYOPT(3), may be either Cartesian or cylindrical.

Section 2.8: Node and Element Loads describes element loads. For a fluid-solid interaction analysis, you can apply a fluid-solid interaction flag using the SF family of commands (**SF**, **SFA**, **SFE**, or **SFL**) and the FSIN surface load label. You must also apply the same interface number to the solid interface where load transfer takes place. See Sequentially Coupled Physics Analysis in the *Coupled-Field Analysis Guide* for more information on the use of the fluid-solid interaction flag.

The *Fluids Analysis Guide* includes a discussion of which ANSYS commands are unavailable or inappropriate for FLUID142.

FLUID142 Fluid Elements

If the material number [**MAT**] of a FLUID142 element is 1, it is assumed to be a fluid element. You define its properties - density, viscosity, thermal conductivity and specific heat - with a series of **FLDATA** commands. Only one fluid can be analyzed, and it must be in a single phase. Thermal conductivity and specific heat are relevant (and necessary) only if the problem is thermal in nature. The properties can be a function of temperature through relationships specified by the **FLDATA7**,PROT command or through a property database (the file `floprp.ans`). In addition, the density may vary with pressure (per the ideal gas law) if the fluid is specified to be air or a gas.

Six turbulence models are available. You can activate turbulence modeling with the **FLDATA1**,SOLU,TURB,T command. The Standard k-ε Model and the Zero Equation Turbulence Model are available along with four extensions of the Standard k-ε Model. See the *Theory Reference for ANSYS and ANSYS Workbench* and the *Fluids Analysis Guide* for more information on the models.

KEYOPT(1) activates *multiple species transport*, which allows you to track the transport of up to six different fluids (species) in the main fluid. KEYOPT(4) allows you to use displacement DOFs to specify motion of boundaries when using the Arbitrary Lagrangian-Eulerian (ALE) formulation.

Real constants, shown in *Table 1, "FLUID142 Real Constants"*, are required only if a *distributed resistance* (*FLUID142 Distributed Resistance*), a *fan model* (*FLUID142 Fan Model*), a wall roughness (*FLUID142 Wall Roughness*), or an ALE formulation is to be included.

FLUID142 Distributed Resistance

A *distributed resistance* is a convenient way to approximate the effect of porous media (such as a filter) or other such flow domain features without actually modeling the geometry of those features. It is an artificially imposed, unrecoverable loss associated with geometry not explicitly modeled. Any fluid element with a distributed resistance will have a real constant set number greater than 1.

The resistance to flow, modeled as a distributed resistance, may be due to one or a combination of these factors: a localized head loss (K), a friction factor (f), or a permeability (C). The total pressure gradient is the sum of these three terms, as shown below for the X direction.

$$\frac{\partial p}{\partial x}_{\text{resistance}} = \{-K\rho V_x \left| V \right| + \frac{f}{D_h}\rho Vx \left| V \right| + C\mu V_x\}$$

where:

ρ = is the density (mass/length^3)
μ = is the viscosity (mass/(length*time))
RE = is the local value of the Reynolds Number (calculated by the program): RE = (ρ V D_h) / μ
f = is a friction coefficient (calculated by the program): f = a RE^{-b}
C = is the FLOTRAN permeability (1/length2). FLOTRAN permeability is the inverse of the intrinsic or physical permeability.

The ANSYS program also offers non-Newtonian viscosity models for this element. Currently, Power Law, Bingham, and Carreau models are available.

In addition, ANSYS provides a user-defined subroutine for computing viscosity. The *Theory Reference for ANSYS and ANSYS Workbench* and the *Fluids Analysis Guide* describes these models and how to use them. The *Guide to ANSYS User Programmable Features* describes how to use the user-defined subroutine, called UserVisLaw.

If large velocity gradients exist in the velocity field within a distributed resistance region, deactivate the turbulence model by setting the ENKE DOF to 0 and the ENDS DOF to 1 in this region.

FLUID142 Fan Model

The *fan model* provides a convenient way to approximate the effect of a fan or pump in the flow domain. It is an artificially imposed pressure source that provides momentum source terms associated with a fan or a pump not explicitly modeled.

The pressure rise associated with a fan model is given by the pressure gradient times the flow length through the elements with the fan model real constants. The pressure gradient can be treated as a quadratic function of velocity, as shown below for the X direction:

$$\frac{\partial p}{\partial x}_{fan} = C_1 + C_2 \left| V_x \right| + C_3 V_x^2$$

V is the fluid velocity and C_1, C_2, and C_3 are the coefficients specified as real constants. For an arbitrary direction fan model (real constant TYPE = 5), the three coefficients are the components of the actual coefficients along a coordinate direction. See also the *Fluids Analysis Guide*.

FLUID142 Wall Roughness

The FLOTRAN default condition is smooth walls. For information on applying roughness values, see Flow Boundary Conditions in the *Fluids Analysis Guide*.

FLUID142 Non-Fluid Elements

If the material number [**MAT**] of the element is greater than 1, it is assumed to be a non-fluid element. Only the energy equation is solved in the non-fluid elements. You can define up to 100 different non-fluid materials. To specify density, specific heat, and thermal conductivity for the non-fluid elements, issue the **MP** command. Temperature variation of the non-fluid properties is permitted, and you specify it using **MP** or **MPDATA**. Orthotropic variation also is allowed, with the restriction that the spatial variation is always with respect to the global coordinate system. Note that element real constants have no meaning for non-fluid FLUID142 elements.

FLUID142 Input Summary summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

FLUID142 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
VX, VY, VZ, PRES, TEMP, ENKE, ENDS

Real Constants
See *Table 1, "FLUID142 Real Constants"*

Material Properties

Non-fluid: KXX, KYY, KZZ, C, DENS
Fluid: Density, viscosity, thermal conductivity, specific heat (use **FLDATA** commands)

Surface Loads
HFLU, CONV, RAD, RDSF, FSIN

Body Loads
HGEN, FORC

Special Features

Nonlinear
Six turbulence models
Incompressible or compressible algorithm
Transient or steady state algorithm
Rotating or stationary coordinate system
Algebraic solvers particular to FLOTRAN
Optional distributed resistance and fan models
Multiple species transport

KEYOPT(1)
Number of species:

0 --
Species transport is not activated.

2 - 6 --
Number of species transport equations to be solved.

KEYOPT(3)
Element coordinate system:

0 --
Cartesian coordinates (default)

3 --
Cylindrical coordinates

KEYOPT(4)
Support mesh displacement DOFs:

0 --
Do not include displacement DOFs.

1 --
Include displacement DOFs (UX, UY, and UZ).

Table 1 FLUID142 Real Constants

No.	Name	Definition	Units
1	TYPE	Type of distributed resistance or fan model:	
		1 = Distributed resistance: isotropic	-
		2 = Distributed resistance: one-directional	-
		3 = Distributed resistance: direction-dependent	-
		4 = Fan model: aligned with a coordinate axis	-
		5 = Fan model: arbitrary direction	-
2	(Blank)	1, 2, 3 - Not used	-
	DIR	4 - Fan orientation: 1 = X, 2 = Y, 3 = Z	-
	(Blank)	5 - Not Used	-
3	K	1, 2 - Dimensionless head loss / length	1/L
	K_x	3 - Head loss in X direction	1/L
	C_1	4 - Constant term	M/L^2t^2
	C_{1x}	5 - Vector component of C_1 in X direction	M/L^2t^2
4	C	1, 2 - Permeability	$1/L^2$
	C_x	3 - Permeability in X direction	$1/L^2$
	C_2	4 - Linear coefficient	M/L^3t
	C_{2x}	5 - Vector component of C_2 in X direction	M/L^3t
5	D_h	1, 2 - Hydraulic diameter	L
	D_{hx}	3 - Hydraulic diameter in X direction	L
	C_3	4 - Quadratic coefficient	M/L^4
	C_{3x}	5 - Vector component of C_3 in X direction	M/L^4
6	a	1, 2 - Coefficient of Reynolds number, used in friction factor calculations	-
	a_x	3 - Coefficient a in X direction	-
	(Blank)	4, 5 - Not Used	-
7	b	1, 2 - Exponent of Reynolds number, used in friction factor calculations	-
	b_x	3 - Exponent b in X direction	-
	(Blank)	4, 5 - Not Used	-
8	(Blank)	1 - Not Used	-
	FLDIR	2 - Flow direction: 1 = X, 2 = Y, 3 = Z	-
	K_y	3 - Head loss in Y direction	1/L
	(Blank)	4 - Not Used	-
	C_{1y}	5 - Vector component of C_1 in Y direction	M/L^2t^2
9	(Blank)	1, 2 - Not Used	-
	C_y	3 - Permeability in Y direction	$1/L^2$
	(Blank)	4 - Not Used	-
	C_{2y}	5 - Vector component of C_2 in Y direction	M/L^3t
10	(Blank)	1, 2 - Not Used	-
	D_{hy}	3 - Hydraulic diameter in Y direction	L
	(Blank)	4 - Not Used	-
	C_{3y}	5 - Vector component of C_3 in Y direction	M/L^4

No.	Name	Definition	Units
11	(Blank)	1, 2 - Not Used	-
	a_y	3 - Coefficient of Reynolds number in Y direction	-
	(Blank)	4, 5 - Not Used	-
12	(Blank)	1, 2 - Not Used	-
	b_y	3 - Exponent of Reynolds number in Y direction	-
	(Blank)	4, 5 - Not Used	-
13	(Blank)	1, 2 - Not Used	-
	K_z	3 - Head loss in Z (swirl) direction	1/L
	(Blank)	4 - Not Used	-
	C_{1z}	5 - Vector component of C_1 in Z (swirl) direction	M/L^2t^2
14	(Blank)	1, 2 - Not Used	-
	C_z	3 - Permeability in Z (swirl) direction	$1/L^2$
	(Blank)	4 - Not Used	-
	C_{2z}	5 - Vector component of C_2 in Z (swirl) direction	M/L^3t
15	(Blank)	1, 2 - Not Used	-
	D_{hz}	3 - Hydraulic diameter in Z (swirl) direction	L
	(Blank)	4 - Not Used	-
	C_{3z}	5 - Vector component of C_3 in Z (swirl) direction	M/L^4
16	(Blank)	1, 2 - Not Used	-
	a_z	3 - Coefficient of Reynolds number in Z (swirl) direction	-
	(Blank)	4, 5 - Not Used	-
17	(Blank)	1, 2 - Not Used	-
	b_z	3 - Exponent of Reynolds number in Z (swirl) direction	-
	(Blank)	4, 5 - Not Used	-
18	BDTOL	Element birth/death tolerance	L
19	MMFAC	Mesh morphing multiplier	-
20	K_s	Local uniform wall roughness	L
21	CK_s	An empirical dimensionless factor between 0.5 and 1.0 that specifies the degree of nonuniformity of the surface.	-

FLUID142 Output Data

The solution output associated with the element takes the form of nodal quantities. Additional intermediate properties and derived quantities supplement the degrees of freedom. See the *Basic Analysis Guide* for ways to view results.

Table 1, "FLUID142 Real Constants" describes quantities that are output on a nodal basis. Some quantities are not output if the relevant options are not activated. Once an option is used, the relevant DOF quantities are always stored. For example, if a temperature field has been obtained and upon restart the energy equation is no longer to be solved, the temperatures are stored anyway. You control the storage of derived properties such as effective viscosity by issuing the **FLDATA5**,OUTP command.

The `Jobname.PFL` file provides additional output. This file contains periodic tabulations of the maximum, minimum, and average values of the velocities, pressure, temperature, turbulence quantities, and properties. The file also records the convergence monitoring parameters calculated at every global iteration. The `Jobname.PFL` file also tabulates the mass flow at all the inlets and outlets and the heat transfer information at all the boundaries.

A wall results file (Jobname.RSW) contains information associated with the boundary faces of wall elements. Average pressure, temperature, shear stress, Y-plus values and wall heat fluxes are stored, along with vectors denoting the normal direction from the surface (Normal Vector) and the direction of the velocity immediately adjacent to the wall (Tangent Vector).

An optional residual file (Jobname.RDF) shows how well the current solution satisfies the implied matrix equations for each DOF.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The R column indicates the availability of the items in the results file.

A Y in the R column indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 FLUID142 Element Output Definitions

Name	Definition	R
UX	Displacement in the X direction (Cartesian coordinates)	*9*
UY	Displacement in the Y direction (Cartesian coordinates)	*9*
UZ	Displacement in the Z direction (Cartesian coordinates)	*9*
VX:	Velocity in the X direction (Cartesian coordinates); Velocity in the radial direction (Cylindrical coordinates)	Y
VY:	Velocity in the Y direction (Cartesian coordinates); Velocity in the tangential direction (Cylindrical coordinates)	Y
VZ:	Velocity in the Z direction (Cartesian coordinates); Velocity in the axial direction (Cylindrical coordinates)	Y
PRES:	Relative Pressure	Y
ENKE:	Turbulent kinetic energy	*2*
ENDS:	Turbulence dissipation rate	*2*
TEMP:	Temperature	*1*
DENS:	Nodal fluid density	*8*
VISC:	Nodal fluid viscosity	*8*
COND:	Nodal fluid thermal conductivity	*8*
SPHT:	Nodal fluid specific heat	*8*
EVIS:	Effective viscosity (includes effects of turbulence)	*8*
ECON:	Effective thermal conductivity (includes the effects of turbulence)	*2*
CMUV:	Turbulent viscosity coefficient	*2*
TTOT:	Stagnation (Total) Temperature (Only relevant to compressible analyses)	*7*
HFLU:	Heat Flux at external surfaces nodes (per unit area)	*1*
HFLM:	Heat Transfer (film) coefficient at external surface nodes	*1*
MACH:	Mach Number (must be requested if incompressible)	*6*
PTOT:	Stagnation (Total) Pressure	Y
PCOE:	Pressure Coefficient	*3*
YPLU:	Y+ a turbulent law of the wall parameter	*3*
TAUW:	Shear Stress at the wall	*3*
SP0*N*:	Mass fraction of species *N*, where *N* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*N*.	*4*

Name	Definition	R
LMD*N*:	Laminar mass diffusion coefficient for species *N*, where *N* = 1 to 6. (Only relevant if species defined.)	*3*
EMD*N*:	Effective mass diffusion coefficient for species *N*, where *N* = 1 to 6. (Only relevant if species defined.)	*2*

1. Available if thermal is on.
2. Available if turbulence is on.
3. Must be requested.
4. Available if species defined.
5. Available if compressible.
6. Available if compressible and thermal.
7. For solid material elements in FLOTRAN, when nodes are connected only to solid nodes, the column for density (DENS) in the `Jobname.RFL` results file actually stores the product of the solid material's density and its specific heat.
8. Available if property is variable.
9. Available if KEYOPT(4) = 1.

FLUID142 Assumptions and Restrictions

- The element must not have a negative or a zero volume.
- You must define the connectivity of an element such that the normal defined by the right hand rule associated with the first four nodes (hexahedral elements) or three nodes (tetrahedral elements) must point into the element.
- When a tetrahedron is formed by specifying duplicate nodes, the FLOTRAN element will ignore the duplicate nodes and base the geometry on nodes I, J, K, and M.
- Only linear elements are supported.
- You cannot use FLUID142 with any other ANSYS elements.
- Not all ANSYS commands are relevant to the use of FLUID142. See the *Fluids Analysis Guide* for a description of the command restrictions.
- FLOTRAN CFD analyses are highly nonlinear.
- In some cases, convergence is difficult to achieve and requires the use of stability and relaxation parameters.
- Highly turbulent cases may benefit from preconditioning (the initialization of the flow field with a laminar analysis), particularly if a coarse finite element mesh is being used.
- You must determine if use of the turbulence and/or compressible option is warranted. The turbulence option requires a fine mesh near the walls and a fine mesh is recommended near any regions where shock waves occur. If the larger gradients occur in regions with the coarsest mesh, rerun the problems with adjusted meshes.
- For a flow analysis, especially turbulent, you should not use pyramid elements near the walls because it may lead to inaccuracies in the solution.
- Surface-to-surface radiation (RDSF) is not supported for compressible flow thermal analysis and R-θ and R-θ-Z coordinate systems.
- The FLOTRAN element must be in counterclockwise order for a 2-D FSI analysis (for *Figure 1, "FLUID141 Geometry"*, I, J, K, L order) and it must be in positive volume order for a 3-D FSI analysis (for *Figure 1, "FLU-*

ID142 Geometry", I, J, K, L, M, N, O order). If the element order is not proper, you will need to recreate the mesh to reverse it.

The following assumptions have been made in the formulation:

- The nodal coordinate system and the global coordinate system must remain the same.
- The problem domain and the finite element mesh may not change during an analysis.
- The fluid is a single phase fluid.
- Non-fluid thermal conductivities can vary with temperature. Orthotropic variation of non-fluid thermal conductivity also is supported. For more information, see the descriptions of **MP**, **MPDATA**, and related commands.
- Free surfaces are not permitted.
- The equation of state of gases is the ideal gas law. This is the case regardless of whether the incompressible or compressible algorithm is invoked. The ideal gas law is not valid at Mach numbers above 5.
- In the incompressible option, work done on the fluid by pressure forces, viscous dissipation, and kinetic energy terms are neglected in the energy equation. The incompressible energy equation is a thermal transport equation.
- In the compressible adiabatic case, the stagnation (total) temperature is assumed constant and the static temperature is calculated from it by subtracting a kinetic energy term.
- Load case operations are not permitted with the FLOTRAN elements.

FLUID142 Product Restrictions

There are no product-specific restrictions for this element.

ROM144

Reduced Order Electrostatic-Structural

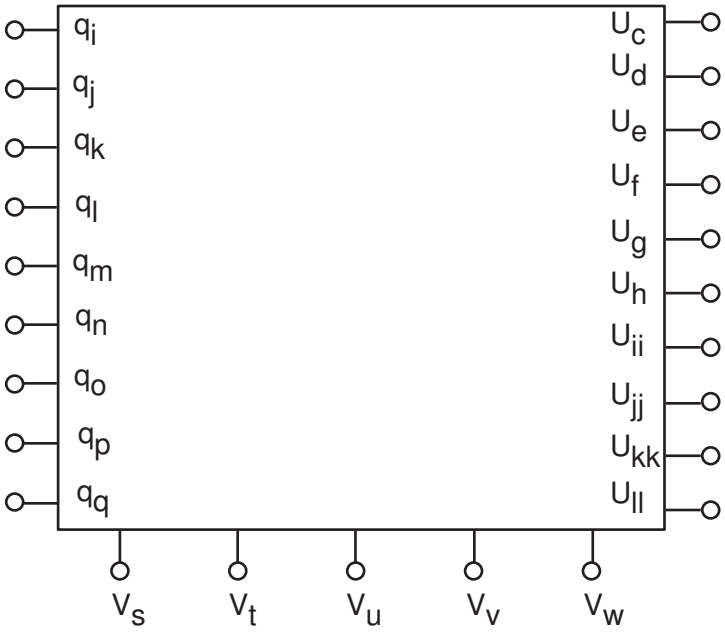
MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

ROM144 Element Description

ROM144 represents a 2-D or 3-D reduced order model of a coupled electrostatic-structural system. The element fully couples the electromechanical domains and represents a reduced order model suitable for use in finite element analysis as well as electromechanical circuit simulations. The element has ten modal degrees of freedom relating modal forces and modal displacements (EMF), ten voltage degrees of freedom relating electrical current and potential (VOLT) and, optionally, 10 master nodes relating nodal forces to nodal displacements (UX). Only nine of the 10 modal degrees of freedom and five of the 10 voltage degrees of freedom are actually used. The element is suitable for simulating the electromechanical response of micro-electromechanical devices (MEMS) such as clamped beams, micromirror actuators, and RF switches.

The element is derived from a series of uncoupled structural and electrostatic domain simulations using the electrostatic elements (such as PLANE121, SOLID122, SOLID123, and INFIN111) and structural elements (such as PLANE42, SOLID45, PLANE82, SOLID95, SHELL63, SHELL93) which are compatible with electrostatic elements. The ROM144 element represents a complicated flexible structure whose nodes move mainly in one direction either X, Y or Z referred to the global Cartesian axes. For instance, torsional systems with angles less than ten degree or flexible bending of cantilevers or membranes obey those restrictions (pressure sensors, cantilever for AF microscopy, RF filter). Geometrical nonlinearities caused by stress stiffening or initial prestress are considered as well as multiple conductor systems. See ROM144 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 ROM144 Schematic



ROM144 Input Data

The element is defined by 20 (KEYOPT(1) = 0) or 30 nodes (KEYOPT(1) = 1). A reduced order model file `filename.rom` and the appropriate polynomial coefficients for the strain energy and capacitance functions stored in `jobname_ijk.pcs` must be available in the working directory. Furthermore, the model database `file-`

name . db and the reduced solution file (.rdsp) generated by the Use Pass are required to perform an Expansion Pass.

Real constant number 1 (R1) is the element identification number (ID). It is automatically created by the circuit builder (see Using the Circuit Builder in the *Low-Frequency Electromagnetic Analysis Guide*), and is not required input for analysis purposes. The element supports nodal forces F and displacements D applied at ROM master nodes (21 to 30). The UX degree of freedom must be chosen independent from the physical direction of the original master node. Electrode current and voltage can be applied only to the first five active voltage nodes (11–15). Modal displacements may be set by the EMF degree of freedom using the **D** command. Element loads defined in the Generation Pass may be scaled and superimposed by the **RMLVSCALE** command. ROM144 can be attached to other finite elements such as COMBIN14 and COMBIN40 at the master DOF. The "reaction force" for the modal displacement degree of freedom (EMF) is a modal force, labeled CURT, and should be used when defining the solution convergence criteria (**CNVTOL** command). The "reaction force" for the electric potential degree of freedom (VOLT) is current, labeled AMPS. The element is compatible with the electric circuit elements CIRCU124 and CIRCU125 and the electromechanical transducer element TRANS126.

Modal damping ratios may be altered by the **RMMRANGE** command. Save the ROM database before using the changed data in the Use Pass.

A summary of the element input is given in *ROM144 Input Summary*.

ROM144 Input Summary

Nodes

20 nodes if KEYOPT(1) = 0:

I, J, K, L, M, N, O, P, Q, Blank, S, T, U, V, W, Blank, Blank, Blank, Blank, Blank

30 nodes if KEYOPT(1) = 1:

I, J, K, L, M, N, O, P, Q, Blank, S, T, U, V, W, Blank, Blank, Blank, Blank, Blank, C, D, E, F, G, H, II, JJ, KK, LL

Degrees of Freedom

EMF, VOLT, UX

Real Constants

R1 - Element identification number

Material Properties

None

Surface Loads

via **RMLVSCALE** command

Body Loads

via **RMLVSCALE** command

Special Features

Nonlinear
Prestress

KEYOPT(1)

Select DOF set:

0 --

No ROM master nodes will be used (default).

1 --
ROM master nodes are used.

KEYOPT(2)
Select matrix option:

0 --
Unsymmetric matrix option (default).

1 --
Symmetric matrix option (must be activated in case of **ANTYPE** = MODAL).

ROM144 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution.
- Additional element output as shown in the following table.

Table 1 ROM144 Element Output Definitions

Name	Definition
SENG	Strain energy
CAP1	First capacitance defined by **RMCAP**
CAP2	Second capacitance defined by **RMCAP**
CAP3	Third capacitance defined by **RMCAP**
CAP4	Forth capacitance defined by **RMCAP**
CAP5	Fifth capacitance defined by **RMCAP**
CAP6	Sixth capacitance defined by **RMCAP**
CAP7	Seventh capacitance defined by **RMCAP**
CAP8	Eighth capacitance defined by **RMCAP**
CAP9	Ninth capacitance defined by **RMCAP**
CAP10	Tenth capacitance defined by **RMCAP**

Table 2,"ROM Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table in the *Elements Reference* for more information. The following notation is used in *Table 2,"ROM Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1,"ROM144 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 ROM Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SENG	NMISC	1

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CAP1	NMISC	2
CAP2	NMISC	3
CAP3	NMISC	4
CAP4	NMISC	5
CAP5	NMISC	6
CAP6	NMISC	7
CAP7	NMISC	8
CAP8	NMISC	9
CAP9	NMISC	10
CAP10	NMISC	11

ROM144 Assumptions and Restrictions

- Modal forces may not be applied to the ROM element.
- Harmonic and modal analyses are valid only for small-signal analyses after a static prestress calculation.
- Using different ROM elements (i.e., based on different ROM database and polynomial coefficient files) in the same Use Pass is not supported.

ROM144 Product Restrictions

There are no product-specific restrictions for this element.

PLANE145

2-D Quadrilateral Structural Solid p-Element

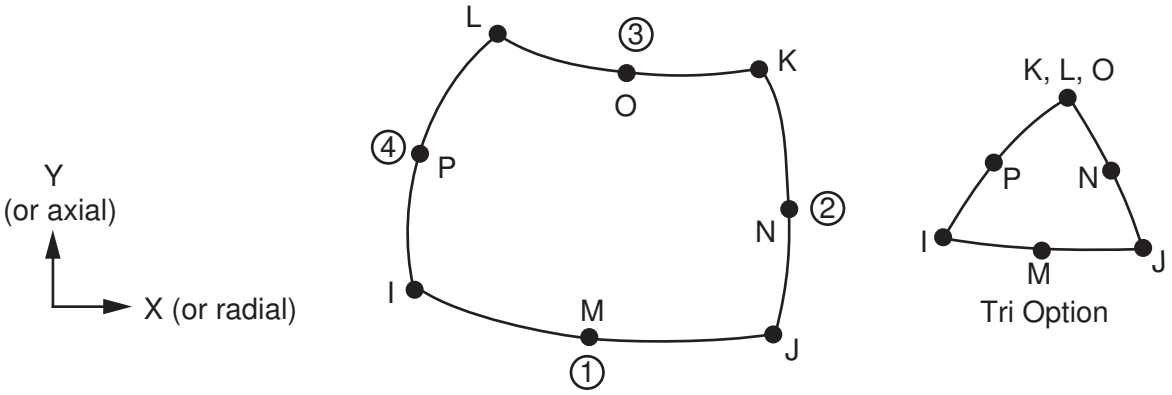
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

PLANE145 Element Description

PLANE145 is a quadrilateral p-element that supports a polynomial with a maximum order of eight.

The element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress or plane strain) or as an axisymmetric element. See PLANE145 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE145 Geometry



PLANE145 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE145 Geometry"*. Midside nodes may not be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. PLANE146 is a similar but 6-node triangular element. In addition to the nodes, the element input data includes a thickness for the plane stress option only (KEYOPT(3) = 3), and the orthotropic material properties. Orthotropic material directions correspond to the global coordinate directions.

Element loads are described in *Section 2.8: Node and Element Loads* of the *Elements Reference*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "PLANE145 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified nodal temperatures default to TUNIF.

The nodal forces, if any, should be input per unit depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

A summary of the element input is given in *PLANE145 Input Summary*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE145 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT (3) = 0, 1, 2
TK - Thickness, if KEYOPT (3) = 3

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (I-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T (M), T(N), T(O), T(P)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default)

N --
Maximum possible p-level ($2 \leq N \leq 8$)

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness input (real constant TK)

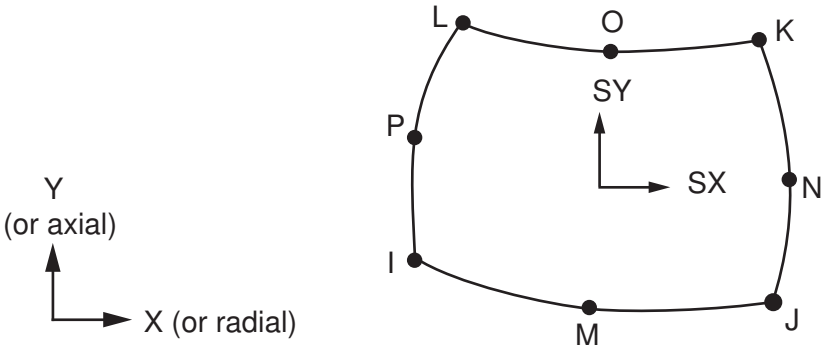
PLANE145 Output Data

No solution output is produced for this element. The results output associated with the element is in two forms:

- Displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE145 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE145 Stress Output"*. Displacements at the midside nodes are approximate for curved edges.

Figure 2 PLANE145 Stress Output



See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE145 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*1*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	Y
S:1, 2, 3	Principal stresses	-	-
S:INT	Stress intensity	-	-
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY	Elastic strains	-	Y
EPEL:1, 2, 3	Principal elastic strains	-	-
EPEL:EQV	Equivalent elastic strains [*2*]	-	Y
P-LEVEL	p-level used	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strain uses an effective Poisson's ratio; for elastic, this value is set by the user (**MP**,PRXY).

Note

For axisymmetric solutions, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains.

Table 2, "PLANE145 Item and Component Labels" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE145 Item and Component Labels"*:

Name
output quantity as defined in the *Table 1, "PLANE145 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 PLANE145 Item and Component Labels

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-LEVEL	NMISC	1

PLANE145 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE145 Geometry"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the positive X quadrants.
- Nodal forces should only be applied to corner nodes.
- Imposed displacements may only vary linearly along an edge. Any nonlinear variation is ignored.
- This element does not support inertia relief.

PLANE145 Product Restrictions

There are no product-specific restrictions for this element.

PLANE146

2-D Triangular Structural Solid p-Element

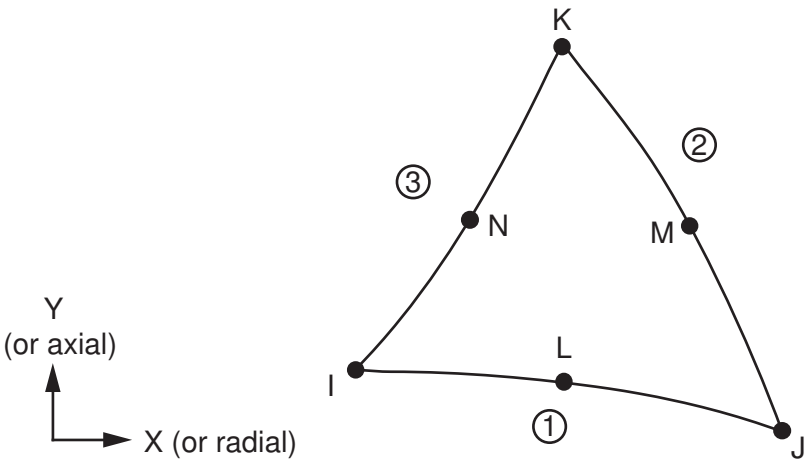
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

PLANE146 Element Description

PLANE146 is a triangular p-element that supports a polynomial with a maximum order of eight.

The element is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress or plane strain) or as an axisymmetric element. See PLANE146 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE146 Geometry



PLANE146 Input Data

The geometry and node locations for this element are shown in *Figure 1, "PLANE146 Geometry"*. Midside nodes may not be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

In addition to the nodes, the element input data includes a thickness (only if KEYOPT(3) = 3) and the orthotropic material properties. Orthotropic material directions correspond to the global coordinate directions.

Element loads are described in *Section 2.8: Node and Element Loads* of the *Elements Reference*. Pressures may be input as surface loads on the element faces, shown by the circled numbers in *Figure 1, "PLANE146 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

A summary of the element input is given in *PLANE146 Input Summary*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE146 Input Summary

Nodes
I, J, K, L, M, N

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT (3) = 0, 1, 2
TK - Thickness, if KEYOPT (3) = 3

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ, (or CTEX, CTEY, CTEZ *or* THSX, THSY,THSZ), DENS, GXY

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (I-K)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default) Defaults to 8.

N --
Maximum possible p-level ($2 \leq N \leq 8$)

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness input (real constant TK)

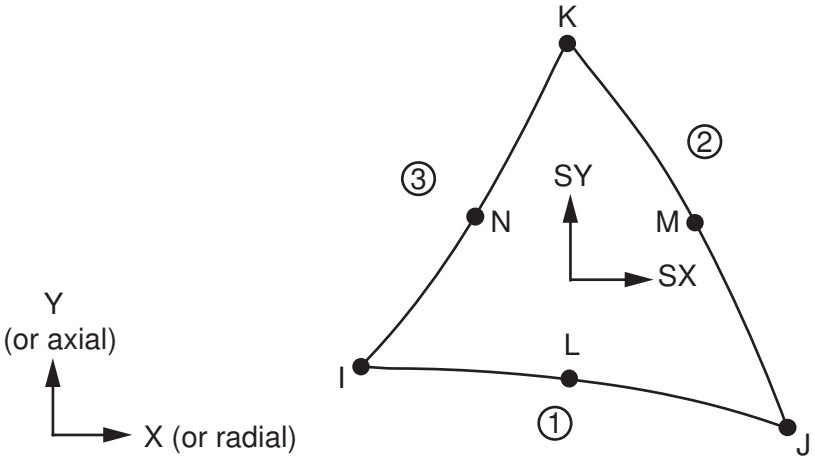
PLANE146 Output Data

No solution output is produced for this element. The results output associated with the element is in two forms:

- Displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE146 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE146 Stress Output"*. Displacements at the midside nodes are approximate for curved edges.

Figure 2 PLANE146 Stress Output



See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE146 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*1*
NODES	Nodes - I, J, K, L, M, N	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N)	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY	Elastic strains	-	Y

Name	Definition	O	R
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain [*2*]	-	Y
P-LEVEL	p-level used	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strain uses an effective Poisson's ratio; for elastic, this value is set by the user (**MP**,PRXY).

Table 2, "PLANE146 Item and Component Labels" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE146 Item and Component Labels"*:

Name
: output quantity as defined in the *Table 1, "PLANE146 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 PLANE146 Item and Component Labels

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-LEVEL	NMISC	1

PLANE146 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE146 Geometry"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the positive X quadrants.
- Nodal forces should only be applied to corner nodes.
- Imposed displacements may only vary linearly along an edge. Any nonlinear variation is ignored.
- This element does not support inertia relief.

PLANE146 Product Restrictions

There are no product-specific restrictions for this element.

SOLID147

3-D Brick Structural Solid p-Element

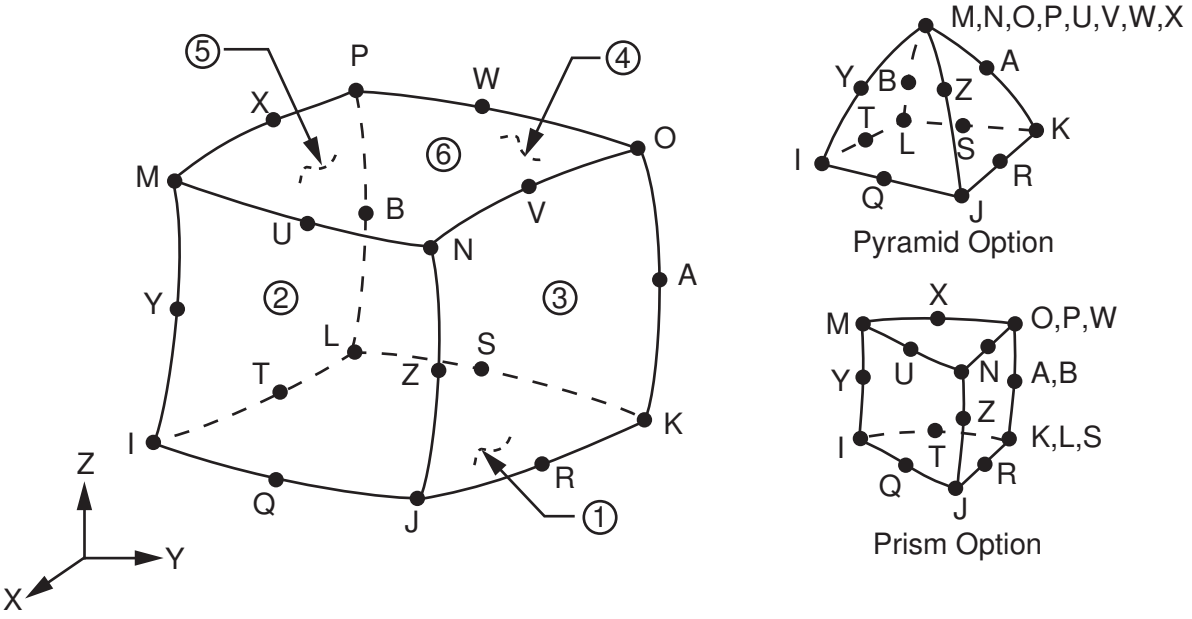
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SOLID147 Element Description

SOLID147 is a brick p-element that supports a polynomial with a maximum order of eight.

The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. SOLID148 is a tetrahedron-shaped p-element. See SOLID147 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID147 Geometry



SOLID147 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID147 Geometry"*. Midside nodes may not be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes. A prism-shaped element can be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W.

In the addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the global coordinate directions.

Element loads are described in *Section 2.8: Node and Element Loads* of the *Elements Reference*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID147 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *SOLID147 Input Summary*.

SOLID147 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), . . . , T(Z), T(A), T(B)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default)

N --
Maximum possible p-level ($2 \leq N \leq 8$)

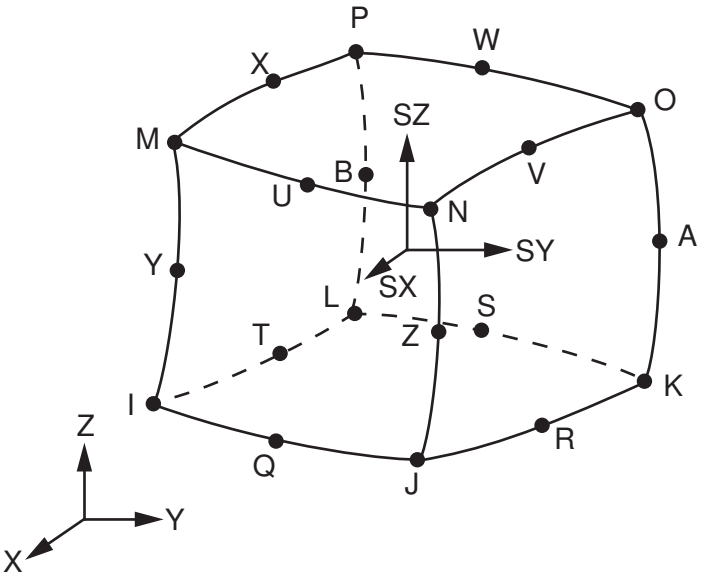
SOLID147 Output Data

No solution output is produced for this element. The results output associated with the element is in two forms:

- Displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID147 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID147 Stress Output"*. Displacements at the midside nodes are approximate for curved edges.

Figure 2 SOLID147 Stress Output



See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID147 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, . . . Z, A, B	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*1*
TEMP	Temperatures T(I), T(J), ... , T(Z), T(A), T(B)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [*2*]	-	Y
P-LEVEL	p-level used	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strain uses an effective Poisson's ratio: for elastic this value is set by the user (**MP**,PRXY).

Table 2, "SOLID147 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID147 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID147 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 SOLID147 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-LEVEL	NMISC	1

SOLID147 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID147 Geometry"* or may have the planes IJKL and MNOP interchanged.
- Nodal forces should only be applied to corner nodes.
- Imposed displacements may only vary linearly along an edge or face. Any nonlinear variation is ignored.
- This element does not support inertia relief.

SOLID147 Product Restrictions

There are no product-specific restrictions for this element.

SOLID148

3-D Tetrahedral Structural Solid p-Element

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

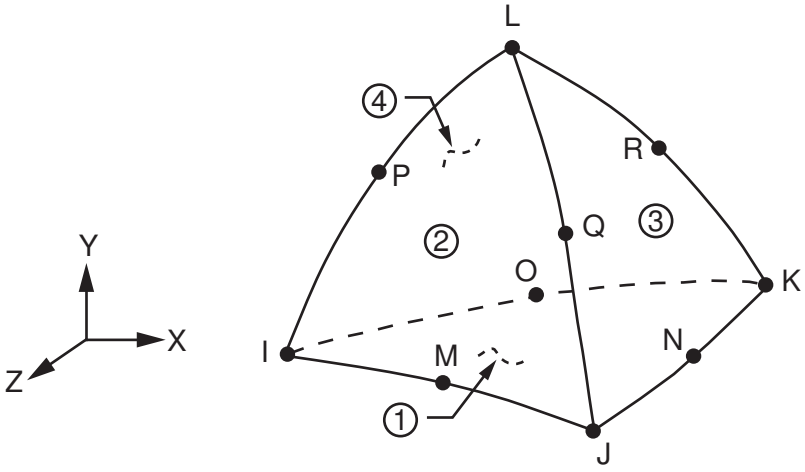
SOLID148 Element Description

SOLID148 is a tetrahedron-shaped p-element that supports a polynomial with a maximum order of eight.

The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

SOLID147 is a brick-shaped p-element. See SOLID148 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID148 Geometry



SOLID148 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID148 Geometry"*. Midside nodes cannot be removed. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes. Elements may be numbered either as shown in *Figure 1, "SOLID148 Geometry"* or may have node L below the IJK plane. In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the global coordinate directions.

Element loads are described in *Section 2.8: Node and Element Loads* in the *Elements Reference*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID148 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

A summary of the element input is given in *SOLID148 Input Summary*.

SOLID148 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features
None

KEYOPT(1)
Starting p-level:

0 --
Use global starting p-level [**PPRANGE**] (default)

N --
Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)
Maximum possible p-level:

0 --
Use global maximum p-level [**PPRANGE**] (default)

N --
Maximum possible p-level ($2 \leq N \leq 8$)

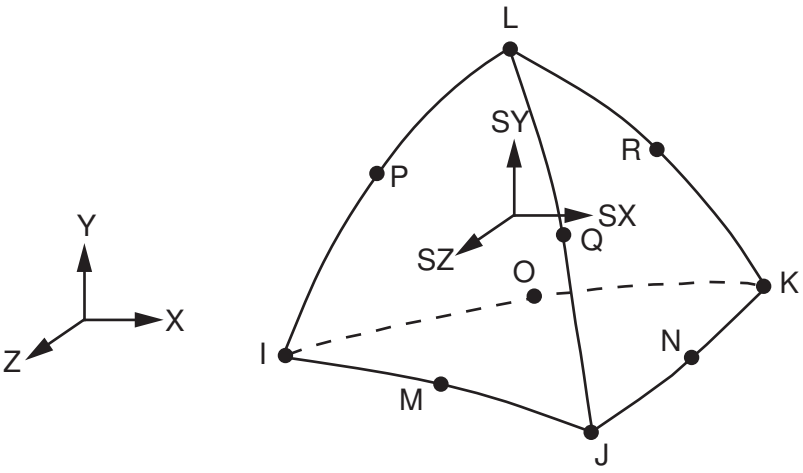
SOLID148 Output Data

No solution output is produced for this element. The results output associated with the element is in two forms:

- Displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID148 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID148 Stress Output"*. Displacements at the midside nodes are approximate for curved edges.

Figure 2 SOLID148 Stress Output



See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID148 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*1*
NODES	Nodes - I, J, ... , R	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), ... , T(R)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [*2*]	Y	Y
P-LEVEL	p-level used	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strain uses an effective Poisson's ratio: for elastic this value is set by the user (**MP**,PRXY).

Additional data are available for this element by using the **ETABLE** command. *Table 2, "SOLID148 Item and Sequence Numbers"* lists the data items needed for the **ETABLE** command.

Table 2, "SOLID148 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID148 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLID148 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 SOLID148 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-LEVEL	NMISC	1

SOLID148 Assumptions and Restrictions

- The element must not have a zero volume.
- Nodal forces should only be applied to corner nodes.
- Imposed displacements may only vary linearly along an edge or face. Any nonlinear variation is ignored.
- This element does not support inertia relief.

SOLID148 Product Restrictions

There are no product-specific restrictions for this element.

SHELL150

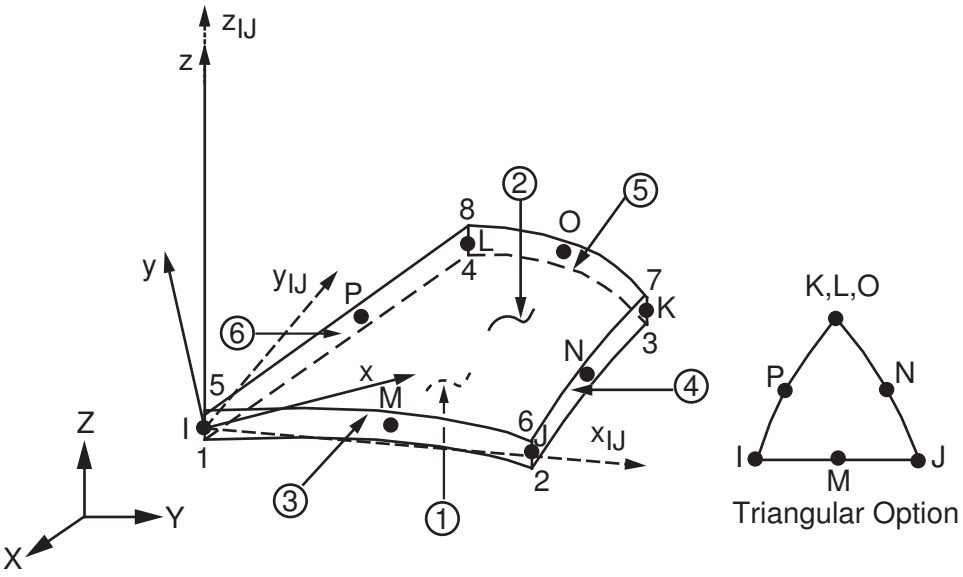
8-Node Structural Shell p-Element

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL150 Element Description

SHELL150 is a structural shell p-element that supports a polynomial with a maximum order of eight. This element is particularly well suited to model curved shells. It has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. See SHELL150 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL150 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.
x = Element x-axis if ESYS is supplied.

SHELL150 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SHELL150 Geometry"*. The element is defined by eight nodes and four thicknesses. Midside nodes may not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

The element has orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the corner nodes. The thickness at the midside nodes is taken as the average of the corresponding corner nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input. The total thickness of each shell element must be less than twice the radius of curvature, and should be less than one-fifth the radius of curvature. ADMSUA is the added mass per unit area.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL150 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length. Temperatures may be input as element body loads at the "corner" locations (1-8) shown in *Figure 1, "SHELL150 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T1 is used for T1, T2, T3, and T4, while T2 (as input) is used for T5, T6, T7, and T8. For any other input pattern, unspecified temperatures default to TUNIF. Only the lumped mass matrix is available.

A summary of the element input is given in *SHELL150 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL150 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J
TK(K) - Shell thickness at node K
TK(L) - Shell thickness at node L
(Blank)
ADMSUA - Added mass/unit area

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperature --
T1, T2, T3, T4, T5, T6, T7, T8

Special Features
None

KEYOPT(1)
Starting p-level:

0 --

Use global starting p-level [**PPRANGE**] (default)

N --

Starting p-level ($2 \leq N \leq 8$)

KEYOPT(2)

Maximum possible p-level:

0 --

Use global maximum p-level [**PPRANGE**] (default)

N --

Maximum possible p-level ($2 \leq N \leq 8$)

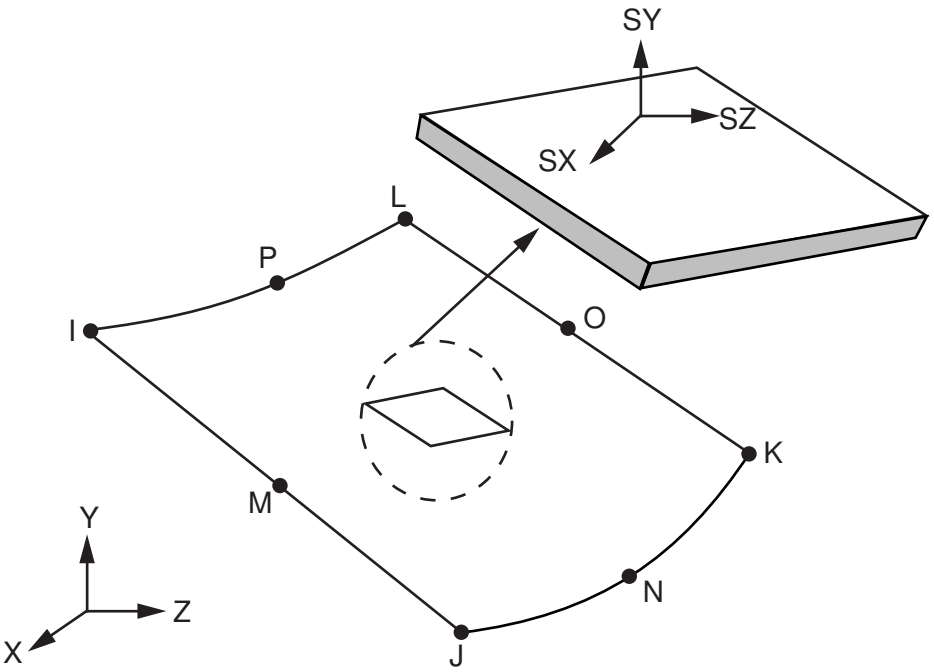
SHELL150 Output Data

No solution output is produced for this element. The results output associated with the element is in two forms:

- Displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL150 Element Output Definitions"*

Several items are illustrated in *Figure 2, "Stress Output"*. The element output (RSYS = SOLU) is always in the global coordinate system. Displacements at the midside nodes are approximate for curved edges.

Figure 2 Stress Output



See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL150 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*1*
TEMP	T1, T2, T3, T4, T5, T6, T7, T8	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	Y
S:1, 2, 3	Principal stress	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	Y
EPEL:1, 2, 3	Principal stress	-	Y
EPEL:EQV	Equivalent elastic strains [*2*]	-	Y
P-LEVEL	p-level used	-	Y

1. Available only at centroid as a ***GET** item.
2. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 2, "SHELL150 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SHELL150 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SHELL150 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 SHELL150 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
p-LEVEL	NMISC	1

SHELL150 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness.
- Shear deflections are included in this element.

- This element does not support inertia relief.

SHELL150 Product Restrictions

There are no product-specific restrictions for this element.

SURF151

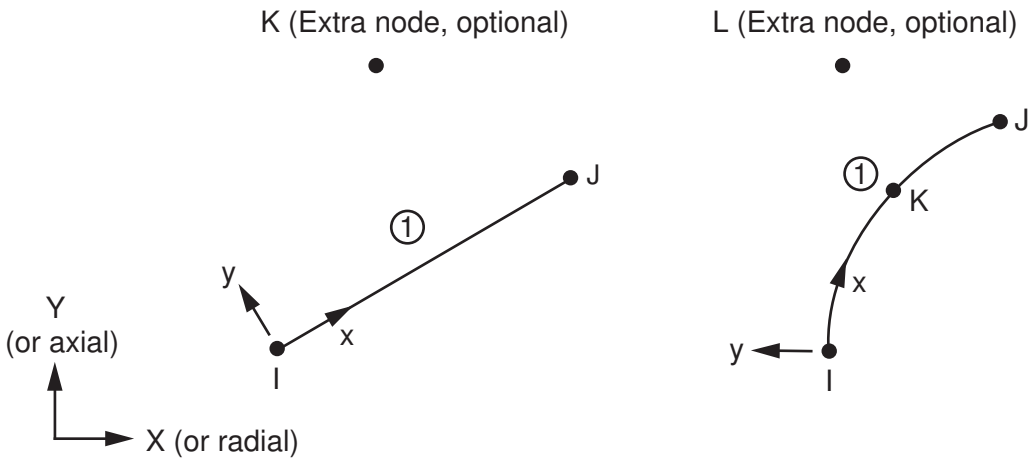
2-D Thermal Surface Effect

MP ME ST PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

SURF151 Element Description

SURF151 may be used for various load and surface effect applications. It may be overlaid onto a face of any 2-D thermal solid element (except axisymmetric harmonic elements PLANE75 and PLANE78). The element is applicable to 2-D thermal analyses. Various loads and surface effects may exist simultaneously. See SURF151 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SURF151 Geometry



SURF151 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF151 Geometry"*. The element is defined by two to four node points and the material properties. An extra node (away from the base element) may be used for convection or radiation effects.

The mass, volume, and heat generation calculations use the in-plane element thicknesses at node I and J (real constants TKI and TKJ, respectively). TKJ defaults to TKI, which defaults to 1.0. If KEYOPT(3) = 3, the out-of-plane thickness is input as the real constant TKPS (defaults to 1.0). The mass calculation uses the density (material property DENS).

See *Section 2.8: Node and Element Loads* for a description of element loads. Convections or heat fluxes may be input as surface loads on the element.

The convection surface conductivity matrix calculation uses the film coefficient (input on the **SFE** command with *KVAL* = 0 and *CONV* as the label). If the extra node option is used, its temperature becomes the bulk temperature. If the extra node is not used, the *CONV* value input with *KVAL* = 2 becomes the bulk temperature. The convection surface heat flow vector calculation uses the bulk temperature. On a given face, either a heat flux or a convection may be specified, but not both simultaneously.

Setting KEYOPT(7) = 1 multiplies the evaluated film coefficient by the empirical term ITS-TBI^n, where TS is the element surface temperature, TB is the fluid bulk temperature, and n is an empirical coefficient (real constant ENN).

If KEYOPT(5) = 1 and flow information is available from FLUID116 with KEYOPT(2) = 1, the bulk temperature may be adjusted to the adiabatic wall temperature using KEYOPT(6) = 1, real constants OMEG (rotational speed) and NRF (recovery factor), and the logic described in the *Theory Reference for ANSYS and ANSYS Workbench*. For this adjustment, the global Y Cartesian coordinate axis is used as the axis of rotation (KEYOPT(3) = 1). When using the OMEG real constant, you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %*tabname*%). Rotational speed (OMEG) can vary with time and location. Use the ***DIM** command to dimension the table and identify the variables. For more information and examples on using table inputs, see Array Parameters of the *ANSYS Parametric Design Language Guide*, Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide* and Doing a Thermal Analysis Using Tabular Boundary Conditions in the *Thermal Analysis Guide*, as well as the description of ***DIM** in the *Commands Reference*.

A film coefficient specified by the **SFE** command may be modified by activating the user subroutine USERCV with the **USRCAL** command. USERCV may be used to modify the film coefficient of a surface element with or without an extra node. It may be used if the film coefficient is a function of temperature and/or location.

If the surface element has an extra node (KEYOPT(5) = 1), the bulk temperature and/or the film coefficient may be redefined in a general way by user programmable routine USRSURF116. USRSURF116 may be used if the bulk temperature and/or the film coefficient is a function of fluid properties, velocity and/or wall temperature. If a bulk temperature is determined by USRSURF116, it overrides any value specified by **SFE** or according to KEYOPT(6). Also, if a film coefficient is determined by USRSURF116, it overrides any values specified by **SFE** or USRCAL, USERCV. USRSURF116 calculation are activated by modifying the USRSURF116 subroutine and creating a customized version of ANSYS; there will be no change in functionality without modifying USRSURF116. For more information, see *Section 14.1: User-Programmable Features (UPFs)*.

Heat generation rates are input on a per unit volume basis and may be input as an element body load at the nodes, using the **BFE** command. Element body loads are not applied to other elements connected at the same nodes. The node I heat generation HG(I) defaults to zero. The node J heat generation defaults to HG(I). The heat generation load vector calculation uses the heat generation rate values.

As an alternative to using the **BFE** command, you can specify heat generation rates directly at the nodes using the **BF** command. For more information on body loads, see Body Loads in the *Basic Analysis Guide*.

SURF151 allows for radiation between the surface and the extra node. The emissivity of the surface (input as material property EMIS for the material number of the element) is used for the radiation surface conductivity matrix. The form factor FORMF and the Stefan-Boltzmann constant SBCONST are also used for the radiation surface conductivity matrix. The form factor can be either input as a real constant (defaults to 1) using KEYOPT(9) = 1 or it can be calculated automatically as a cosine effect using KEYOPT(9) = 2 or 3. For information on how the cosine effect depends on basic element orientation and the extra node location, see the *Theory Reference for ANSYS and ANSYS Workbench*. There is no distance effect included in the cosine effect. For axisymmetric analyses, the automatic form factor calculation is used only with the extra node on the Y-axis. The Stefan-Boltzmann constant defaults to 0.119×10^{-10} (Btu/hr*in^2* °R^4).

When KEYOPT(4) = 0, an edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

If a single PLANE element lies beneath SURF151, you can automatically set the element behavior (plane stress, axisymmetric, or plane stress with thickness [including TKPS if applicable]) to that of the underlying solid element using KEYOPT(3) =10. This option is valid only when a single PLANE element lies beneath the SURF element. For example, if you apply a SURF151 element over a PLANE77 (thermal) element whose nodes are also used in the definition of a PLANE82 (structural) element, a warning appears and the load is not applied to the element.

A summary of the element input is given in *SURF151 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

SURF151 Input Summary

Nodes

I, J if KEYOPT (4) = 1, and KEYOPT (5) = 0
I, J, K if KEYOPT (4) = 1, and KEYOPT(5) = 1
I, J, K if KEYOPT (4) = 0, and KEYOPT(5) = 0
I, J, K, L if KEYOPT (4) = 0, and KEYOPT(5) = 1

Degrees of Freedom
TEMP

Real Constants

FORMF, SBCONST, (Blank), OMEG, NRF, VABS,
TKI, TKJ, (Blank), (Blank), (Blank), TKPS,
ENN, GC, JC
See *Table 1, "SURF151 Real Constants"* for a description of the real constants

Material Properties

DENS (for density)
EMIS (for emissivity, if KEYOPT(9) > 0)

Surface Loads

Convections --
face 1 (I-J) if KEYOPT(8) > 1

Heat Fluxes --
face 1 (I-J) if KEYOPT(8) = 1

Body Loads

Heat Generation --
HG(I), HG(J); also HG(K) if KEYOPT(4) = 0

Special Features
Birth and death

KEYOPT(1)
Adiabatic wall temperature option:

0, 1, 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(2)
Recovery factor (F_R) option:

0, 1, or 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

3 --
Plane with thickness input (TKPS)

10 --
Use the element behavior (plane, axisymmetric, or plane with thickness input [include TKPS if applicable]) of the underlying solid element.

KEYOPT(4)
Midside nodes:

0 --
Has midside node (that matches the adjacent solid element)

1 --
No midside node

KEYOPT(5)
Extra node for radiation and/or convection calculations:

0 --
No extra nodes

1 --
Has extra node (optional if KEYOPT(8) > 1; required if KEYOPT(9) > 0)

KEYOPT(6) (used only if KEYOPT(5) = 1 and KEYOPT(8) > 1)
Use of bulk temperatures:

0 --
Extra node temperature used as bulk temperature

1 --
Adiabatic wall temperature used as bulk temperature

KEYOPT(7)
Empirical term:

0 --
Do not multiply film coefficient by empirical term.

1 --
Multiply film coefficient by empirical term $|TS-TB|^n$.

KEYOPT(8)
Heat flux and convection loads:

0 --
Ignore heat flux and convection surface loads (if any)

1 --
Include heat flux, ignore convection

Use the following to include convection (ignore heat flux):

2 --
Evaluate film coefficient h_f (if any) at average film temperature, (TS +TB)/2

3 --
Evaluate h_f at element surface temperature, TS

4 --
Evaluate h_f at fluid bulk temperature, TB

5 --
Evaluate h_f at differential temperature, | TS - TB |

KEYOPT(9)

Radiation form factor calculation:

0 --

Do not include radiation

1 --

Use radiation with the form factor real constant

2 --

Use radiation with cosine effect computed as an absolute value (ignore real constant)

3 --

Use radiation with cosine effect computed as zero if negative (ignore real constant)

Table 1 SURF151 Real Constants

No.	Name	Description
1	FORMF	Form factor
2	SBCONT	Stefan-Boltzmann constant
3	(Blank)	--
4	OMEGA	Angular velocity
5	NRF	Recovery factor
6	VABS	Absolute value of fluid velocity (KEYOPT(1) = 0)
7	TKI	In-plane thickness at node I
8	TKJ	In-plane thickness at node J
9-11	(Blank)	--
12	TKPS	Out-of-plane thickness (if KEYOPT(3) = 3)
13	ENN	Empirical coefficient
14	GC	Gravitational constant used for units consistency
15	JC	Joule constant used to convert work units to heat units

SURF151 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "SURF151 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SURF151 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*7*
VN(X, Y)	Components of unit vector normal to center of element	-	Y
DENSITY	Density	-	*1*
MASS	Mass of Element	-	*1*
HGEN	Heat generations HG(I), HG(J), HG(K)	*2*	-
HEAT GEN. RATE	Heat generation rate over entire element (HGTOT)	*2*	*2*
HFLUX	Input heat flux at nodes I, J	*3*	-
HEAT FLOW RATE	Input heat flux heat flow rate over element surface area (HFCTOT)	*3*	*3*
HFILM	Film coefficient at each face node	*4*	*4*
TBULK	Bulk temperature at each face node or temperature of extra node	*4*	*4*
TAVG	Average surface temperature	*4*	*4*
TAW	Adiabatic wall temperature	*5*	*5*
RELVEL	Relative velocity	*5*	*5*
SPHTFL	Specific heat of the fluid	*5*	*5*
RECFAC	Recovery factor	*5*	*5*
CONV. HEAT RATE	Convection heat flow rate over element surface area (HFCTOT)	*4*	*4*
CONV. HEAT RATE/AREA	Average convection heat flow rate per unit area	*4*	-
EMISSUR	Average emissivity of surface (for element material number)	*6*	*6*
EMISEXT	Emissivity of extra node	*6*	*6*
TEMPSUR	Average temperature of surface	*6*	*6*
TEMPEXT	Temperature of extra node	*6*	*6*
FORM FACTOR	Average form factor of element	*6*	*6*
RAD. HEAT RATE	Radiation heat flow rate over entire element (HRTOT)	*6*	*6*
RAD. HEAT RATE/AREA	Average radiation heat flow rate per unit area	*6*	-

1. If dens > 0
2. If heat generation load is present
3. If KEYOPT(8) = 1
4. If KEYOPT(8) > 1
5. If KEYOPT(6) = 1 and KEYOPT(8) > 1
6. If KEYOPT(9) > 0

7. Available only at centroid as a ***GET** item.

Table 3, "SURF151 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SURF151 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 2, "SURF151 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 3 SURF151 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HGTOT	SMISC	1
HFCTOT	SMISC	2
HRTOT	SMISC	3
AREA	NMISC	1
VNX	NMISC	2
VNY	NMISC	3
HFILM	NMISC	5
TAVG	NMISC	6
TBULK	NMISC	7
TAW	NMISC	8
RELVEL	NMSC	9
SPHTFL	NMSC	10
RECFAC	NMSC	11
EMISSUR	NMISC	12
EMISEXT	NMISC	13
TEMPSUR	NMISC	14
TEMPEXT	NMISC	15
FORM FACTOR	NMISC	16
DENS	NMISC	17
MASS	NMISC	18

SURF151 Assumptions and Restrictions

- The element must not have a zero length.
- If KEYOPT(9) > 0 (radiation is used):

- element is nonlinear and requires an iterative solution
- extra node must be present.
- if KEYOPT(4) = 0, midside nodes may not be dropped.

SURF151 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- KEYOPT(3) = 3 is not applicable.
- The TKPS real constant (R12) is not applicable.
- The only allowable material property is EMIS.
- No special features are allowed.

SURF152

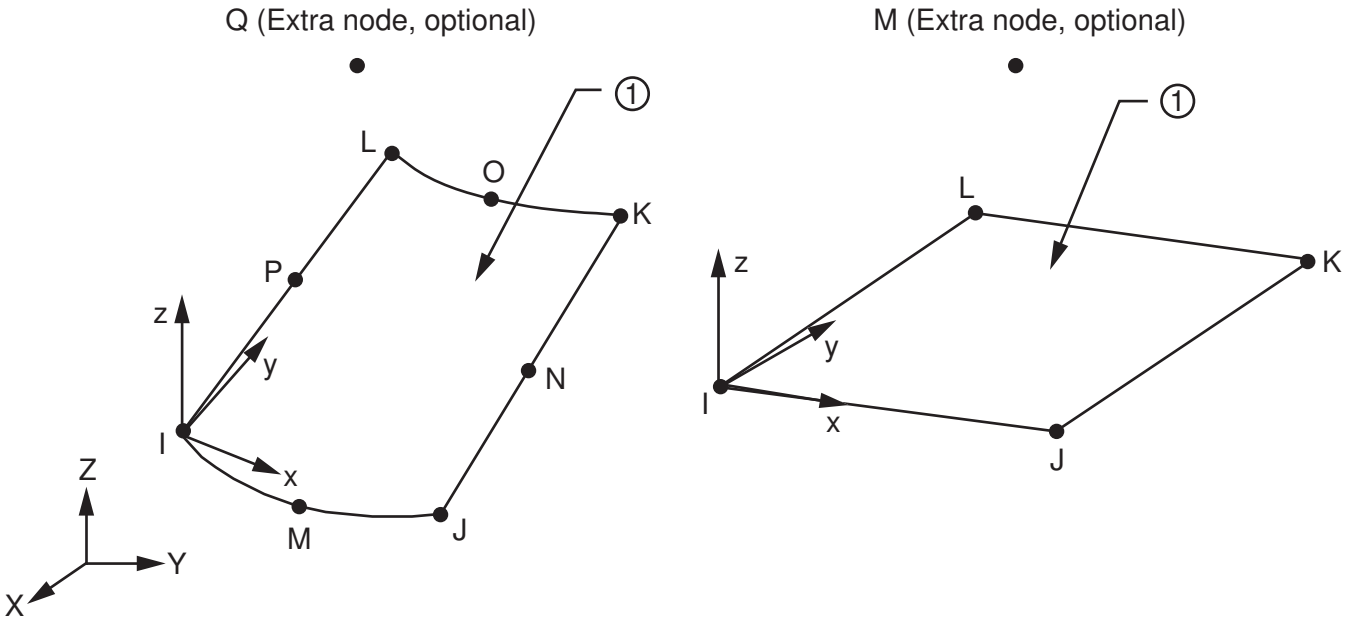
3-D Thermal Surface Effect

MP ME ST PR PRN DS <> <> <> <> <> PP VT
Product Restrictions

SURF152 Element Description

SURF152 may be used for various load and surface effect applications. It may be overlaid onto an area face of any 3-D thermal element. The element is applicable to 3-D thermal analyses. Various loads and surface effects may exist simultaneously. See SURF152 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SURF152 Geometry



SURF152 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF152 Geometry"*. The element is defined by four to nine nodes and the material properties. An extra node (away from the base element) may be used for convection or radiation effects. A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. The element x-axis is parallel to the I-J side of the element.

The mass, volume, and heat generation calculations use the element thicknesses (real constants TKI, TKJ, TKK, TKL). Thicknesses TKJ, TKK, and TKL default to TKI, which defaults to 1.0. The mass calculation uses the density (material property DENS).

See *Section 2.8: Node and Element Loads* for a description of element loads. Convections or heat fluxes may be input as surface loads on the element.

The convection surface conductivity matrix calculation uses the film coefficient (input on the **SFE** command with *KVAL* = 0 and *CONV* as the label. If the extra node is used, its temperature becomes the bulk temperature. If the extra node is not used, the *CONV* value input with *KVAL* = 2 becomes the bulk temperature. The convection

surface heat flow vector calculation uses the bulk temperature. On a given face, either a heat flux or a convection may be specified, but not both simultaneously.

Setting KEYOPT(7) = 1 multiplies the evaluated film coefficient by the empirical term ITS - TBI^n, where TS is the element surface temperature, TB is the fluid bulk temperature, and n is an empirical coefficient (real constant ENN).

If KEYOPT(5) = 1 and flow information is available from FLUID116 with KEYOPT(2) = 1, the bulk temperature may be adjusted to the adiabatic wall temperature using KEYOPT(6) = 1, real constants OMEG (rotational speed) and NRF (recovery factor), and the logic described in the *Theory Reference for ANSYS and ANSYS Workbench*. For this adjustment, the axis of rotation may be defined as the global Cartesian X, Y or Z coordinate axis (KEYOPT(3)). When using the OMEG real constant, you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %*tabname*%). Rotational speed (OMEG) can vary with time and location. Use the ***DIM** command to dimension the table and identify the variables. For more information and examples on using table inputs, see Array Parameters of the *ANSYS Parametric Design Language Guide*, Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide*, and Doing a Thermal Analysis Using Tabular Boundary Conditions in the *Thermal Analysis Guide*, as well as the description of the ***DIM** command in the *Commands Reference*.

A film coefficient specified by the **SFE** command may be modified by activating the user subroutine USERCV with the **USRCAL** command. USERCV may be used to modify the film coefficient of a surface element with or without an extra node. It may be used if the film coefficient is a function of temperature and/or location.

If the surface element has an extra node (KEYOPT(5) = 1), the bulk temperature and/or the film coefficient may be redefined in a general way by user programmable routine USRSURF116. USRSURF116 may be used if the bulk temperature and/or the film coefficient is a function of fluid properties, velocity and/or wall temperature. If a bulk temperature is determined by USRSURF116, it overrides any value specified by **SFE** or according to KEYOPT(6). Also, if a film coefficient is determined by USRSURF116, it overrides any values specified by **SFE** or USRCAL, USERCV. USRSURF116 calculation are activated by modifying the USRSURF116 subroutine and creating a customized version of ANSYS; there will be no change in functionality without modifying USRSURF116. For more information, see *Section 14.1: User-Programmable Features (UPFs)*.

Heat generation rates are input on a per unit volume basis and may be input as an element body load at the nodes, using the **BFE** command. Element body loads are not applied to other elements connected at the same nodes. The node I heat generation HG(I) defaults to zero. If all other heat generations are unspecified, they default to HG(I). If all corner node heat generations are specified, each midside node heat generation defaults to the average heat generation of its adjacent corner nodes. For any other input heat generation pattern, unspecified heat generations default to zero. The heat generation load vector calculation uses the heat generation rate values.

As an alternative to using the **BFE** command, you can specify heat generation rates directly at the nodes using the **BF** command. For more information on body loads, see Body Loads in the *Basic Analysis Guide*.

SURF152 allows for radiation between the surface and the extra node. The emissivity of the surface (input as material property EMIS for the material number of the element) is used for the radiation surface conductivity matrix. The form factor FORMF and the Stefan-Boltzmann constant SBCONST are also used for the radiation surface conductivity matrix. The form factor can be either input as a real constant (defaults to 1) using KEYOPT(9) = 1 or it can be calculated automatically as a cosine effect using KEYOPT(9) = 2 or 3. For information on how the cosine effect depends on basic element orientation and the extra node location, see the *Theory Reference for ANSYS and ANSYS Workbench*. There is no distance effect included in the cosine effect. The Stefan-Boltzmann constant defaults to 0.119×10^{-10} (Btu/hr*in^2* °R^4)).

When KEYOPT(4) = 0, an edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A summary of the element input is given in *SURF152 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SURF152 Input Summary

Nodes

I, J, K, L if KEYOPT (4) = 1 and KEYOPT (5) = 0
I, J, K, L, M if KEYOPT (4) = 1 and KEYOPT (5) = 1
I, J, K, L, M, N, O, P if KEYOPT (4) = 0 and KEYOPT (5) = 0
I, J, K, L, M, N, O, P, Q if KEYOPT (4) = 0 and KEYOPT (5) = 1

Degrees of Freedom

KEYOPT(11) Setting	DOF for all nodes except extra node	DOF for extra node (KEYOPT(5) = 1)
0	TEMP	TEMP
1	TTOP	TEMP
2	TBOT	TEMP

Real Constants

FORMF, SBCONST, (Blank), OMEG, NRF, VABS,
TKI, TKJ, TKK, TKL, (Blank), (Blank),
ENN, GC, JC
See *Table 1, "SURF152 Real Constants"* for a description of the real constants

Material Properties

DENS (for density)
EMIS (for emissivity, if KEYOPT(9) > 0)

Surface Loads

Convections --
face 1 (I-J-K-L) if KEYOPT(8) > 1

Heat Fluxes --
face 1 (I-J-K-L) if KEYOPT(8) = 1

Body Loads

Heat Generation --
HG(I), HG(J), HG(K), HG(L), and, if KEYOPT(4) = 0, HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(1)
Adiabatic wall temperature option:

0, 1, 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(2)
Recovery factor (F_R) option:

0, 1, or 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(3)
Axis of symmetry:

0 --
OMEG used about global Cartesian X-axis

1 --
OMEG used about global Cartesian Y-axis

2 --
OMEG used about global Cartesian Z-axis

KEYOPT(4)
Midside nodes:

0 --
Has midside nodes (that match the adjacent solid element)

1 --
Does not have midside nodes

KEYOPT(5)
Extra node for radiation and/or convection calculations:

0 --
No extra node

1 --
Has extra node (optional if KEYOPT (8) > 1; required if KEYOPT (9) > 0)

KEYOPT(6) (used only if KEYOPT(5) = 1 and KEYOPT(8) > 1)
Use of bulk temperature:

0 --
Extra node temperature used as bulk temperature

1 --
Adiabatic wall temperature used as bulk temperature

KEYOPT(7)
Empirical term:

0 --
Do not multiply film coefficient by empirical term.

1 --
Multiply film coefficient by empirical term $|TS-TB|^n$.

KEYOPT(8)
Heat flux and convection loads:

0 --
Ignore heat flux and convection surface loads (if any)

1 --
Include heat flux, ignore convection

Use the following to include convection (ignore heat flux):

2 --
Evaluate film coefficient h_f (if any) at average film temperature, (TS +TB)/2

3 --
Evaluate h_f at element surface temperature, TS

4 --
Evaluate h_f at fluid bulk temperature, TB

5 --
Evaluate h_f at differential temperature, | TS - TB |

KEYOPT(9)
Radiation form factor calculation:

0 --
Do not include radiation

1 --
Use radiation with the form factor real constant

2 --
Use radiation with cosine effect calculated as an absolute value (ignore real constant)

3 --
Use radiation with cosine effect calculated as zero if negative (ignore real constant)

KEYOPT(11)
Label used for all nodal degrees of freedom (except for the extra node):

0 --
TEMP

1 --
TTOP

2 --
TBOT

The extra node, if requested with KEYOPT(5) = 1, is always TEMP.

Table 1 SURF152 Real Constants

No.	Name	Description
1	FORMF	Form factor
2	SBCONT	Stefan-Boltzmann constant
3	(Blank)	--
4	OMEGA	Angular velocity (KEYOPT(6) = 1)
5	NRF	Recovery factor
6	VABS	Absolute value of fluid velocity (KEYOPT(1) = 0)
7	TKI	Thickness at node I
8	TKJ	Thickness at node J
9	TKK	Thickness at node K
10	TKL	Thickness at node L
11-12	(Blank)	--
13	ENN	Empirical coefficient
14	GC	Gravitational constant used for units consistency
15	JC	Joule constant used to convert work units to heat units

SURF152 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "SURF152 Element Output Definitions"*

Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SURF152 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J, K, L	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*7*
VN(X, Y, Z)	Components of unit vector normal to center of element	-	Y
DENSITY	Density	-	*1*
MASS	Mass of element	-	*1*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	*2*	-
HEAT GEN. RATE	Heat generation rate over entire element (HGTOT)	*2*	*2*
HFLUX	Input heat flux at nodes I, J, K, L	*3*	-
HEAT FLOW RATE	Input heat flux heat flow rate over element surface area (HFCTOT)	*3*	*3*
HFILM	Film coefficient at each face node	*4*	*4*
TBULK	Bulk temperature at each face node or temperature of extra node	*4*	*4*
TAVG	Average surface temperature	*4*	*4*
TAW	Adiabatic wall temperature	*5*	*5*
RELVEL	Relative velocity	*5*	*5*
SPHTFL	Specific heat of the fluid	*5*	*5*
RECFAC	Recovery factor	*5*	*5*
CONV. HEAT RATE	Convection heat flow rate over element surface area (HFCTOT)	*4*	*4*

Name	Definition	O	R
CONV. HEAT RATE/AREA	Average convection heat flow rate per unit area	*4*	-
EMISSUR	Average emissivity of surface (for element material number)	*6*	*6*
EMISEXT	Emissivity of extra node	*6*	*6*
TEMPSUR	Average temperature of surface	*6*	*6*
TEMPEXT	Temperature of extra node	*6*	*6*
FORM FACTOR	Average form factor of element	*6*	*6*
RAD. HEAT RATE	Radiation heat flow rate over entire element (HRTOT)	*6*	*6*
RAD. HEAT RATE/AREA	Average radiation heat flow rate per unit area	*6*	-

1. If dens > 0
2. If heat generation load is present
3. If KEYOPT(8) = 1
4. If KEYOPT(8) > 1
5. If KEYOPT(6) = 1 and KEYOPT(8) > 1
6. If KEYOPT(9) > 0
7. Available only at centroid as a ***GET** item.

Table 3, "SURF152 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SURF152 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 2, "SURF152 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 3 SURF152 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HGTOT	SMISC	1
HFCTOT	SMISC	2
HRTOT	SMISC	3
AREA	NMISC	1
VNX	NMISC	2
VNY	NMISC	3
VNZ	NMISC	4

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HFILM	NMISC	5
TAVG	NMISC	6
TBULK	NMISC	7
TAW	NMISC	8
RELVEL	NMISC	9
SPHTFL	NMISC	10
RECFAC	NMISC	11
EMISSUR	NMISC	12
EMISEXT	NMISC	13
TEMPSUR	NMISC	14
TEMPEXT	NMISC	15
FORM FACTOR	NMISC	16
DENS	NMISC	17
MASS	NMISC	18

SURF152 Assumptions and Restrictions

- The element must not have a zero area.
- If KEYOPT(9) > 0 (radiation is used):
  - element is nonlinear and requires an iterative solution
  - extra node must be present.
  - if KEYOPT(4) = 0, midside nodes may not be dropped.

SURF152 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only allowable material property is EMIS.
- No special features are allowed.

SURF153

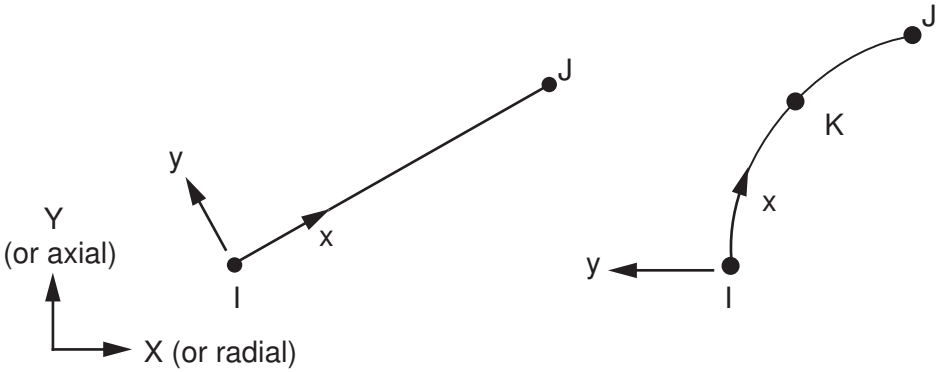
2-D Structural Surface Effect

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SURF153 Element Description

SURF153 may be used for various load and surface effect applications. It may be overlaid onto a face of any 2-D structural solid element (except axisymmetric harmonic elements PLANE25, PLANE83, and FLUID81). The element is applicable to 2-D structural analyses. See SURF153 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SURF153 Geometry



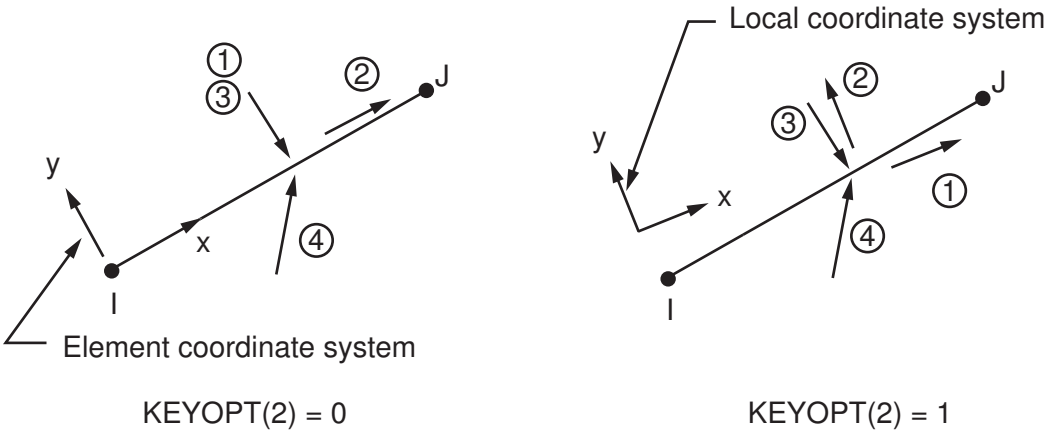
SURF153 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF153 Geometry"*. The element is defined by two or three node points and the material properties. The element x-axis is along to the I-J line of the element.

The mass and volume calculations use the in-plane element thicknesses at node I and J (real constants TKI and TKJ, respectively). TKJ defaults to TKI, which defaults to 1.0. If KEYOPT(3) = 3, the out-of-plane thickness is input as the real constant TKPS (defaults to 1.0). The mass calculation uses the density (material property DENS, mass per unit volume) and the real constant ADMSUA, the added mass per unit area. The stiffness matrix calculation uses the in-plane force per unit length (input as real constant SURT) and the elastic foundation stiffness using pressure-per-length (or force-per-length-cubed) units (input as real constant EFS). The foundation stiffness can be damped, either by using the material property DAMP as a multiplier on the stiffness or by directly using the material property VISC.

See *Section 2.8: Node and Element Loads* for a description of element loads. Pressures may be input as surface loads as force-per-length-squared on the element faces as shown by the circled numbers on *Figure 2, "Pressures"*. SURF153 allows complex pressure loads.

Figure 2 Pressures



Faces 1 and 2 [KEYOPT(2) = 0] Positive values of pressure on the first two faces act in the positive element coordinate directions (except for the normal pressure which acts in the negative z direction). For face 1, positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid.

Faces 1 and 2 [KEYOPT(2) = 1] Pressure loads are applied to the element faces according to the local coordinate system as follows: face 1 in the local x direction and face 2 in the local y direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command. KEYOPT(6) does not apply.

Face 3 The magnitude of the pressure at each integration point is $P_I + XP_J + YP_K$, where P_I through P_K are input as VAL1 through VAL3 on the **SFE** command, and X and Y are the global Cartesian coordinates at the current location of the point. The **SFFUN** and **SFGRAD** commands do not work with face 3.

Face 4 Tthe magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j)/(P_J^2 + P_K^2)^{1/2}$ where i and j are unit vectors in the global Cartesian directions. The load magnitude can be adjusted with KEYOPTS(11) and (12). When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered, but the load magnitude is the average of the computed corner node magnitudes.

The effects of pressure load stiffness are automatically included for this element for real pressure on face 1 if KEYOPT(2) = 0 or on face 3. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

Temperatures may be input as element body loads at the nodes. Element body load temperatures are not applied to other elements connected at the same nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I). Temperatures are used for material property evaluation only.

When KEYOPT(4) = 0, a removed midside node implies that the displacement varies linearly, rather than parabolically. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

If a single PLANE element lies beneath SURF153, you can automatically set the element behavior (plane stress, axisymmetric, or plane stress with thickness [including TKPS if applicable]) to that of the underlying solid element using KEYOPT(3) =10. This option is valid only when a single PLANE element lies beneath the SURF element. For example, if you apply a SURF153 element over a PLANE77 (thermal) element whose nodes are also used in the definition of a PLANE82 (structural) element, a warning appears and the load is not applied to the element.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times an area; however, if the area changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the area changes.

A summary of the element input is given in *SURF153 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

If using SURF153 with **VTGEOM** parameters, see *Section 4.2.4: Element Support*.

SURF153 Input Summary

Nodes

I, J if KEYOPT (4) = 1,
I, J, K if KEYOPT(4) = 0

Degrees of Freedom
UX, UY

Real Constants

(Blank), (Blank), (Blank), EFS, SURT, ADMSUA,
TKI, TKJ, (Blank), (Blank), (Blank), TKPS
See *Table 1, "SURF153 Real Constants"* for a description of the real constants

Material Properties
DENS, VISC, DAMP

Surface Loads

Pressures --

face 1 (I-J) (in -y normal direction)
face 2 (I-J) (in +x tangential direction)
face 3 (I-J) (in -y normal direction, global taper)
face 4 (I-J) (oriented by input vector)

Body Loads

Temperatures --
T(I), T(J); also T(K) if KEYOPT(4) = 0

Special Features

Stress stiffening
Large deflection
Birth and death

KEYOPT(2)
Pressure applied to faces 1 and 2 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain

3 --
Plane stress with thickness input (TKPS)

5 --
Generalized plane strain

10 --
Use the element behavior--plane stress, axisymmetric, plain strain, plane stress with thickness input (include TKPS if applicable), or generalized plane strain--of the underlying solid element.

KEYOPT(4)
Midside nodes:

0 --
Has midside node (that matches the adjacent solid element)

1 --
No midside node

KEYOPT(6)
Applicable only to normal direction pressure (faces 1 and 3):

0 --
Use pressures as calculated (positive and negative)

1 --
Use positive pressures only (negative set to zero)

2 --
Use negative pressures only (positive set to zero)

KEYOPT(7)
Loaded area during large-deflection analyses:

0 --
Use new area

1 --
Use original area

KEYOPT(11)
Pressure applied by vector orientation (face 4):

0 --
On projected area and includes tangential component

1 --
On projected area and does not include tangential component

2 --
On full area and includes the tangential component

KEYOPT(12)
Effect of the direction of the element normal (element y-axis) on vector oriented (face 4) pressure:

0 --
Pressure load is applied regardless of the element normal orientation

1 --
Pressure load is not used if the element normal is oriented in the same general direction as the pressure vector

Table 1 SURF153 Real Constants

No.	Name	Description
1 ... 3	(Blank)	--
4	EFS	Foundation stiffness
5	SURT	Surface tension
6	ADMSUA	Added mass/unit area
7	TKI	In-plane thickness at node I (defaults to 1.0)
8	TKJ	In-plane thickness at node J (defaults to TKI)
9 ... 11	(Blank)	--
12	TKPS	Out-of-plane thickness if KEYOPT(3) = 3 (defaults to 1.0)

SURF153 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "SURF153 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SURF153 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*6*
VN(X, Y)	Components of unit vector normal to center of element	-	Y
PRES	Pressures P1, P2, P3, P4 at nodes I, J	*1*	-

Name	Definition	O	R
PY, PX	Pressures at nodes in element coordinate system (P4 uses an average element coordinate system)	-	*1*
AVG. FACE PRESSURE	Average normal pressure (P1AVG), Average tangential pressure (P2AVG), Average tapered normal pressure (P3AVG), Effective value of vector oriented pressure (P4EFF)	*1*	*1*
DVX, DVY	Direction vector of pressure P4	*1*	*1*
TEMP	Surface temperatures T(I), T(J), T(K)	*2*	*2*
DENSITY	Density	*3*	*3*
MASS	Mass of Element	*3*	*3*
FOUNDATION STIFFNESS	Foundation Stiffness (input as EFS)	*4*	*4*
FOUNDATION PRESSURE	Foundation Pressure	*4*	*4*
SURFACE TENSION	Surface Tension (input as SURT)	*5*	*5*

1. If pressure load
2. If temperature load
3. If dens > 0
4. If EFS > 0
5. If SURT > 0
6. Available only at centroid as a ***GET** item.

Table 3, "SURF153 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SURF153 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SURF153 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

I,J
: sequence number for data at nodes I and J

Table 3 SURF153 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PY (real)	SMISC	-	1	2
PX (real)	SMISC	-	3	4
PY (imaginary)	SMISC		27	28

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PX (imaginary)	SMISC		29	30
P1AVG (real)	SMISC	13	-	-
P2AVG (real)	SMISC	14	-	-
P3AVG (real)	SMISC	15	-	-
P4EFF (real)	SMISC	16	-	-
P1AVG (imaginary)	SMISC	39	-	-
P2AVG (imaginary)	SMISC	40	-	-
P3AVG (imaginary)	SMISC	41	-	-
P4EFF (imaginary)	SMISC	42	-	-
FOUNPR	SMISC	21	-	-
AREA	NMISC	1	-	-
VNX	NMISC	2	-	-
VNY	NMISC	3	-	-
EFS	NMISC	5	-	-
SURT	NMISC	6	-	-
DENS	NMISC	7	-	-
MASS	NMISC	8	-	-
DVX	NMISC	9	-	-
DVY	NMISC	10	-	-

SURF153 Assumptions and Restrictions

- The element must not have a zero length.
- The surface tension load vector acts along the line connecting nodes I and J as a force applied to the nodes seeking to minimize the length of the line. If the nodes of the element are not coplanar when using surface tension, equilibrium may be lost.
- For structural large deflection analyses, the loads are applied to the current size of the element, not the initial size.
- Surface printout and foundation stiffness are not valid for elements deactivated [**EKILL**] and then reactivated [**EALIVE**]. Surface printout does not include large strain effects.

SURF153 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.
- The VISC and DAMP material properties are not applicable.

ANSYS Structural

- The only allowable material property is DENS.

SURF154

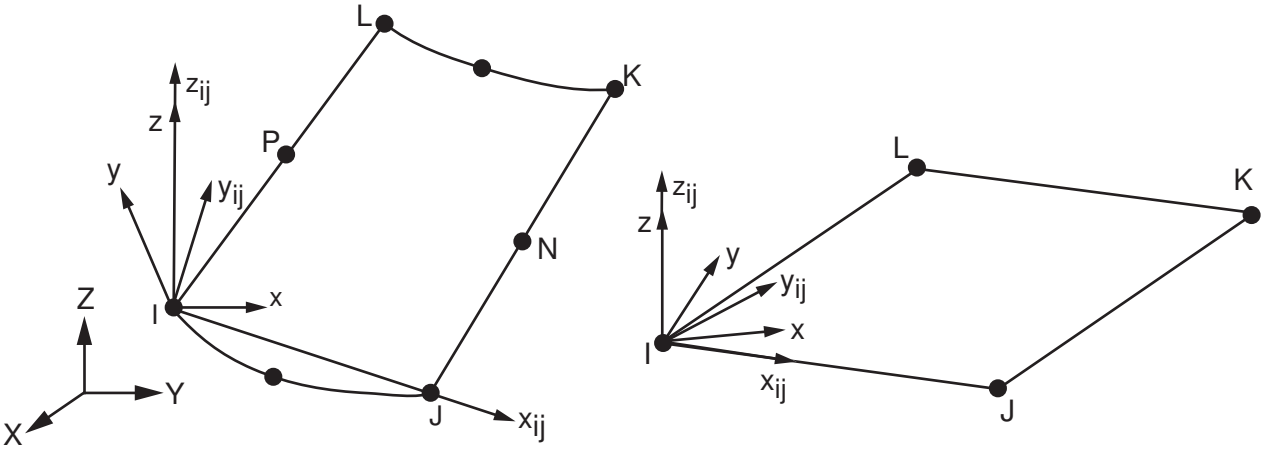
3-D Structural Surface Effect

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SURF154 Element Description

SURF154 may be used for various load and surface effect applications. It may be overlaid onto an area face of any 3-D element. The element is applicable to 3-D structural analyses. Various loads and surface effects may exist simultaneously. See SURF154 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SURF154 Geometry



SURF154 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF154 Geometry"*. The element is defined by four to eight nodes and the material properties. A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. The default element x-axis is parallel to the I-J side of the element.

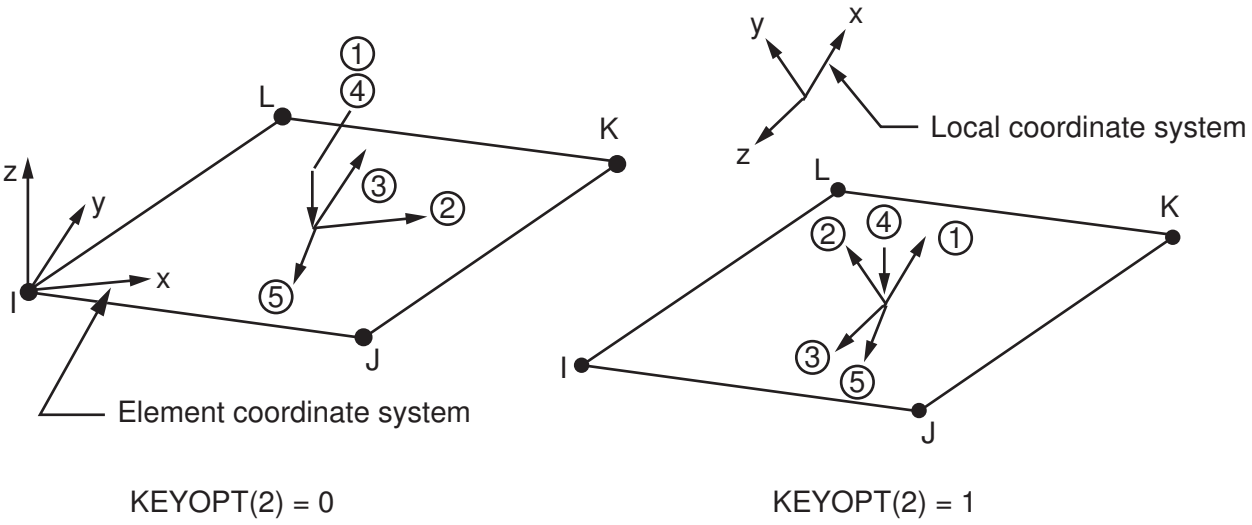
The mass and volume calculations use the element thicknesses (real constants TKI, TKJ, TKK, TKL). Thicknesses TKJ, TKK, and TKL default to TKI, which defaults to 1.0. The mass calculation uses the density (material property DENS, mass per unit volume) and the real constant ADMSUA, the added mass per unit area. The stiffness matrix calculation uses the in-plane force per unit length (input as real constant SURT) and the elastic foundation stiffness using pressure-per-length (or force-per-length-cubed) units (input as real constant EFS). The foundation stiffness can be damped, either by using the material property DAMP as a multiplier on the stiffness or by directly using the material property VISC.

See *Section 2.8: Node and Element Loads* for a description of element loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 2, "Pressures"*. SURF154 allows complex pressure loads.

Faces 1, 2, and 3 [KEYOPT(2) = 0] Positive values of pressure on the first three faces act in the positive element coordinate directions (except for the normal pressure which acts in the negative z direction). For face 1, positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. For faces 2 and 3, the direction of the load is controlled by the element coordinate system; therefore, the **ESYS** command is normally needed.

Faces 1, 2, and 3 [KEYOPT(2) = 1] Pressure loads are applied to the element faces according to the local coordinate system, as follows: face 1 in the local x direction, face 2 in the local y direction, and face 3 in the local z direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command. KEYOPT(6) does not apply.

Figure 2 Pressures



Face 4 The direction is normal to the element and the magnitude of the pressure at each integration point is $P_I + XP_J + YP_K + ZP_L$, where P_I through P_L are input as *VAL1* through *VAL4* on the **SFE** command, and X, Y, Z are the global Cartesian coordinates at the current location of the point. Positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. The **SFFUN** and **SFGRAD** commands do not work with face 4.

Face 5 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j + P_L k)/(P_J^2 + P_K^2 + P_L^2)^{1/2}$ where i, j, and k are unit vectors in the global Cartesian directions. The load magnitude may be adjusted with KEYOPT(11) and KEYOPT(12). When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered but the load magnitude is the average of the computed corner node magnitudes.

The effects of pressure load stiffness are automatically included for this element for real pressure on face 1 if KEYOPT(2) = 0 or on face 4. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command.

Temperatures may be input as element body loads at the nodes. Element body load temperatures are not applied to other elements connected at the same nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF. Temperatures are used for material property evaluation only.

When KEYOPT(4) = 0, an edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times an area; however, if the area changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the area changes.

A summary of the element input is given in *SURF154 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

If using SURF154 with **VTGEOM** parameters, see *Section 4.2.4: Element Support*.

SURF154 Input Summary

Nodes

I, J, K, L if KEYOPT (4) = 1
I, J, K, L, M, N, O, P if KEYOPT (4) = 0

Degrees of Freedom
UX, UY, UZ

Real Constants

(Blank), (Blank), (Blank), EFS, SURT, ADMSUA,
TKI, TKJ, TKK, TKL
See *Table 1, "SURF154 Real Constants"* for a description of the real constants

Material Properties
DENS, VISC, DAMP

Surface Loads

Pressures --

face 1 (I-J-K-L) (in -z normal direction)
face 2 (I-J-K-L) (tangential (+x))
face 3 (I-J-K-L) (tangential (+y))
face 4 (I-J-K-L) (in -z normal direction, global taper)
face 5 (I-J-K-L) (oriented by input vector)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L); also T(M), T(N), T(O), T(P) if KEYOPT(4) = 0

Special Features

Stress stiffening
Large deflection

Birth and death

KEYOPT(2)
Pressure applied to faces 1, 2, and 3 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(4)
Midside nodes:

0 --
Has midside nodes (that match the adjacent solid element)

1 --
Does not have midside nodes

KEYOPT(6)
Applicable only to normal direction pressure (faces 1 and 4):

0 --
Use pressures as calculated (positive and negative)

1 --
Use positive pressures only (negative set to zero)

2 --
Use negative pressures only (positive set to zero)

Note

To use KEYOPT(6), KEYOPT(2) must equal 0.

KEYOPT(7)
Loaded area during large-deflection analyses:

0 --
Use new area

1 --
Use original area

KEYOPT(11)
Pressure applied by vector orientation (face 5):

0 --
On projected area and includes tangential component

1 --
On projected area and does not include tangential component

2 --
On full area and includes the tangential component

KEYOPT(12)
Effect of the direction of the element normal (element z-axis) on vector oriented (face 5) pressure:

0 --
Pressure load is applied regardless of the element normal orientation

1 --
Pressure load is not used if the element normal is oriented in the same general direction as the pressure vector.

Table 1 SURF154 Real Constants

No.	Name	Description
1 ... 3	(Blank)	--
4	EFS	Foundation stiffness
5	SURT	Surface tension
6	ADMSUA	Added mass/unit area
7	TKI	Thickness at node I (defaults to 1.0)
8	TKJ	Thickness at node J (defaults to TKI)

No.	Name	Description
9	TKK	Thickness at node K (defaults to TKI)
10	TKL	Thickness at node L (defaults to TKI)

SURF154 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 2, "SURF154 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SURF154 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J, K, L	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*6*
VN(X, Y, Z)	Components of unit vector normal to center of element	-	Y
PRES	Pressures P1, P2, P3, P4, P5 at nodes I, J, K, L	*1*	-
PZ, PX, PY	Pressures at nodes in element coordinate system (P5 uses an average element coordinate system)	-	*1*
DVX, DVY, DVZ	Direction vector of pressure P5	*1*	*1*
AVG. FACE PRESSURE	Average normal pressure (P1AVG), Average tangential-X pressure (P2AVG), Average tangential-Y pressure (P3AVG), Average tapered normal pressure (P4AVG), Effective value of vector oriented pressure (P5EFF)	*1*	*1*
TEMP	Surface temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	*2*	*2*
DENSITY	Density	*3*	*3*
MASS	Mass of element	*3*	*3*
FOUNDATION STIFFNESS	Foundation Stiffness (input as EFS)	*4*	*4*

Name	Definition	O	R
FOUNDATION PRESSURE	Foundation Pressure	*4*	*4*
SURFACE TENSION	Surface Tension (input as SURT)	*5*	*5*

1. If pressure load
2. If temperature load
3. If dens > 0
4. If EFS > 0
5. If SURT > 0
6. Available only at centroid as a ***GET** item.

Table 3, "SURF154 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (**/POST1**) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 3, "SURF154 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SURF154 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 3 SURF154 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PZ (real)	SMISC	-	1	2	3	4
PX (real)	SMISC	-	5	6	7	8
PY (real)	SMISC	-	9	10	11	12
PZ (imaginary)	SMISC	-	27	28	29	30
PX (imaginary)	SMISC	-	31	32	33	34
PY (imaginary)	SMISC	-	35	36	37	38
P1AVG (real)	SMISC	13	-	-	-	-
P2AVG (real)	SMISC	14	-	-	-	-
P3AVG (real)	SMISC	15	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P4AVG (real)	SMISC	16	-	-	-	-
P5EFF (real)	SMISC	17	-	-	-	-
P1AVG (imaginary)	SMISC	39	-	-	-	-
P2AVG (imaginary)	SMISC	40	-	-	-	-
P3AVG (imaginary)	SMISC	41	-	-	-	-
P4AVG (imaginary)	SMISC	42	-	-	-	-
P5EFF (imaginary)	SMISC	43	-	-	-	-
FOUNPR	SMISC	21	-	-	-	-
AREA	NMISC	1	-	-	-	-
VNX	NMISC	2	-	-	-	-
VNY	NMISC	3	-	-	-	-
VNZ	NMISC	4	-	-	-	-
EFS	NMISC	5	-	-	-	-
SURT	NMISC	6	-	-	-	-
DENS	NMISC	7	-	-	-	-
MASS	NMISC	8	-	-	-	-
DVX	NMISC	9	-	-	-	-
DVY	NMISC	10	-	-	-	-
DVZ	NMISC	11	-	-	-	-

SURF154 Assumptions and Restrictions

- The element must not have a zero area.
- The surface-tension load vector acts in the plane of the element as a constant force applied to the nodes seeking to minimize the area of the surface. If the nodes of the element are not coplanar when using surface tension, equilibrium may be lost.
- For structural large-deflection analyses, the loads are applied to the current size of the element, not the initial size.
- Surface printout and foundation stiffness are not valid for elements deactivated [**EKILL**] and then reactivated [**EALIVE**]. Surface printout does not include large strain effects.

SURF154 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

- The VISC and DAMP material properties are not applicable.

ANSYS Structural

- The only allowable material property is DENS.

SURF156

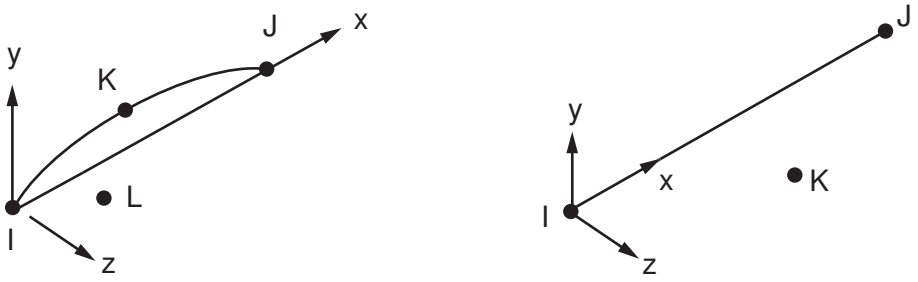
3-D Structural Surface Line Load Effect

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

SURF156 Element Description

SURF156 may be used for applying line pressure loads on structures. It may be overlaid onto the edge of any 3-D element. The element is applicable to 3-D structural analyses. Various loads and surface effects may exist simultaneously. See the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SURF156 Geometry



SURF156 Input Data

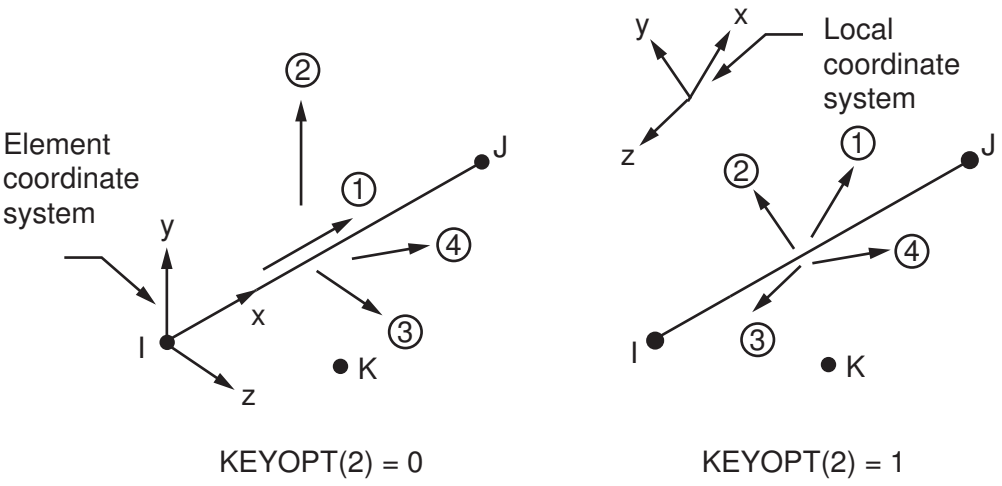
The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF156 Geometry"*. The element is defined by two to four nodes (KEYOPT(4) = 0 or 1). The extra node is required for orientation of the element loads and lies in the element x-z plane. The element x-axis is parallel to the line connecting nodes I and J of the element.

See *Section 2.8: Node and Element Loads* for a description of element loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 2, "Pressures"*. SURF156 allows complex pressure loads. The input units are force per length.

Faces 1, 2, and 3 [KEYOPT(2)]=0 Positive values of pressure on the first three faces act in the positive element coordinate directions. For faces 2 and 3, the direction of the load is controlled by the element coordinate system which is oriented by the extra node; therefore, the **ESYS** command has no effect. When using large deflection (**NLGEOM**,ON), the orientation of the loads may change based on the new location of the nodes. If the extra node is on another element that moves, the extra node will move with it. If the extra node is not on another element, the node cannot move.

Faces 1, 2, and 3 [KEYOPT(2)]=1 Pressure loads are applied to the element faces according to the local coordinate system, as follows: face 1 in the x direction, face 2 in the local y direction, and face 3 in the local z direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command.

Figure 2 Pressures



Face 4 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j + P_L k)/(P_J^2 + P_K^2 + P_L^2)^{1/2}$ where i, j, and k are unit vectors in the global Cartesian directions. When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered but the load magnitude is the average of the computed corner node magnitudes.

The effects of pressure load stiffness are automatically included for this element for real pressure on faces 2 and 3 if KEYOPT(2) = 0. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times a unit length; however, if the length changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the length changes.

A summary of the element input is given in *SURF156 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SURF156 Input Summary

Nodes

I, J, K, if KEYOPT (4) = 1 or 2
I, J, K, L, if KEYOPT (4) = 0

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
None

Surface Loads

Pressures --

face 1 (parallel to x direction)
face 2 (parallel to y direction)
face 3 (parallel to z direction)
face 4 (oriented by input vector)

Body Loads
None

Special Features

Stress stiffening
Large deflection

KEYOPT(2)
Pressure applied to faces 1, 2, and 3 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(4)
Midside node:

0 --
Has extra node and a midside node that matches the adjacent solid element

1 --
Has extra node but does not have a midside node

2 --
Does not have an extra node, and has an *optional* midside node. Use only for load on face 1 or face 4.

KEYOPT(7)
Loaded area during large-deflection analyses:

0 --
Use new area

1 --
Use original area

SURF156 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in *Table 1, "SURF156 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SURF156 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
EXTRA NODE	Extra (orientation) node	Y	Y
PRESSURES	Pressures P1, P2, P3, P4 at nodes I, J	*1*	-
VECTOR DIRECTION	Direction vector of pressure P4	*1*	*1*

1. If pressure load

Table 2, "SURF156 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (**/POST1**) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SURF156 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SURF156 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J
sequence number for data at nodes I, J

Table 2 SURF156 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
P1 (real)	SMISC	-	1	2
P2 (real)	SMISC	-	3	4
P3 (real)	SMISC	-	5	6
P4 (real)	SMISC	7	-	-
P1 (imaginary)	SMISC	-	8	9
P2 (imaginary)	SMISC	-	10	11
P3 (imaginary)	SMISC	-	12	13
P4 (imaginary)	SMISC	14	-	-
P4 (real) VECTOR DIRECTION	NMISC	1 - 3	-	-
P4 (imaginary) VECTOR DIRECTION	NMISC	4 - 6	-	-

SURF156 Assumptions and Restrictions

- The element must not have a zero length, and the extra node (when used) can not be colinear with nodes I and J.

SURF156 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

SHELL157

Thermal-Electric Shell

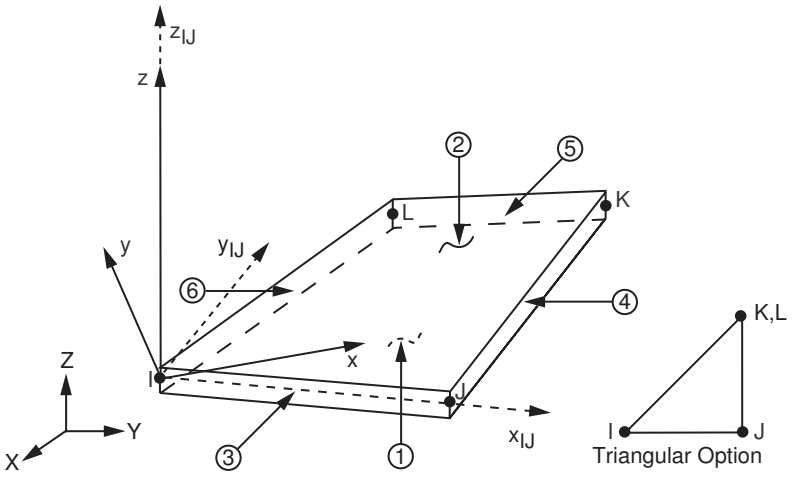
MP ME <> PR PRN <> <> <> EM <> <> PP <>
Product Restrictions

SHELL157 Element Description

SHELL157 is a 3-D element having in-plane thermal and electrical conduction capability. The element has four nodes with two degrees of freedom, temperature and voltage, at each node. The element applies to a 3-D, steady-state or transient thermal analysis, although the element includes no transient electrical capacitance or inductance effects. The element requires an iterative solution to include the Joule heating effect in the thermal solution. See SHELL157 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. If no electrical effects are present, the 3-D thermal shell (SHELL57) may be used.

If the model containing the thermal-electrical element is also to be analyzed structurally, replace the element with an equivalent structural element (such as SHELL63). If both in-plane and transverse thermal-electric conduction are needed, use a thermal-electric solid element (SOLID69).

Figure 1 SHELL157 Geometry



SHELL157 Input Data

The geometry, node locations, and coordinate systems for this element are shown in *Figure 1, "SHELL157 Geometry"*. The element is defined by four nodes, four thicknesses, a material direction angle, and the orthotropic material properties.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, you need to specify only TK(I). If the thickness is not constant, you must specify all four thicknesses.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. The element x-axis may be rotated by an angle THETA (in degrees). You can assign the specific heat and density any values for steady-state solutions. The electrical material property, RSV_, is the resistivity of the material. You can specify the resistivity, like any other material property, as a function of temperature. Properties not specified default as described in *Section 2.4: Linear Material Properties*.

Specify the word VOLT for the *Lab* variable on the **D** command and the voltage input for the value. Specify the word AMPS for the *Lab* variable on the **F** command and the current into the node input for the value.

Element loads are described in *Section 2.8: Node and Element Loads*. Convection or heat flux (but not both) and radiation may be specified as surface loads at the element faces as shown by the circled numbers on *Figure 1, "SHELL157 Geometry"*. Edge convection and flux loads are input on a per unit length basis.

Heat generation rates may be specified as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). This rate is in addition to the Joule heat generated by the current flow.

SHELL157 Input Summary summarizes the element input. A general description of element input appears in *Section 2.1: Element Input*.

SHELL157 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP, VOLT

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J; defaults to TK(I)
TK(K) - Shell thickness at node K; defaults to TK(I)
TK(L) - Shell thickness at node L; defaults to TK(I)
THETA - Element X-axis rotation

Material Properties
KXX, KYY, DENS, C, ENTH, RSVX, RSVY

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF)--

face 1 (I-J-K-L) (bottom, -Z side), face 2 (I-J-K-L) (top, +Z side),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Special Features

Requires an iterative solution for electrical-thermal coupling
Birth and death

KEYOPT(2)
Evaluation of film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS - TB|

SHELL157 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in *Table 1, "SHELL157 Element Output Definitions"*

Heat flowing out of the element is considered to be positive. The element output directions are parallel to the element coordinate system. The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used in the temperature distribution calculated for the next substep. A general description of solution output is given in *Section 2.2: Solution Output*. See The General Post-processor (POST1) in the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL157 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Convection face area	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
EF:X, Y, SUM	Component electric fields and vector sum	Y	Y
JS:X, Y	Component current densities	Y	Y
JSSUM	Component current density vector sum	Y	-
JHEAT:	Joule heat generation per unit volume	Y	Y
FACE	Face label	*1*	*1*
AREA	Face area	*1*	*1*
NODES	Face nodes	*1*	*1*
HFILM	Film coefficient	*1*	*1*
TAVG	Average face temperature	*1*	*1*
TBULK	Fluid bulk temperature	*1*	-
HEAT RATE	Heat flow rate across face by convection	*1*	*1*
HFAVG	Average film coefficient of the face	-	*1*

Name	Definition	O	R
TBAVG	Average face bulk temperature	-	*1*
HFLXAVG	Heat flow rate across face caused by input heat flux	-	*1*
HEAT RATE/AREA	Heat flow rate/area across face by convection	*1*	-
HEAT FLUX	Heat flux at each node of face	*1*	-

1. If a surface load is input
2. Available only at centroid as a ***GET** item.

Table 2, "SHELL157 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SHELL157 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL157 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

Table 2 SHELL157 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FACE 1 (BOT)	FACE 2 (TOP)	FACE 3 (J-I)	FACE 4 (K-J)	FACE 5 (L-K)	FACE 6 (I-L)
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SHELL157 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the elements are not numbered properly. The element must not taper down to a zero thickness at any corner. A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as for melting) within a coarse grid. If a current is specified at the same node that a voltage is specified, the current is ignored. The electrical and the thermal solutions are coupled through an iterative procedure.
- No conversion is included between electrical heat units and mechanical heat units. The resistivity may be divided by a conversion factor, such as 3.415 BTU/Hr per Watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

SHELL157 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS Emag

- This element has only electric field capability, and does not have thermal capability.
- The element may only be used in a steady-state electric analysis.
- The only valid degree of freedom is VOLT.
- The only allowable material properties are RSVX and RSVY.
- No surface loads or body loads are applicable.
- The birth and death special feature is not allowed.

LINK160

Explicit 3-D Spar (or Truss)

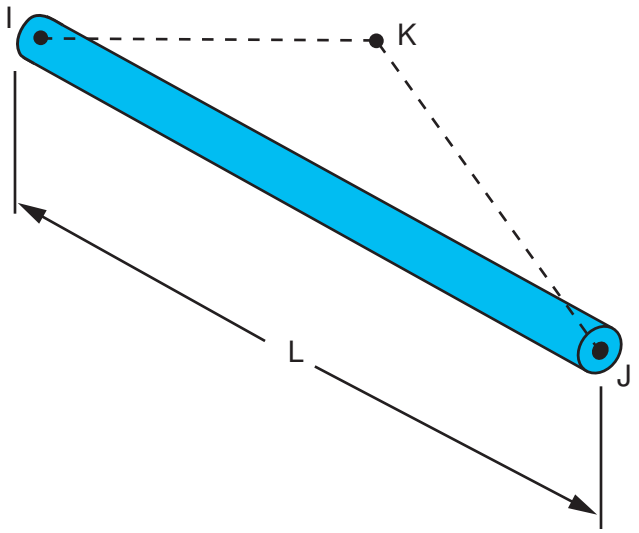
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

LINK160 Element Description

LINK160 has three degrees of freedom at each node and carries an axial force.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 LINK160 Geometry



LINK160 Input Data

The geometry and node locations are shown in *Figure 1, "LINK160 Geometry"*. Node K determines the initial orientation of the cross section. For this element, you can choose three materials: isotropic elastic, plastic kinematic, and bilinear kinematic.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the length of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element.

Use the **EDLOAD** command to apply nodal loads (displacements, forces, etc.). Also use **EDLOAD** to apply loads on rigid bodies. For more information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*.

A summary of the element input is given in *LINK160 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

LINK160 Input Summary

Nodes
I, J, K (K is the orientation node)

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamics analyses, V (X, Y, Z) refers to nodal velocity, and A (X, Y, Z) refers to nodal acceleration. Although V (X, Y, Z) and A (X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants
Area - Cross-sectional area

Material Properties

EX, NUXY, DENS, DAMP (**MP** command)
RIGID (**EDMP** command)
BKIN, PLAW (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads
None

Body Loads
None

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPTS
None

LINK160 Output Data

Output data for LINK160 consists of the following: Axial force

To output the data, you must use the **ETABLE** command. For the ITEM label, specify SMISC. For the COMP label, specify 1 for axial force. Then, you can use the **PRETAB** command to print the output data.

LINK160 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends with uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The displacement shape function implies a uniform stress in the spar.

LINK160 Product Restrictions

There are no product-specific restrictions for this element.

BEAM161

Explicit 3-D Beam

<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

BEAM161 Element Description

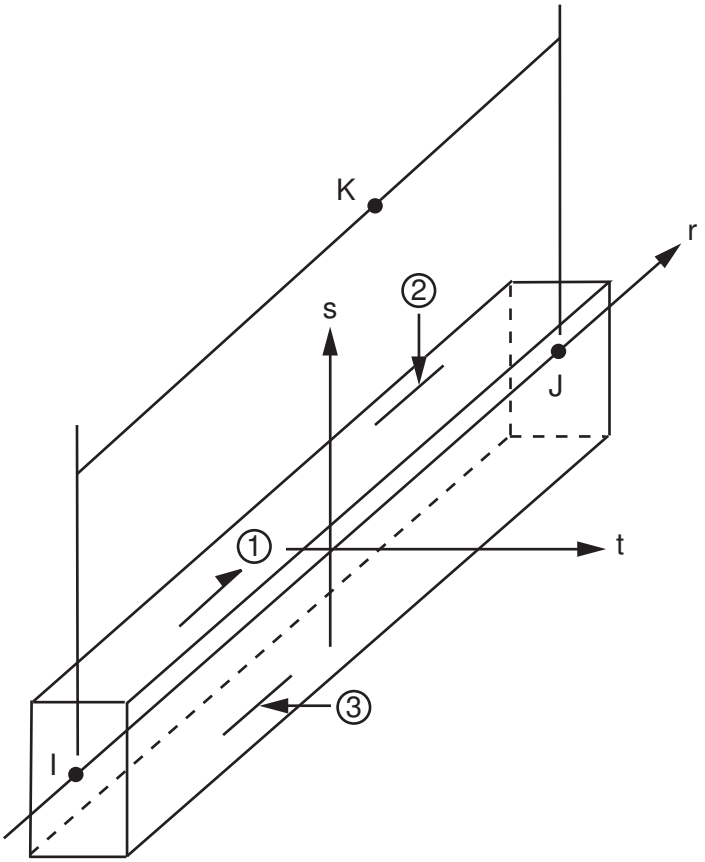
BEAM161 has several characteristics:

- It is incrementally objective (rigid body rotations do not generate strains), allowing for the treatment of finite strains that occur in many practical applications.
- It is simple for computational efficiency and robustness.
- It is compatible with the brick elements.
- It includes finite transverse shear strains. However, the added computations needed to retain this strain component, compared to those for the assumption of no transverse shear strain, are significant.

The Belytschko beam element formulation (KEYOPT(1) = 2, 4, 5) is part of a family of structural finite elements that use a "co-rotational technique" for treating large rotation.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 BEAM161 Geometry



BEAM161 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "BEAM161 Geometry"*. Node K determines the initial orientation of the cross section.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the centroidal line of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element. (For information about orientation nodes and beam meshing, see Meshing Your Solid Model in the *Modeling and Meshing Guide*.)

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "BEAM161 Geometry"*. Note, however, that pressure *is actually a traction load applied to the center line of the element*. Use the **EDLOAD** command to apply the pressure load, and input the pressure as a force per unit length value. Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

You can choose from the following materials when working with BEAM161, with the restrictions as noted:

- Isotropic Elastic
- Bilinear Kinematic (Except KEYOPT(1) = 2)
- Plastic Kinematic (Except KEYOPT(1) = 2)
- Viscoelastic (KEYOPT(1) = 1 only)
- Power Law Plasticity (KEYOPT(1) = 1 only)
- Piecewise Linear Plasticity (KEYOPT(1) = 1 only)

KEYOPT(1) allows you to specify one of four element formulations for BEAM161 (see *BEAM161 Input Summary*). For details of real constants to be specified for each element formulation, see *Table 1, "BEAM161 Real Constants"*.

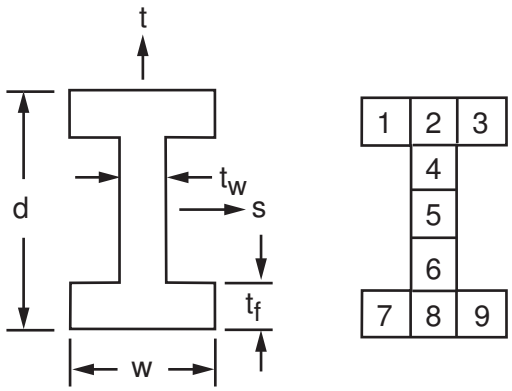
KEYOPT(2) is valid only with rectangular element formulations (KEYOPT(1) = 0, 1, 4).

The following illustrations show the valid standard beam cross sections when KEYOPT(4)>0, and KEYOPT(5) = 2 (standard beam cross section).

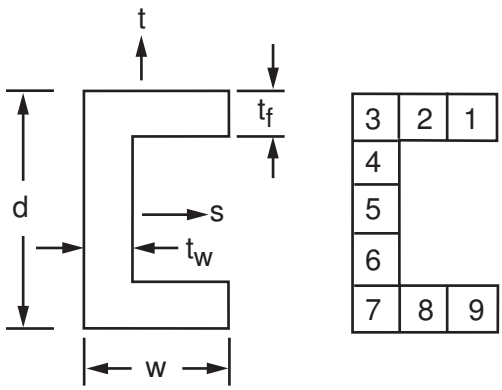
Figure 2 Standard Beam Cross Sections

w = flange width
t_f = flange thickness
d = depth
t_w = web thickness
s_{ref} = location of reference surface normal to s, Hughes-Liu beam only
t_{ref} = location of reference surface normal to t, Hughes-Liu beam only

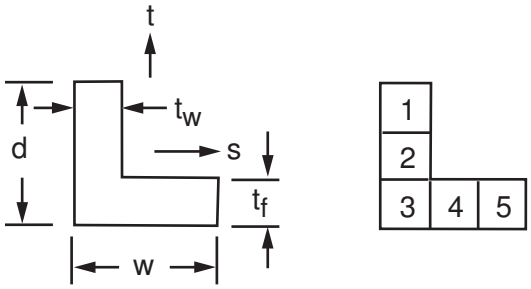
Type 1: W-section



Type 2: C-section



Type 3: Angle section



Type 4: T-section

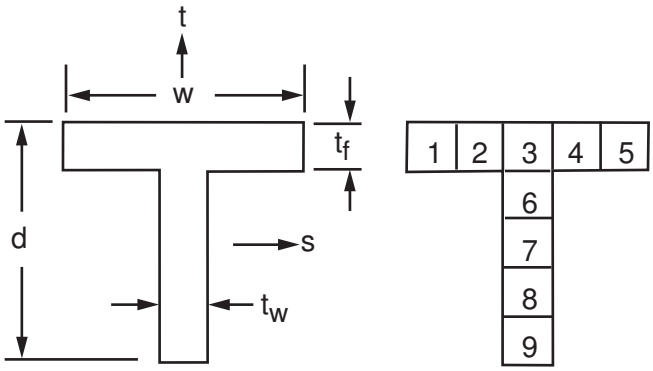
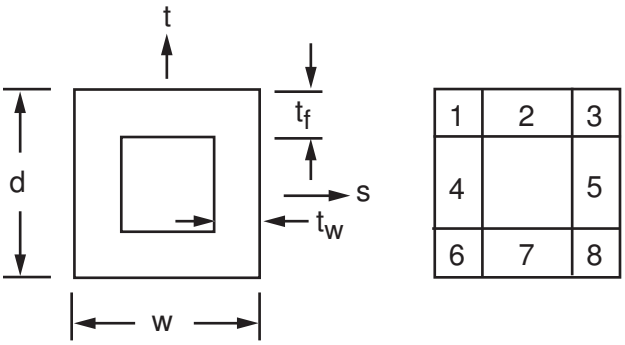
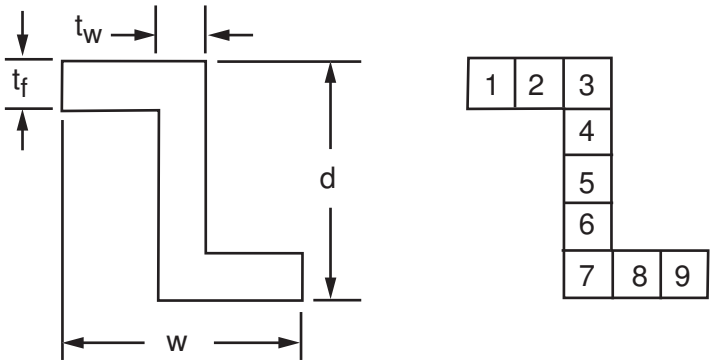


Figure 3 Standard Beam Cross Sections (continued)

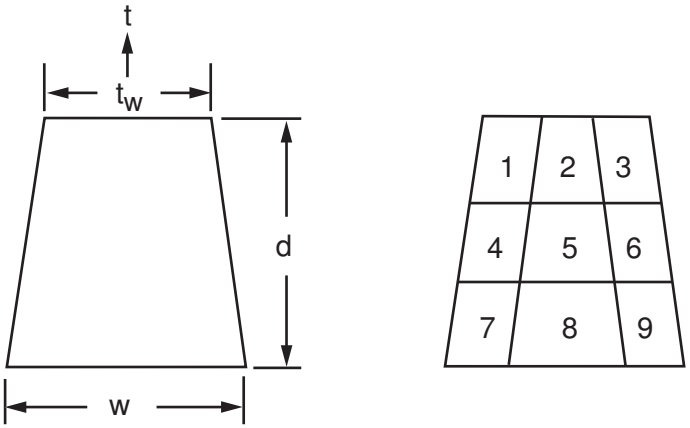
Type 5: Rectangular tubing



Type 6: Z-section



Type 7: Trapezoidal section



KEYOPT(5) is not valid when KEYOPT(1) = 2.

A summary of the element input is given in *BEAM161 Input Summary*. Additional information about real constants for this element is provided in *Table 1, "BEAM161 Real Constants"*. For more information about this element, see the *ANSYS LS-DYNA User's Guide*.

BEAM161 Input Summary

Nodes
I, J, K (K is the orientation node)

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ

Note

For explicit dynamics analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for post-processing.

Real Constants
See *Table 1, "BEAM161 Real Constants"* for a description of the real constants.

Material Properties

EX, NUXY, DENS, DAMP (**MP** command)
RIGID (KEYOPT(1) = 1,2) (**EDMP** command)
BKIN, EVISC, PLAW (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads

Pressure --
face 1 (I-J) (+r tangential direction), face 2 (I-J) (-s normal direction), face 3 (I) (-t normal direction)

Body Loads
None

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Element formulation:

0, 1 --
Hughes-Liu with cross section integration (default)

2 --
Belytschko-Schwer resultant beam (resultant)

4 --
Belytschko-Schwer full cross section integration

5 --
Belytschko-Schwer circular beam with cross section integration

KEYOPT(2)
Quadrature rule:

1 --
One integration point

0, 2 --
2 x 2 Gauss quadrature (default)

3 --
3 x 3 Gauss quadrature

4 --
3 x 3 Lobatto quadrature

5 --
4 x 4 Gauss quadrature

Note

KEYOPT(2) is valid only with rectangular element formulations (KEYOPT(1) = 0, 1, 4).

KEYOPT(4)
Integration rule for section:

0 --
Standard integration option

n --
User-defined integration rule ID (valid range: 1 to 9999)

KEYOPT(5)
Cross section type:

0 --
Rectangular cross section

1 --
Circular cross section

2 --
Arbitrary cross section (user defined integration rule) or standard beam cross section, if KEYOPT (4) > 0.

Table 1 BEAM161 Real Constants

No.	Name	Description	Use if...
1	SHRF	Shear factor. Default = 1.0 Recommended for rectangular sections = 5/6.	KEYOPT (1) = 0,1, 4, or 5
2	TS1	Beam thickness in s direction at node 1; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
3	TS2	Beam thickness in s direction at node 2; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
4	TT1	Beam thickness in t direction at node 1; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
5	TT2	Beam thickness in t direction at node 2; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
2	DS1	Beam outer diameter at node 1[1]	KEYOPT (1) = 0, 1, or 5

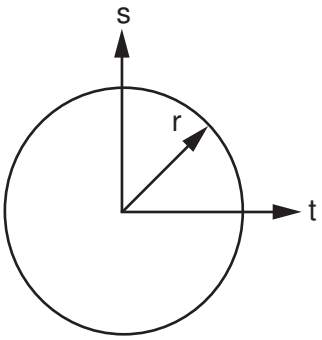
No.	Name	Description	Use if...
			KEYOPT (4) = 0 KEYOPT (5) = 1
3	DS2	Beam outer diameter at node 2[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
4	DT1	Beam inner diameter at node 1[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
5	DT2	Beam inner diameter at node 2[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
6	NSLOC	Location of reference surface normal to s-axis = 1 side at s = 1 = 0 center = -1 side at s = -1	KEYOPT (1) = 0, 1, 4, or 5 KEYOPT (4) = 0
7	NTLOC	Location of reference surface normal to t-axis = 1 side at t = 1 = 0 center = -1 side at t = -1	KEYOPT (1) = 0, 1, 4, or 5 KEYOPT (4) = 0
8	A	Cross sectional area See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (4) = 0 KEYOPT (1) = 2
9	ISS	Moment of inertia about s-axis See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (4) = 0 KEYOPT (1) = 2
10	ITT	Moment of inertia about t-axis See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (4) = 0 KEYOPT (1) = 2
11	IRR	Polar moment of inertia See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (4) = 0 KEYOPT (1) = 2
12	SA	Shear area	KEYOPT (4) = 0

No.	Name	Description	Use if...
		See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (1) = 2
13	NIP	Number of integration points See *Figure 6, "Definition of Integration Points for User Defined Integration Rule"*	KEYOPT (4) > 0 and KEYOPT (5) = 2
14	RA	Relative area of cross section; that is, the actual cross-sectional area divided by the area defined by the product of the specified thickness in the s direction and the thickness in the t direction. See *Figure 5, "Definition of Relative Area for User Defined Integration Rule"*.	KEYOPT (4) > 0 and KEYOPT (5) = 2
15	ICST	Standard cross section type. **Note** If this type is nonzero, then NIP and RA should be zero. Cross section types are: 1 - W-section 2 - C-section 3 - Angle section 4 - T-section 5 - Rectangular tubing 6 - Z-section 7 - Trapezoidal section See *Figure 2, "Standard Beam Cross Sections"*, *Figure 3, "Standard Beam Cross Sections (continued)"*.	KEYOPT (4) > 0[2] and KEYOPT (5) = 2 (standard cross section only)
16	W	Flange width	ICST > 0, and NIP = RA = 0
17	TF	Flange thickness	ICST > 0, and NIP = RA = 0
18	D	Depth	ICST > 0, and NIP = RA = 0
19	TW	Web thickness	ICST > 0, and NIP = RA = 0
20	SREF	Location of reference surface normal to s **Note** If KEYOPT (1) = 1 only	ICST > 0, and NIP = RA = 0
21	TREF	Location of reference surface normal to t **Note** If KEYOPT (1) = 1 only	ICST > 0, and NIP = RA = 0

No.	Name	Description	Use if...
22, 25, 28, ...79	S(i)	s coordinate of integration point i = 1, NIP (NIP = 20 max)[3]	KEYOPT (4) > 0 KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0
23, 26, 29, ...80	T(i)	t coordinate of integration point i = 1, NIP (NIP = 20 max)[3]	KEYOPT (4) > 0 KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0
24, 27, 30, ...81	WF(i)	Weighting factor; that is, the area associated with the integration point divided by the actual cross-section area. i = 1, NIP (NIP = 20 max)[3] See *Figure 4, "Properties of Beam Cross Sections for Several Common Cross Sections"*	KEYOPT (4) > 0 KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0

1. DS1, DS2, DT1, and DT2 are used only if KEYOPT (5) = 1. If KEYOPT (5) = 0 or 2, then use TS1, TS2, TT1, and TT2.
2. For KEYOPT (5) = 2, standard cross-section type, the integration point ID (KEYOPT (4) > 0) is not used since NIP = RA = 0. However, you must provide this input in any case.
3. Specify S(i), T(i), and WF(i) for each integration point. For example, for 20 integration points, specify S(1), T(1), WF(1), S(2), T(2), WF(2), ... S(20), T(20), WF(20).

Figure 4 Properties of Beam Cross Sections for Several Common Cross Sections

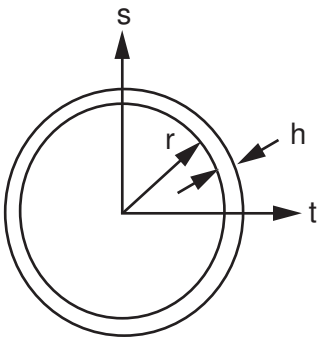


$$I_{tt} = I_{ss} = \frac{\pi r^4}{2}$$

$$J = \frac{\pi r^4}{2}$$

$$f_{tt} = f_{ss} = \frac{10}{9}$$

$$A = \pi r^2$$

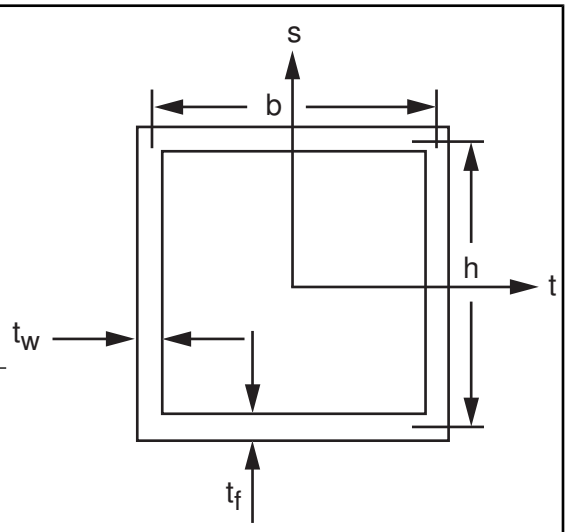


$$I_{tt} = I_{ss} \cong \pi r^3 h$$

$$J \cong 2\pi r^3 h$$

$$f_{tt} = f_{ss} = 2$$

$$A \cong 2\pi r h$$



$$I_{tt} \cong \left(\frac{h^2}{6}\right)(ht_w + 3bt_f)$$

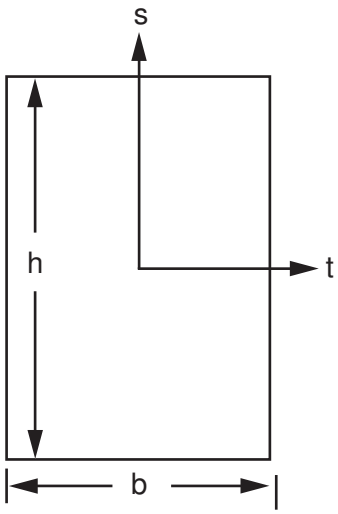
$$I_{ss} \cong \left(\frac{b^2}{6}\right)(bt_f + 3ht_w)$$

$$J \cong \frac{2b^2h^2(t_w t_f)}{(bt_w + ht_f)}$$

$$f_{tt} = \frac{A}{[2(b + t_w)t_f]}$$

$$f_{ss} = \frac{A}{[2(h + t_f)t_w]}$$

$$A = 2(bt_f + ht_w)$$



$$I_{tt} = \frac{bh^3}{12}$$

$$I_{ss} = \frac{b^3 h}{12}$$

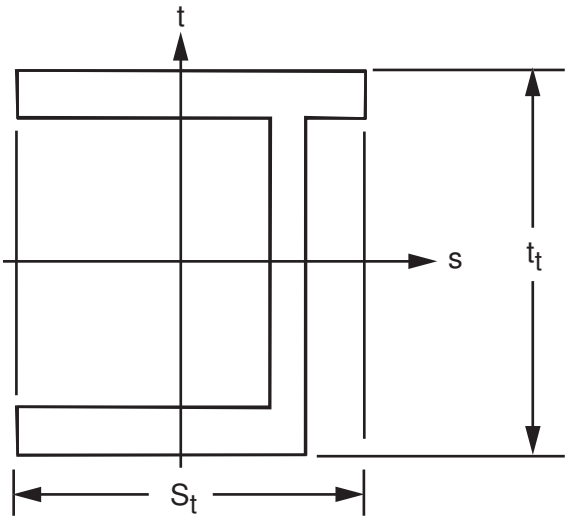
$$J \cong \left[\frac{1}{3} - 0.21\left(\frac{b}{h}\right)\left(1 - \frac{b^4}{12h^4}\right)\right]hb^3$$

$$f_{tt} = f_{ss} = \frac{6}{5}$$

$$A = bh$$

$$\text{Shear area} = \frac{A}{f} = \mu A$$

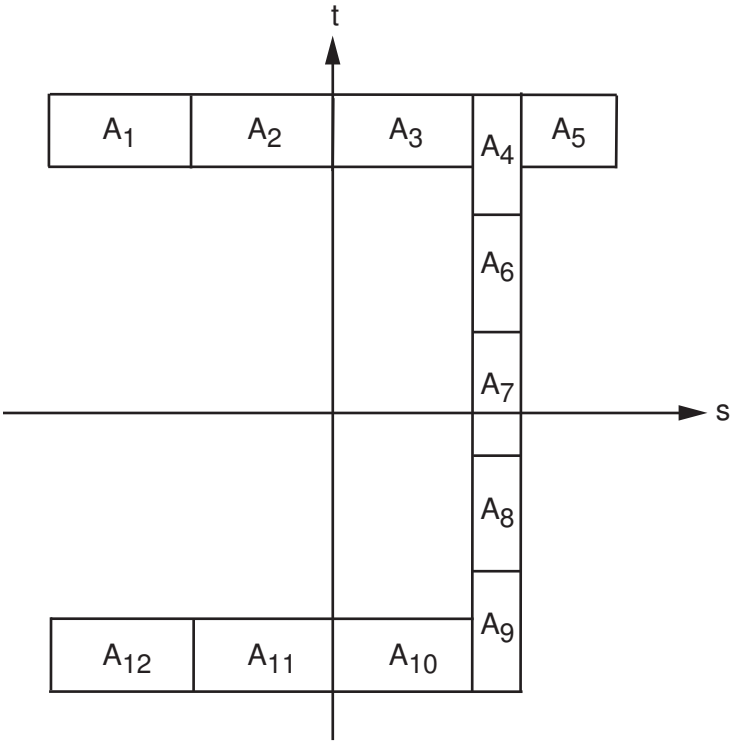
Figure 5 Definition of Relative Area for User Defined Integration Rule



$$\text{Relative area} = \frac{A}{S_t\, t_t}$$

t_t = input as TT1 or TT2 on R command
S_t = input as TS1 or TS2 on R command

Figure 6 Definition of Integration Points for User Defined Integration Rule



BEAM161 Output Data

To store output data for this element, you first need to specify the number of integration points for which you want output data. Use the **EDINT**,,*BEAMIP* command during the solution phase of your analysis to specify the number of integration points. By default, output is written for 4 integration points. For the resultant beam for-

mulation (KEYOPT(1) = 2), there is no stress output (regardless of the *BEAMIP* setting). If you set *BEAMIP* = 0, no stress output is written for any of the beam elements. In this case, the beams will not appear in any POST1 plots because the program assumes they are failed elements.

To display the data for BEAM161, you must use the **ETABLE** command. Then, you can use the **PRETAB** command to print the output data. The **RSYS** command has no effect when postprocessing output for this element.

The following items are available on the results file.

Table 2 BEAM161 Element Output Definitions

Name	Definition
S (r, rs, rt)	Stresses
EPEQ	Equivalent plastic strain
EPTO	Axial strain
MFORr	Member force in the element coordinate system, r direction
N (s, t)	Out-of-plane (s, t) shear
M (s, t)	Element (s, t) moments
TORQ	Torsional resultant

For each of these output data, one set of values, given at the centroid, is output for the entire beam.

Table 3, "BEAM161 Item and Sequence Numbers" lists output available through the **ETABLE** and **ESOL** commands using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "BEAM161 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "BEAM161 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** or **ESOL** command

E
: sequence number for single-valued or constant element data

1st IP
: sequence number for the first integration point

*n*th IP
: sequence number for the *n*th integration point as defined by the **EDINT** command.

Table 3 BEAM161 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	1st IP	*n*th IP
MFORr	SMISC	1	-	-
Ns	SMISC	2	-	-
Nt	SMISC	3	-	-
Ms	SMISC	4	-	-
Mt	SMISC	5	-	-
TORQ	SMISC	6	-	-
Sr	LS	-	1	5 x (*n*-1) +1

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	1st IP	*n*th IP
Srs	LS	-	2	5 x (*n*-1) +2
Srt	LS	-	3	5 x (*n*-1) +3
EPEQ	LS	-	4	5 x (*n*-1) +4
EPTO	LS	-	5	5 x (*n*-1) +5

1. In this table, *n* refers to the current integration point for which you want output data.

BEAM161 Assumptions and Restrictions

- The beam must not have a zero length.
- The beam can have any open or single-cell closed cross-sectional shape for which the area and moments of inertia are nonzero.
- Warping torsion is assumed negligible and the warping moment of inertia is not used in the stiffness computation.
- Warping of the cross section is unconstrained and is the same for all cross-sections; therefore, the torsional rotation of the cross-section is assumed to vary linearly along the length. However, warping is not applicable to the resultant beam formulation (KEYOPT(1) = 2).

BEAM161 Product Restrictions

There are no product-specific restrictions for this element.

PLANE162

Explicit 2-D Structural Solid

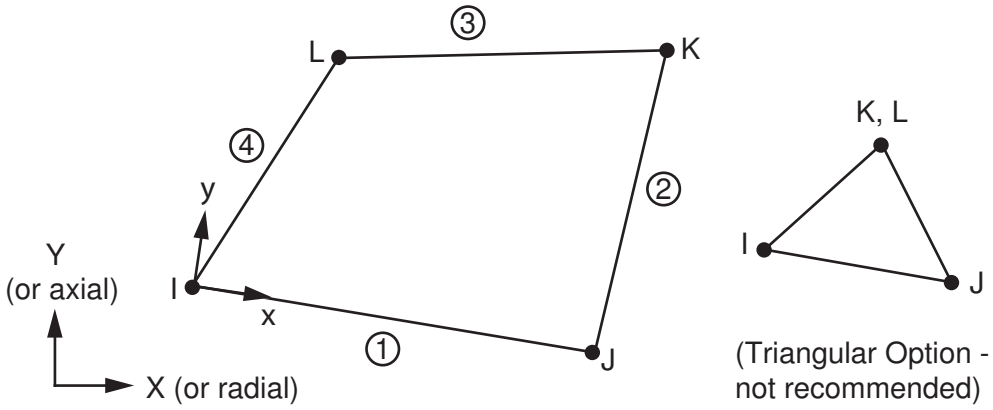
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

PLANE162 Element Description

PLANE162 is used for modeling 2-D solid structures in ANSYS LS-DYNA. The element can be used either as a planer or as an axisymmetric element. The element is defined by four nodes having six degrees of freedom at each node: translations, velocities, and accelerations in the nodal x and y directions. A three-node triangle option is also available, but not recommended.

The element is used in explicit dynamic analyses only. When using this element, the model must only contain PLANE162 elements - you cannot mix 2-D and 3-D explicit elements in the same model. Furthermore, all PLANE162 elements in the model must be the same type (plane stress, plane strain, or axisymmetric). Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 PLANE162 Geometry



PLANE162 Input Data

The geometry, node locations, and coordinate system for this element are shown in *Figure 1, "PLANE162 Geometry"*. Use KEYOPT(3) to specify whether the element is a plane stress, plane strain, or axisymmetric element. For the axisymmetric option (KEYOPT(3) = 1), you may also use KEYOPT(2) to specify either area or volume weighted axisymmetric elements.

KEYOPT(5) defines the element continuum treatment. Two different formulations are available: Lagrangian (default) and Arbitrary Lagrangian-Eulerian (ALE). In addition to setting KEYOPT(5) = 1, you must also set appropriate parameters on the **EDALE** and **EDGCALE** commands in order for the ALE formulation to take affect. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information.

Use the **EDLOAD** command to apply nodal loads and other types of loads described below. For detailed information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*. Note that when the axisymmetric option (KEYOPT(3) = 1) is selected and KEYOPT(2) = 0 (area weighted option), nodal loads should be input per unit length of circumference. Likewise, when KEYOPT(3) = 1 and KEYOPT(2) = 1 (volume weighted option), nodal loads should be input per radian. Other aspects of axisymmetric elements are covered in *Section 2.12: Axisymmetric Elements*. Pressures are always on a 360° basic, irrespective of the KEYOPT(2) setting.

Pressures can be input as surface loads on the element faces (edges) as shown by the circled numbers in *Figure 1, "PLANE162 Geometry"*. Positive normal pressures act into the element.

Other loads that can be applied using the **EDLOAD** command include base accelerations and angular velocities in the x and y directions, and displacements and forces on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

The material models available to use with this element will depend on the KEYOPT(3) setting. KEYOPT(3) controls whether the element is a plane stress, plane strain, or axisymmetric element. For all three of these options (KEYOPT(3) = 0, 1, or 2), you can choose the following materials:

- Isotropic Elastic
- Orthotropic Elastic
- Elastic Fluid
- Viscoelastic
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Bilinear Kinematic
- Plastic Kinematic
- Power Law Plasticity
- Rate Sensitive Power Law Plasticity
- Strain Rate Dependent Plasticity
- Piecewise Linear Plasticity
- Composite Damage
- Johnson-Cook Plasticity
- Bamman

For the plane stress option (KEYOPT(3) = 0), you can also choose the following materials:

- 3-Parameter Barlat Plasticity
- Barlat Anisotropic Plasticity
- Transversely Anisotropic Elastic Plastic
- Transversely Anisotropic FLD

For the axisymmetric and plane strain options (KEYOPT(3) = 1 or 2), you can also choose the following materials:

- Blatz-Ko Rubber
- Mooney-Rivlin Rubber
- Elastic-Plastic Hydrodynamic
- Closed Cell Foam
- Low Density Foam
- Crushable Foam
- Honeycomb
- Null

- Zerilli-Armstrong
- Steinberg

PLANE162 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, VX, VY, AX, AY

Note

For explicit dynamic analyses, V(X, Y) refers to nodal velocity, and A(X, Y) refers to nodal acceleration. Although V(X, Y) and A(X, Y) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants
None

Material Properties

EX, EY, PRXY or NUXY, ALPX (or CTEX *or* THSX),
DENS, GXY, DAMP (**MP** command),
RIGID, HGLS, ORTHO, FLUID (**EDMP** command),
BKIN, BISO, MOONEY, EVISC, PLAW, FOAM, HONEY, COMP, EOS (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads
Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(2)
Weighting option (used for axisymmetric elements, KEYOPT(3) = 1):

0 --
Area weighted axisymmetric element

1 --
Volume weighted axisymmetric element

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

KEYOPT(5)
Element continuum treatment:

0 --
Lagrangian (default)

1 --
ALE (Arbitrary Lagrangian-Eulerian)

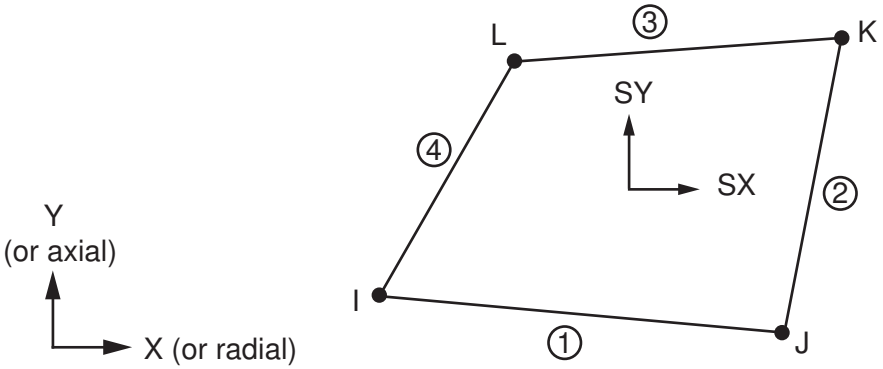
PLANE162 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE162 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE162 Stress Output"*. The element stresses are output in terms of the global Cartesian coordinate system by default. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE162 Stress Output



You can rotate stress results for PLANE162 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 1 PLANE162 Element Output Definitions

Name	Definition
S(X, Y, XY)	Stresses
S(1, 2, 3)	Principal stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, XY)	Total strains
EPTO(1, 2, 3)	Total principle strains
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain
EPEL(X, Y, XY)	Elastic strains
EPEL(1, 2, 3)	Principle elastic strains

Name	Definition
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain

Note

Stress and total strain are always available. Some components of stress and strain (for example, yz and zx components) are always zero. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

Table 2, "PLANE162 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE162 Item and Sequence Numbers"*:

Name
output quantity

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 PLANE162 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
EPEQ (equivalent plastic strain)	NMISC	1

PLANE162 Assumptions and Restrictions

- The area of the element must be nonzero.
- The element must lie in the global X-Y plane as shown in *Figure 1, "PLANE162 Geometry"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).

PLANE162 Product Restrictions

There are no product-specific restrictions for this element.

SHELL163

Explicit Thin Structural Shell

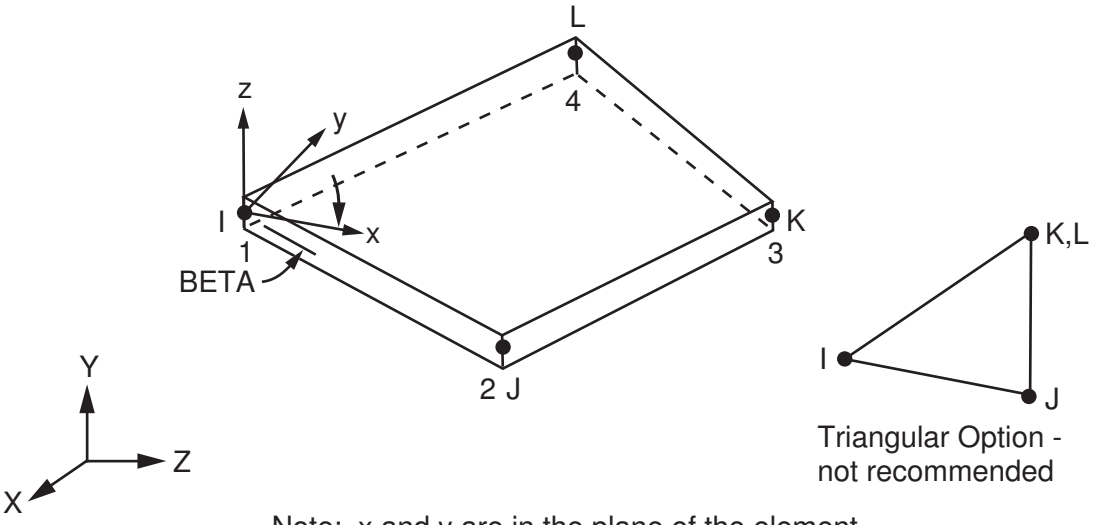
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

SHELL163 Element Description

SHELL163 is a 4-node element with both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has 12 degrees of freedom at each node: translations, accelerations, and velocities in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 SHELL163 Geometry



Note: x and y are in the plane of the element

SHELL163 Input Data

The following real constants are provided for SHELL163. SHRF is the shear factor. NIP is the number of integration points through the thickness of the element, up to a maximum of 100. If NIP is input as 0 or blank, ANSYS defaults the value to 2. T1 - T4 indicate the shell thickness at each of the 4 nodes. NLOC specifies the location of the reference surface for KEYOPT(1) = 1, 6, or 7. The reference surface is used in the formulation of the element stiffness matrix. (NLOC does not define the location of the contact surface.) If you set NLOC = 1 or -1 (top or bottom surface), you must set *SHNU* = -2 on the **EDSHELL** command.

ESOP is the option for the spacing of integration points, and can be either 0 or 1. ESOP is used only if KEYOPT(4) > 0. If you set ESOP = 0, you must define real constants S(i), and WF(i) to define the integration point locations. If KEYOPT(3) = 1, then you must also define BETA(i) and MAT(i) for each integration point. Set ESOP = 1 if the integration points are equally spaced through the thickness such that the shell is subdivided into NIP layers of equal thickness (up to 100 layers).

The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the 4 nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

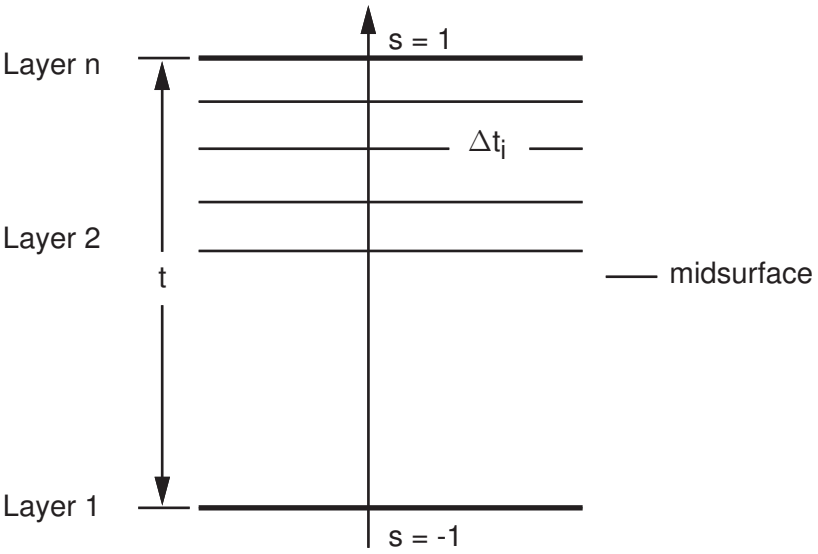
If you set ESOP = 0 and define the integration points using S(i), and WF(i), and possibly BETA(i) and MAT(i), note the following:

- If KEYOPT(1) = 1, 6, 7, or 11, then the thicknesses you define will remain defined through the results determination.
- If KEYOPT(1) = 2, 3, 4, 5, 8, 9, 10, or 12, then the ANSYS program overrides any thickness values you specify and averages the thicknesses for the results determination.

S(i) is the relative coordinate of the integration point and must be within the range -1 to 1. WF(i) is the weighting factor for the i-th integration point. It is calculated by dividing the thickness associated with the integration point by the actual shell thickness (that is, $\Delta t_i/t$); see *Figure 2, "Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule"*. In the user defined shell integration rule, the ordering of the integration points is arbitrary. If using these real constants to define integration points, then S(i) and WF(i) must both be specified for each integration point (maximum of 100). BETA(i) is the material angle (in degrees) at the i-th integration point and must be specified for each integration point. The material model (BKIN, MKIN, MISO, etc.) is not allowed to change within an element, although the material properties (EX, NUXY, etc.), as defined per MAT(i), can change. However, the density may not vary through the thickness of the shell element. If more than one material is used, and the densities vary between materials, the density of the material of the first layer will be used for the entire element.

If KEYOPT(4) = 0, the integration rule is defined by KEYOPT(2). The Gauss rule (KEYOPT(2) = 0) is valid for up to five layers (integration points). The trapezoidal rule (KEYOPT(2) = 1) allows up to 100 layers, but is not recommended for less than 20 layers, especially if bending is involved.

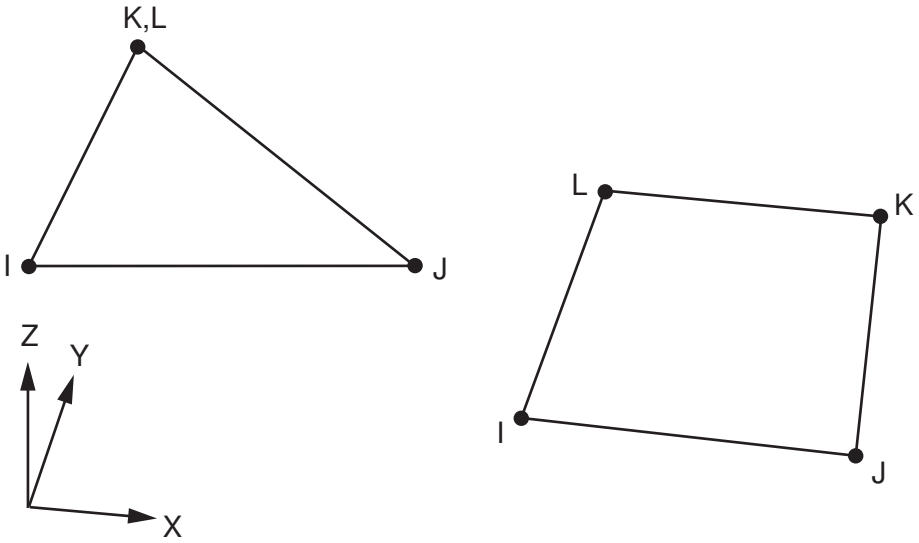
Figure 2 Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule



Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element midsurfaces. Positive normal pressures act into the element (that is, positive pressure acts in the negative z direction). Note, however, that pressure *is actually applied to the midsurface*. See *Figure 3, "Nodal Numbering for Pressure Loads (Positive Pressure Acts in Negative Z Direction)"*.

Figure 3 Nodal Numbering for Pressure Loads (Positive Pressure Acts in Negative Z Direction)



Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component. Each node in the component will have the specified load.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

For this element, you can choose from the following materials:

- Isotropic Elastic
- Orthotropic Elastic
- Bilinear Kinematic
- Plastic Kinematic
- Blatz-Ko Rubber
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage
- Piecewise Linear Plasticity
- Modified Piecewise Linear Plasticity
- Mooney-Rivlin Rubber
- Barlat Anisotropic Plasticity
- 3-Parameter Barlat Plasticity
- Transversely Anisotropic Elastic Plastic

- Rate Sensitive Power Law Plasticity
- Transversely Anisotropic FLD
- Elastic Viscoplastic Thermal
- Johnson-Cook Plasticity
- Bamman

The orthotropic elastic material model does not accept integration point angles (BETA(i)). Therefore, to model a composite material, you need to use the composite damage material model. If you do not wish to use the damage features of this material model, just set the required strength values to zero.

KEYOPT(1) allows you to specify 1 of 12 element formulations for SHELL163 (see *SHELL163 Input Summary*). A brief description about each element formulation follows:

The Hughes-Liu element formulation (KEYOPT(1) = 1) is based on a degenerated continuum formulation. This formulation results in substantially large computational costs, but it is effective when very large deformations are expected. This formulation treats warped configurations accurately but does not pass the patch test. It uses one-point quadrature with the same hourglass control as the Belytschko-Tsay.

The Belytschko-Tsay (default) element formulation (KEYOPT(1) = 0 or 2) is the fastest of the explicit dynamics shells. It is based on the Mindlin-Reissner assumption, so transverse shear is included. It does not treat warped configurations accurately, so it should not be used in coarse mesh models. One-point quadrature is used with hourglass control. A default value is set for the hourglass parameter. When hourglassing appears, you should increase this parameter to avoid hourglassing. It does not pass the patch test.

The BCIZ Triangular Shell element formulation (KEYOPT(1) = 3) is based on a Kirchhoff plate theory and uses cubic velocity fields. Three sets of quadrature points are used in each element, so it is relatively slow. It passes the patch test only when the mesh is generated from three sets of parallel lines.

The C^0 Triangular Shell element formulation (KEYOPT(1) = 4) is based on a Mindlin-Reissner plate theory and uses linear velocity fields. One quadrature point is used in the element formulation. This formulation is rather stiff, so it should not be used for constructing an entire mesh, only to transition between meshes.

The Belytschko-Tsay membrane element formulation (KEYOPT(1) = 5) is the same as the Belytschko-Tsay but with no bending stiffness.

The S/R Hughes-Liu element formulation (KEYOPT(1) = 6) is the same as the Hughes-Liu, but instead of using one-point quadrature with hourglass control, this formulation uses selective reduced integration. This increases the cost by a factor of 3 to 4, but avoids certain hourglass modes; certain bending hourglass modes are still possible.

The S/R corotational Hughes-Liu element formulation (KEYOPT(1) = 7) is the same as the S/R Hughes-Liu except it uses the corotational system.

The Belytschko-Leviathan shell formulation (KEYOPT(1) = 8) is similar to the Belytschko-Wong-Chiang with one-point quadrature but it uses physical hourglass control, thus no user-set hourglass control parameters need to be set.

The fully-integrated Belytschko-Tsay membrane element formulation (KEYOPT(1) = 9) is the same as the Belytschko-Tsay membrane except is uses a 2 x 2 quadrature instead of a one-point quadrature. This formulation is more robust for warped configurations.

The Belytschko-Wong-Chiang formulation (KEYOPT(1) = 10) is the same as the Belytschko-Tsay except the shortcomings in warped configuration are avoided. Costs about 10% more.

The fast (corotational) Hughes-Liu formulation (KEYOPT(1) = 11) is the same as the Hughes-Liu except this formulation uses the corotational system.

The fully-integrated Belytschko-Tsay shell element formulation (KEYOPT(1) = 12) uses a 2 x 2 quadrature in the shell plane and is about 2.5 times slower than KEYOPT(1) = 2. It is useful in overcoming hourglass modes. The shear locking is remedied by introducing an assumed strain for the transverse shear.

Of the twelve shell element formulations, only KEYOPT(1) = 1, 2, 6, 7, 8, 9, 10, 11, and 12 are valid for an explicit-to-implicit sequential solution. For metal forming analyses, KEYOPT(1) = 10 and 12 are recommended in order to properly account for warping.

When the Mooney-Rivlin Rubber material model is used with SHELL163 elements, the LS-DYNA code will automatically use a total Lagrangian modification of the Belytschko-Tsay formulation instead of using the formulation you specify via KEYOPT(1). This program-chosen formulation is required to address the special needs of the hyperelastic material.

A summary of the element input is given in *SHELL163 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL163 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for post processing.

Real Constants

SHRF, NIP, T1, T2, T3, T4,
NLOC, ESOP, BETA(i), S(i), WF(i), MAT(i)
(BETA(i), S(i), WF(i), MAT(i) may repeat for each integration point, depending on the keyoption settings.)
Specify NLOC only if KEYOPT(1) = 1, 6, 7, or 11.
See *Table 1, "SHELL163 Real Constants"* for descriptions of the real constants.

Material Properties

EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ, ALPX (or CTEX *or* THSX), GXY, GYZ, GXZ,
DENS, DAMP (**MP** command)
RIGID, HGLS (except KEYOPT(1) = 3, 4, 6, 7, 9, and 12), ORTHO (**EDMP** command)
PLAW, BKIN, BISO, COMPOSITE, MOONEY, EOS (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads

Pressure (applied on midsurface)

Body Loads

Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features

This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)

Element formulation:

1 --
Hughes-Liu

0, 2 --
Belytschko-Tsay (default)

3 --
BCIZ triangular shell

4 --
C^0 triangular shell

5 --
Belytschko-Tsay membrane

6 --
S/R Hughes-Liu

7 --
S/R corotational Hughes-Liu

8 --
Belytschko-Levithan shell

9 --
Fully integrated Belytschko-Tsay membrane

10 --
Belytschko-Wong-Chiang

11 --
Fast (corotational) Hughes-Liu

12 --
Fully integrated Belytschko-Tsay shell

KEYOPT(2)

Quadrature rule (used for standard integration rules, KEYOPT(4) = 0):

0 --
Gauss rule (up to five integration points are permitted)

1 --
Trapezoidal rule (up to 100 integration points are permitted)

KEYOPT(3)

Flag for layered composite material mode:

0 --
Non-composite material mode

1 --
Composite material mode; a material angle is defined for each through thickness integration point

KEYOPT(4)

Integration rule ID:

0 --
Standard integration option

n --

User-defined integration rule ID (valid range is 1 to 9999; if selected, it overrides the integration rule set by KEYOPT(2))

Table 1 SHELL163 Real Constants

No.	Name	Description
1	SHRF	Shear factor Suggested value: 5/6; if left blank, defaults to 1
2	NIP	Number of integration points If input as 0 or blank, defaults to 2.
3	T1	Shell thickness at node I
4	T2	Shell thickness at node J
5	T3	Shell thickness at node K
6	T4	Shell thickness at node L
7	NLOC	Location of reference surface = 1, top surface = 0, middle surface = -1, bottom surface Used only if KEYOPT(1) = 1, 6, or 7.
8	ESOP	Option for the spacing of integration points: 0 - Integration points are defined using real constants S(i) and WF(i). 1 - Integration points are equally spaced through the thickness such that the shell is subdivided into NIP layers of equal thickness.
9, 13, 17,... 405	BETA(i)	Material angle at the i-th integration point.[*1*]
10, 14, 18, ... 406	S(i)	Coordinate of integration point in the range -1 to 1. i = 1, NIP (NIP = 100 max)[*1*]
11,15, 19, ... 407	WF(i)	Weighting factor; that is, the thickness associated with the integration point divided by the actual shell thickness. i = 1, NIP (NIP = 100 max)[*1*]
12, 16, 20, ... 408	MAT(i)	Material ID for each layer. [*1*]

1. If KEYOPT(3) = 1, then BETA(i), S(i), WF(i), and MAT(i) should be specified for each integration point. For example, for 20 integration points, you would specify BETA(1), S(1), WF(1), MAT(1), BETA(2), S(2), WF(2),

MAT(2), ..., BETA(20), S(20), WF(20), MAT(20). If KEYOPT(3) = 0, then only S(i) and WF(i) need to be specified. The material used will be that specified by the **MAT** command.

SHELL163 Output Data

To store output data for this element, you must specify the number of output locations for which you want data using the **EDINT**,SHELLIP command during solution. To review the stored data for a specified layer, use the **LAYER**,NUM command. However, be aware that the output location for this data is always at the integration point. "Top" and "bottom" refer to the top or bottom integration point, which is not necessarily the top or bottom surface.

Stress data is always output from the bottom of the shell to the top. See *Figure 2, "Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule"*.

In all cases (default and otherwise), strain is always output for two layers only: Layer 1 = bottom and layer 2 = top.

The number of integration points specified by real constant NIP controls the output locations through the thickness of the shell. If NIP = SHELLIP, then each layer corresponds to an integration point, and those are the locations where you will get output data. If NIP>SHELLIP, then data is output only at the SHELLIP number of locations (first bottom layer, then layers 2 through *n* moving up from the bottom). If NIP<SHELLIP (but NIP>2), then results are output only for NIP number of layers.

By default, the number of integration points (NIP) is 2, and the number of output locations/layers (SHELLIP) is 3. In this case, stress data is output in the following order: Layer 1 = bottom, layer 2 = middle, and layer 3 = top. When SHELLIP = 3, the middle layer will be an interpolated value if NIP is an even number or an actual value at an integration point if NIP is an odd number.

If NIP = 1, the integration point is at the element midplane, and only one stress and one strain value are output.

For elements with 2 x 2 integration points in the shell plane (KEYOPT(1) = 6, 7, 9, 12), LS-DYNA performs an averaging of any data output at those points in every layer so that the output is the same for all shell formulations.

For the default **RSYS** setting, strains (EPTO) and generalized stresses (M,T,N) are output in the element coordinate system, and stresses (S) are output in the global Cartesian system for all formulations associated with SHELL163, except the Hughes-Liu formulation. Strain output (EPTO) for the Hughes-Liu formulation (KEYOPT(1) = 1) is output in the global Cartesian system.

You can rotate stress results for this element into another coordinate system using the **RSYS** command. However, **RSYS** has no effect on the stress results for composite SHELL163 elements (KEYOPT(3) = 1). In addition, **RSYS** cannot be used to rotate strain results for any of the SHELL163 element formulations.

The following items are available in the results file.

Table 2 SHELL163 Element Output Definitions

Name	Definition
S(X, Y, Z, XY, YZ, XZ)	Stresses
S(1, 2, 3)	Principle stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, Z, XY, YZ, XZ)	Total strain
EPTO(1, 2, 3)	Total principle strains

Name	Definition
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain
EPEL(X, Y, Z, XY, YZ, XZ)	Elastic strains
EPEL(1, 2, 3)	Principle elastic strains
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain
M(X, Y, XY)	Element X, Y, and XY moments
N(X. Y)	Out-of-plane X, Y shear
T(X, Y, XY)	In-plane element X, Y, and XY forces
Thick	Element thickness

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

Table 3, "SHELL163 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SHELL163 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SHELL163 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 SHELL163 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	1
MY	SMISC	2
MXY	SMISC	3
NX	SMISC	4
NY	SMISC	5
TX	SMISC	6
TY	SMISC	7
TXY	SMISC	8
EPEQ (top)[*1*]	NMISC	1
EPEQ (middle)[*1*], [*2*]	NMISC	2
EPEQ (bottom)[*1*]	NMISC	3
Thick[*1*]	NMISC	4

1. The sequence numbers for NMISC items in this table are based on the assumption that the number of integration points for output (SHELLIP on the **EDINT** command) is set to the default value of 3.
2. If the number of integration points (NIP) is even, the middle EPEQ value (NMISC,2) will be an interpolated value.

The SMISC quantities in the above table are independent of layers (that is, you will get one set of SMISC quantities output per element). However, the NMISC items are layer-dependent, and the order of the NMISC items is dependent on the SHELLIP and NIP values. The order shown in the table corresponds to the default SHELLIP value (SHELLIP = 3). If NIP > 3, it is strongly recommended that you set SHELLIP = NIP. In this case, the **ETABLE** output will go from top (NMISC,1) to bottom (NMISC,*n* where *n* is the total number of layers). If SHELLIP is not equal to NIP, the order of NMISC items will vary. Therefore, you should not use **ETABLE** to access the NMISC items when NIP > 3 and SHELLIP is not equal to NIP.

SHELL163 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- A triangular element may be formed by defining duplicate K and L node numbers as described in *Section 2.9: Triangle, Prism and Tetrahedral Elements*. In this event, the C^0 triangular shell element (KEYOPT(1) = 4) will be used.
- An assemblage of flat shell elements can produce a good approximation to a curved shell surface provided that each flat element does not extend over more than a 15° arc.

SHELL163 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS ED

- Composite material shell elements are not allowed. KEYOPT(3) defaults to 0.

SOLID164

Explicit 3-D Structural Solid

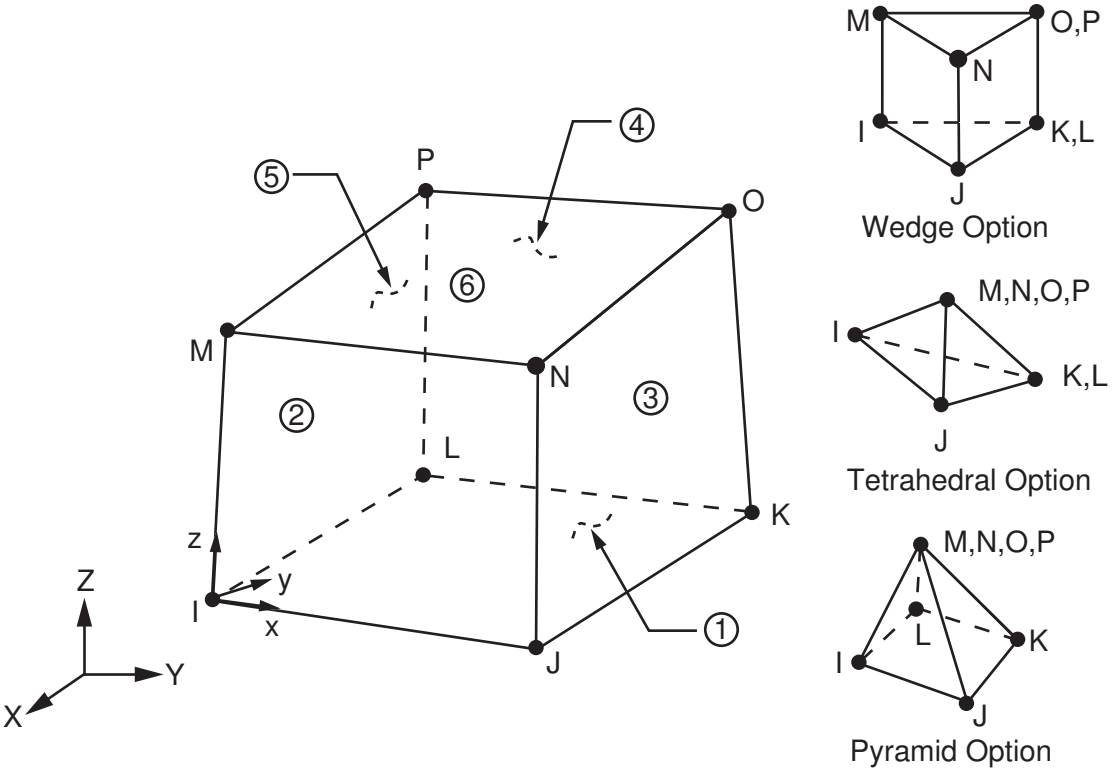
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

SOLID164 Element Description

SOLID164 is used for the 3-E modeling of solid structures. The element is defined by eight nodes having the following degrees of freedom at each node: translations, velocities, and accelerations in the nodal x, y, and z directions.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 SOLID164 Geometry



SOLID164 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID164 Geometry"*. The element is defined by eight nodes. Orthotropic material properties may be defined. Use the **EDMP** command to specify an orthotropic material and the **EDLCS** command to define the orthotropic material directions.

By default, SOLID164 uses reduced (one point) integration plus viscous hourglass control for faster element formulation. A fully integrated solid formulation (KEYOPT(1) = 2) is also available.

KEYOPT(5) defines the element continuum treatment. Two different formulations are available: Lagrangian (default) and Arbitrary Lagrangian-Eulerian (ALE). In addition to setting KEYOPT(5) = 1, you must also set appropriate parameters on the **EDALE** and **EDGCALE** commands in order for the ALE formulation to take affect. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information. Some material models

that are normally available for this element type are not supported when the ALE formulation is used. See the material list below for details.

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "SOLID164 Geometry"*. Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

For this element, you can choose from the materials listed below. The material models marked by an asterisk (*) are not supported by the ALE formulation (KEYOPT(5) = 1).

- Isotropic Elastic
- Orthotropic Elastic*
- Anisotropic Elastic*
- Bilinear Kinematic
- Plastic Kinematic
- Viscoelastic*
- Blatz-Ko Rubber*
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage*
- Concrete Damage*
- Geological Cap
- Piecewise Linear Plasticity*
- Honeycomb*
- Mooney-Rivlin Rubber*
- Barlat Anisotropic Plasticity
- Elastic-Plastic Hydrodynamic
- Rate Sensitive Power Law Plasticity
- Elastic Viscoplastic Thermal
- Closed Cell Foam*
- Low Density Foam

- Viscous Foam*
- Crushable Foam
- Johnson-Cook Plasticity
- Null
- Zerilli-Armstrong
- Bamman*
- Steinberg
- Elastic Fluid

SOLID164 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. AlthoughV (X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants
None

Material Properties

EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ, ALPX (or CTEX *or* THSX), GXY, GYZ, GXZ,
DENS, DAMP (**MP** command)

RIGID, HGLS, ORTHO, FLUID (**EDMP** command)

ANEL, MOONEY, EVISC, BISO, BKIN, PLAW, FOAM, HONEY, COMPOSITE, CONCR, GCAP, EOS (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads

Pressures --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads
Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Element formulation:

0, 1 --
Constant stress solid element (default)

2 --
Fully integrated selectively-reduced solid

KEYOPT(5)
Element continuum treatment:

0 --
Lagrangian (default)

1 --
ALE (Arbitrary Lagrangian-Eulerian)

SOLID164 Output Data

Output for SOLID164 is listed in *Table 1, "SOLID164 Element Output Definitions"*. If you issue **PRNSOL**, a single set of stress and a single set of strain values is output at all eight nodes; that is, you will get the same sets of values at each node. If you issue **PRESOL**, you will get only a single set of values at the centroid.

You can rotate stress results for SOLID164 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 1 SOLID164 Element Output Definitions

Name	Definition
S(X, Y, Z, XY, YZ, XZ)	Stresses
S(1, 2, 3)	Principal stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, Z, XY, YZ, XZ)	Total strains
EPTO(1, 2, 3)	Total principle strains
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain
EPEL(X, Y, Z, XY, YZ, XZ)	Elastic strains
EPEL(1, 2, 3)	Principle elastic strains
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

SOLID164 Assumptions and Restrictions

- Zero volume elements are not allowed.
- The element may not be twisted such that it has two separate volumes. This occurs most frequently when the element is not numbered properly.
- The element must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). A tetrahedron shape is also available.

SOLID164 Product Restrictions

There are no product-specific restrictions for this element.

COMBI165

Explicit Spring-Damper

<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

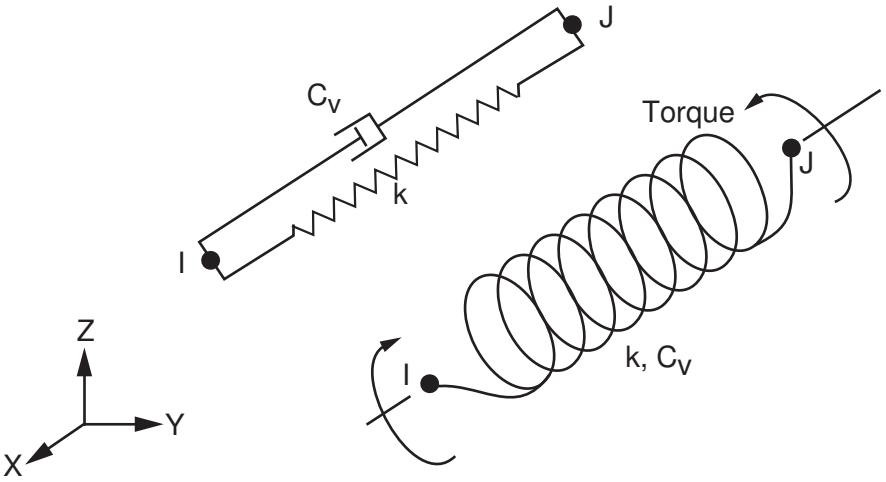
COMBI165 Element Description

COMBI165 allows you to model simple spring or damper systems, as well as the response of more complicated mechanisms such as the energy absorbers used in passenger vehicle bumpers. These mechanisms are often experimentally characterized in terms of force-displacement curves. This element provides a variety of discrete element formulations that can be used individually or in combination to model complex force-displacement relations.

COMBI165 is a two-node, 1-D element. You cannot define both spring and damper properties for the same element. Separate spring and damper elements are required, but they may use the same nodes (that is, you can overlay two different COMBI165 elements). A COMBI165 element can be attached to any of the other explicit elements.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 1 COMBI165 Geometry



Both figures above show two COMBI165 elements (a spring and a damper) attached to the same two nodes.

COMBI165 Input Data

The real constants K_d to TDL are optional and do not need to be defined.

For example, if K_d, the dynamic magnification factor, is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{dynamic} = \left(1. + K_d \frac{V}{V_0}\right) F_{static}$$

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use $K_d = 1.5$ and $V_0 = 15$, where V_0 is the test velocity.

Here, clearance (CL) defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve. If a nonzero clearance is defined, the spring is compressive only.

The deflection limit in compression (CDL) and tension (TDL) is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example, in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection (FD) is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where compression is limited.

For this element, you can choose from the following materials:

- Linear Elastic Spring
- Linear Viscous Damper
- Elastoplastic Spring
- Nonlinear Elastic Spring
- Nonlinear Viscous Damper
- General Nonlinear Spring
- Maxwell Viscoelastic Spring
- Inelastic Tension or Compression-Only Spring

A summary of the element input is given in *COMBI165 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBI165 Input Summary

Nodes
I, J

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ (KEYOPT(1) = 0)
ROTX, ROTY, ROTZ (KEYOPT(1) = 1)

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for post processing.

Real Constants

Kd - Dynamic magnification factor,
Vo - Test velocity,
CL - Clearance,
FD - Failure deflection,
CDL - Deflection limit (compression),
TDL - Deflection limit (tension)

Material Properties
DAMP (**MP** command), DISCRETE (**TB** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads
None

Body Loads
None

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Spring/damper type (translational or torsional):

0 --
The material describes a translational spring/damper

1 --
The material describes a torsional spring/damper

COMBI165 Output Data

Output data for COMBI165 consists of the following:

Table 1 COMBI165 Item and Sequence Numbers

Output Quantity Name[*1*]	ETABLE and ESOL Command Input		
	Description	Item	Seq. No.
MFORX/MMOMX	X-component of member force/moment	SMISC	1
MFORY/MMOMY	Y-component of member force/moment	SMISC	2
MFORZ/MMOMZ	Z-component of member force/moment	SMISC	3
MFORSUM/MMOMSUM	Vector sum of X, Y, and Z components of member force/moment	SMISC	4

1. You must specify *either* force or moment via KEYOPT(1). Note that you cannot specify *both* force and moment.

   MFOR: KEYOPT(1) = 0

   MMOM: KEYOPT(1) = 1

To output the element data in POST1, you must use the **ETABLE** command. Then, you can use the **PRETAB** command to print the output data. The **RSYS** command has no effect when postprocessing output for this element.

In POST26, you can postprocess the element data using the **ESOL** command only when postprocessing the `Jobname.RST` file. The element results are not available on the `Jobname.HIS` file.

COMBI165 Assumptions and Restrictions

- The time step size calculation is approximated by using the instantaneous stiffness and one-half the nodal mass of the nodes joined by the spring. If the global time step size is controlled by an explicit spring-damper element, instabilities can develop with the default time step size due to the approximations in the step size calculation.
- When used to interconnect under-integrated elements, the explicit spring-damper can sometimes excite the zero-energy hourglass modes.

- To ensure that parts are uniquely defined when using COMBI165, specify a unique set of real constants (**R**), the element type (**ET**), and the material properties (**TB**) for each part. Defining a unique material number (**MAT**) alone is insufficient.

COMBI165 Product Restrictions

There are no product-specific restrictions for this element.

MASS166

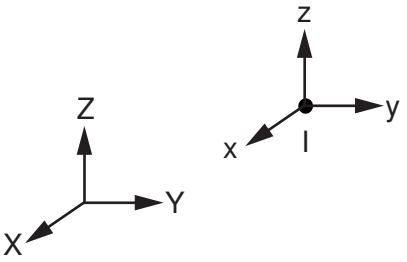
Explicit 3-D Structural Mass

<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

MASS166 Element Description

MASS166 is a point element having up to nine degrees of freedom: translations, velocities, and accelerations in the nodal x, y, and z directions.

Figure 1 MASS166 Geometry



MASS166 Input Data

The mass element is defined by a single node with concentrated mass components (Force*Time^2/Length) in the element coordinate directions about the element coordinate axes. The element also has an option for rotary inertia (without mass) which allows the definition of lumped rotary inertia at a defined nodal point. For the inertia option (KEYOPT(1) = 1), six polar moment of inertia values must be input instead of mass. To include both mass and rotary inertia, you must define two MASS166 elements at the same node.

A summary of the element input is given in *MASS166 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

MASS166 Input Summary

Nodes
I

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

If KEYOPT(1) = 0:
MASS - Concentrated mass (Force*Time^2/Length)
If KEYOPT(1) = 1:
IXX - Moment of inertia,
IXY - Moment of inertia,

IXZ - Moment of inertia,
IYY - Moment of inertia,
IYZ - Moment of inertia,
IZZ - Moment of inertia

Material Properties
None, but you must define realistic dummy material properties to make this element behave correctly.

Note

These dummy properties will not be used in any solution.

Surface Loads
None

Body Loads
None

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Rotary inertia option:

0 --
3-D mass without rotary inertia (default)

1 --
3-D rotary inertia (no mass)

MASS166 Output Data

Nodal displacements are included in the overall displacement solution. There is no printed or post element data output for the mass element.

MASS166 Assumptions and Restrictions

None.

MASS166 Product Restrictions

There are no product-specific restrictions for this element.

LINK167

Explicit Tension-Only Spar

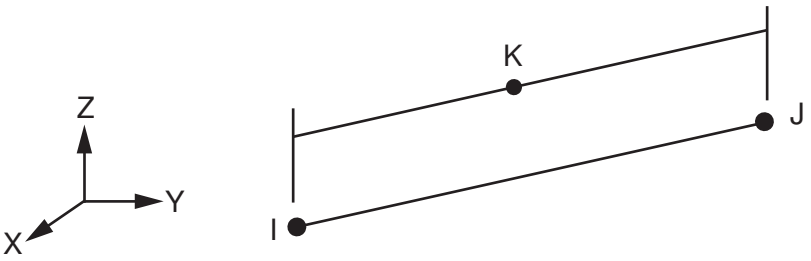
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

LINK167 Element Description

LINK167 allows elastic cables to be realistically modeled; thus, no force will develop in compression.

This element is used in explicit dynamic analyses only.

Figure 1 LINK167 Geometry



LINK167 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "LINK167 Geometry"*. Node K determines the initial orientation of the cross section.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the length of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element.

Real constants for this element are link area (AREA) and offset for cable (OFFSET). For a slack element, the offset should be input as a negative value. For an initial tensile force, the offset should be positive.

The force, *F*, generated by the link is nonzero if and only if the link is in tension. The force is given by:

$F = K \cdot max (\Delta L, 0.)$

where ΔL is the change in length

$\Delta L = current\ length - (initial\ length - offset)$

and the stiffness is defined as:

$$K = \frac{E \times area}{(initial\ length - offset)}$$

You can use only the material type cable for this element. For this material, you need to define the density (DENS) and Young's modulus (EX) or load curve ID. If you specify a load curve ID (**EDMP**,CABLE,VAL1, where VAL1 is the load curve ID), the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain (that is, the change in length over the

initial length). Use the **EDCURVE** command to define the load curve ID. The unloading behavior follows the loading.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

A summary of the element input is given in *LINK167 Input Summary*. Additional information about real constants for this element is provided in *Table 1, "BEAM161 Real Constants"*. For more information about this element, see the *LS-DYNA Theoretical Manual*.

LINK167 Input Summary

Nodes
I, J, K (K is the orientation node)

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

AREA - Cross-sectional area
OFFSET - Offset value for cable

Material Properties

EX (**MP** command) or Load Curve ID (**EDMP** command),
DENS (**MP** command),
DAMP (**MP** command),
CABLE (**EDMP** command; see Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads
None

Body Loads
None

Special Features
This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPTs
None

LINK167 Output Data

Output for LINK167 consists of the following: Axial force

To output the data, you must use the **ETABLE** command. For the ITEM label, specify SMISC. For the COMP label, specify 1 for axial force. Then, you can use the **PRETAB** command to print the output data.

LINK167 Assumptions and Restrictions

- The sum of the element length plus the offset must be greater than zero.
- The cross-sectional area must be greater than zero.

LINK167 Product Restrictions

There are no product-specific restrictions for this element.

SOLID168

Explicit 3-D 10-Node Tetrahedral Structural Solid

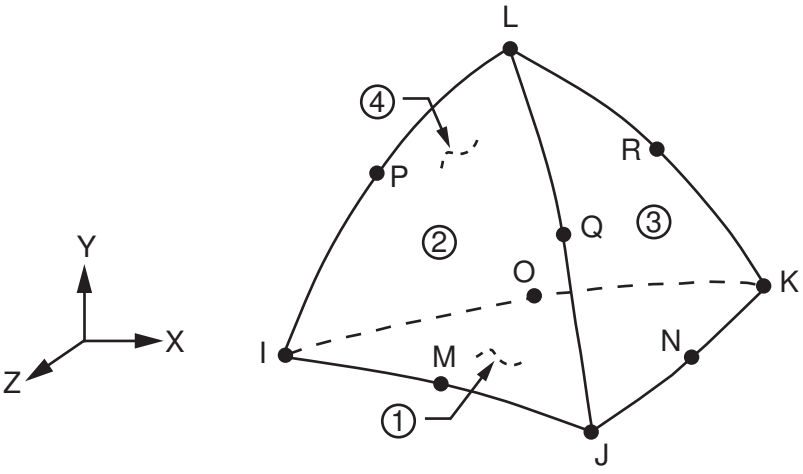
<> <> <> <> <> <> <> <> <> <> DY <> <>
Product Restrictions

SOLID168 Element Description

SOLID168 is a higher order 3-D, 10-node explicit dynamic element. It is well suited to modeling irregular meshes such as those produced from various CAD/CAM systems. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

By default, SOLID168 uses a quadratic displacement behavior with five point integration (KEYOPT(1) = 0 or 1). A composite formulation which is an assemblage of linear sub-tetrahedral shapes (KEYOPT(1) = 2) is also available. This second formulation effectively overcomes the difficulty of lumped mass calculations and volume locking inherent to the quadratic elements.

Figure 1 SOLID168 Geometry



SOLID168 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID168 Geometry"*. The element is defined by ten nodes. Orthotropic material properties may be defined. Use the **EDMP** command to specify an orthotropic material and the **EDLCS** command to define the orthotropic material directions.

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "SOLID168 Geometry"*. Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component. You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Several types of temperature loading are also available for this element. See *Section 4.5: Temperature Loading* in the *ANSYS LS-DYNA User's Guide*. For this element, you can choose from the materials listed below.

- Isotropic Elastic
- Orthotropic Elastic
- Anisotropic Elastic
- Bilinear Kinematic
- Plastic Kinematic
- Viscoelastic
- Blatz-Ko Rubber
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage
- Concrete Damage
- Geological Cap
- Piecewise Linear Plasticity
- Honeycomb
- Mooney-Rivlin Rubber
- Barlat Anisotropic Plasticity
- Elastic-Plastic Hydrodynamic
- Rate Sensitive Power Law Plasticity
- Elastic Viscoplastic Thermal
- Closed Cell Foam
- Low Density Foam
- Viscous Foam
- Crushable Foam
- Johnson-Cook Plasticity
- Null
- Zerilli-Armstrong
- Bamman
- Steinberg
- Elastic Fluid

SOLID168 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

None

Material Properties

EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ (or CTEX or THSXZ),
DENS, DAMP (**MP** command)
RIGID, HGLS, ORTHO, FLUID (**EDMP** command)
ANEL, MOONEY, EVISC, BISO, BKIN, PLAW,
FOAM, HONEY, COMPOSITE, CONCR, GCAP, EOS
(**TB** command; see *Chapter 7: Material Models* in the *ANSYS LS-DYNA User's Guide*)

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
See *Section 4.5: Temperature Loading* in the *ANSYS LS-DYNA User's Guide*

Special Features

This element supports all nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)

Element formulation:

0, 1 --
Quadratic interpolation

2 --
Composite (assemblages of linear tetrahedral shapes)

SOLID168 Output Data

Output for SOLID168 is listed in *Table 1, "SOLID168 Element Output Definitions"*. If you issue **PRNSOL**, a single set of stress and a single set of strain values is output at all ten nodes; that is, you will get the same sets of values at each node. If you issue **PRESOL**, you will get only a single set of values at the centroid.

You can rotate stress results for SOLID168 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 1 SOLID168 Element Output Definitions

Name	Definition
S:X, Y, Z, XY, YZ, XZ	Stresses
S:1, 2, 3	Principal stresses
S:INT	Stress intensity

Name	Definition
S:EQV	Equivalent stress
EPTO:X, Y, Z, XY, YZ, XZ	Total strains
EPTO:1, 2, 3	Total principle strains
EPTO:INT	Total strain intensity
EPTO:EQV	Total equivalent strain
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains
EPEL:1, 2, 3	Principal elastic strains
EPEL:INT	Elastic strain intensity
EPEL:EQV	Equivalent elastic strains
EPPL:EQV	Equivalent plastic strains

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see *Section 12.2.2: Element Output Data* in the *ANSYS LS-DYNA User's Guide* for details).

SOLID168 Assumptions and Restrictions

- Zero volume elements are not allowed.
- The element may not be twisted such that it has two separate volumes. This occurs most frequently when the element is not numbered properly.
- The element must have ten nodes.

SOLID168 Product Restrictions

There are no product-specific restrictions for this element.

TARGE169

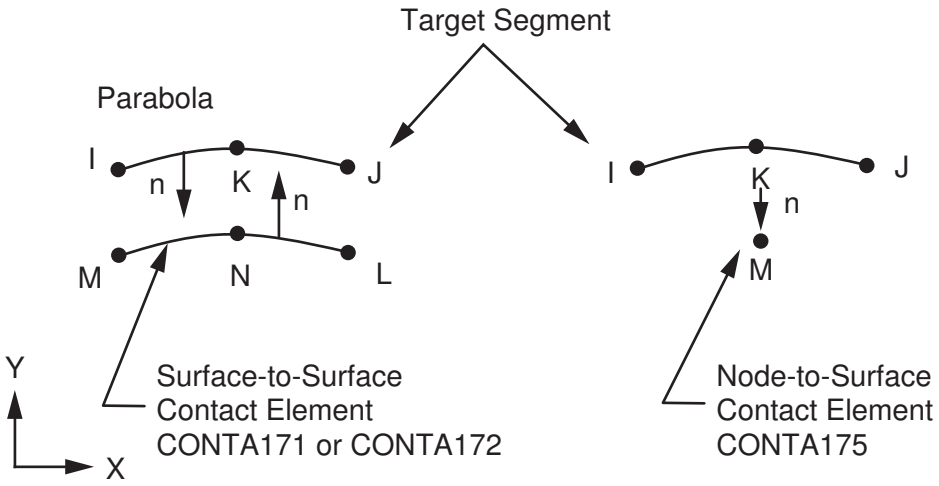
2-D Target Segment

MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

TARGE169 Element Description

TARGE169 is used to represent various 2-D "target" surfaces for the associated contact elements (CONTA171, CONTA172, and CONTA175). The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE169. This target surface is discretized by a set of target segment elements (TARGE169) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements. See TARGE169 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. To represent 3-D target surfaces, use TARGE170, a 3-D target segment element. For rigid targets, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body.

Figure 1 TARGE169 Geometry



TARGE169 Input Data

The target surface is modeled through a set of *target segments,* typically, several target segments comprise one target surface.

The target surface can either be rigid or deformable. For modeling rigid-flexible contact, the rigid surface must be represented by a target surface. For flexible-flexible contact, one of the deformable surfaces must be overlayed by a target surface. See the *Contact Technology Guide* for more information about designating contact and target surfaces.

The target and associated contact surfaces are identified by a shared real constant set. This real constant set includes all real constants for both the target and contact elements.

Each target surface can be associated with only one contact surface, and vice-versa. However, several contact elements could make up the contact surface and thus come in contact with the same target surface. Likewise, several target elements could make up the target surface and thus come in contact with the same contact surface. For either the target or contact surfaces, you can put many elements in a single target or contact surface, but

doing so may increase computational cost. For a more efficient model, localize the contact and target surfaces by splitting the large surfaces into smaller target and contact surfaces, each of which contain fewer elements.

If one contact surface may contact more than one target surface, you must define duplicate contact surfaces that share the same geometry but relate to separate targets, that is, have separate real constant set numbers.

For any target surface definition, the node ordering of the target segment element is critical for proper detection of contact. The nodes must be ordered so that, for a 2-D surface, the associated contact elements (CONTA171, CONTA172, or CONTA175) must lie to the right of the target surface when moving from target node I to target node J. For a rigid 2-D complete circle, contact must occur on the outside of the circle; internal contacting is not allowed.

Considerations for Rigid Targets

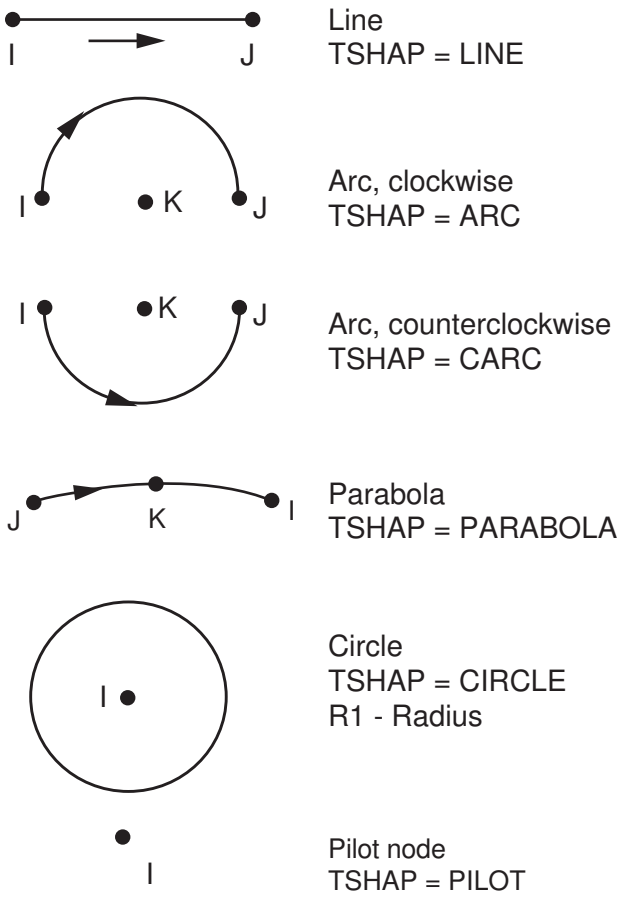
Each target segment is a single element with a specific shape, or *segment type*. The segment types are defined by one, two, or three nodes and a target shape code, **TSHAP**, and are described in *Table 1, "TARGE169 2-D Segment Types, Target Shape Codes, and Nodes"*. The **TSHAP** command indicates the geometry (shape) of the element. The segment dimensions are defined by a real constant (R1), and the segment location is determined by the nodes. ANSYS supports six 2-D segment types; see *Table 1, "TARGE169 2-D Segment Types, Target Shape Codes, and Nodes"*.

Table 1 TARGE169 2-D Segment Types, Target Shape Codes, and Nodes

TSHAP	Segment Type	Node1 (DOF)	Node 2 (DOF)[1]	Node 3 (DOF)	R1[2]	R2
LINE	Straight line	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None	None
ARC	Arc, clock-wise	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
CARC	Arc, counter-clockwise	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
PARA	Parabola	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Midside pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
CIRC	Circle	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None	Radius	None
PILO	Pilot node	2-D: (UX, UY, ROTZ) (TEMP) (VOLT) (AZ)	None	None	None	None

1. The DOF available depends on the setting of KEYOPT(1) for the associated contact element. For more information, see the element documentation for CONTA171, CONTA172, or CONTA175.
2. When creating a circle via direct generation, define the real constant R1 *before* creating the element.

Figure 2 TARGE169 2-D Segment Types



For simple rigid target surfaces, you can define the target segment elements individually by direct generation. You must first specify the *SHAPE* argument for the **TSHAP** command. When creating circles through direct generation, you must also define the real constant R1 before creating the element. Real constant R1 (see *Table 1, "TARGE169 2-D Segment Types, Target Shape Codes, and Nodes"*) defines the radius of the target circle.

For general 2-D rigid surfaces, target segment elements can be defined by line meshing (**LMESH**). You can also use keypoint meshing (**KMESH**) to generate the pilot node.

If the TARGE169 elements will be created via automatic meshing (**LMESH** or **KMESH**), then the **TSHAP** command is ignored and ANSYS chooses the correct shape automatically.

The pilot node provides a convenient, powerful way to assign boundary conditions such as rotations, translations, moments, temperature, and voltage on an entire rigid target surface. You assign the conditions only to the pilot node, eliminating the need to assign boundary conditions to individual nodes and reducing the chance of error. The pilot node, unlike the other segment types, is used to define the degrees of freedom for the entire target surface. This node can be any of the target surface nodes, but it does not have to be. All possible rigid motions of the target surface will be a combination of a translation and a rotation around the pilot node. The boundary conditions (including displacement, rotation, force, moment, temperature, voltage, and magnetic potential) of the entire target surface can be specified only on pilot nodes.

For rotation of a rigid body constrained only by a bonded, rigid-flexible contact pair with a pilot node, use the MPC algorithm or a surface-based constraint as described in Multipoint Constraints and Assemblies. Penalty-based algorithms can create undesirable rotational energies in this situation.

By default, ANSYS automatically fixes the degree of freedom for rigid target nodes if they aren't explicitly constrained (KEYOPT(2) = 0). If you wish, you can override the automatic boundary condition settings by setting KEYOPT(2) = 1.

By default, the temperature is set to the value of TUNIF, and if this has no explicit value the temperature is set to zero. For thermal contact analysis, such as convection and radiation modeling, the behavior of a thermal contact surface (whether a "near-field" or "free" surface) is usually based on the contact status. Contact status affects the behavior of the contact surface as follows:

- If the contact surface is outside the pinball region, its behavior is as a far-field of free surface. In this instance, convection/radiation occurs with the ambient temperature.
- If the contact surface is inside the pinball region, the behavior is as a near-field surface.

However, the thermal contact surface status is ignored if KEYOPT(3) = 1 is set, and the surface is always treated as a free surface (see CONTA171, CONTA172, or CONTA175 for details).

Considerations for Deformable Target Surfaces

For general deformable surfaces, you will normally use the **ESURF** command to overlay the target elements on the boundary of the existing mesh. Note that the segment types (**TSHAP** command) should not be used for this case.

A summary of the element input is given in *TARGE169 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

TARGE169 Input Summary

Nodes
I, J, K (J and K are not required for all segment types)

Degrees of Freedom
UX, UY, ROTZ, TEMP, VOLT, AZ (ROTZ is used for the pilot node only)

Real Constants
R1, R2, [the others are defined through the associated CONTA171, CONTA172, or CONTA175 element]

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Birth and death

KEYOPT(2)
Boundary conditions for rigid target nodes:

0 --
Automatically constrained by ANSYS

1 --
Specified by user

KEYOPT(3)

Behavior of thermal contact surface

0 --

Based on contact status

1 --

Treated as free-surface

KEYOPT(4)

DOF set to be constrained on dependent DOF for internally-generated multipoint constraints (MPCs). This option is used for solid-solid and shell-shell assemblies and for surface-based constraints that use a single pilot node for the target element:

n --

Enter a three digit value that represents the DOF set to be constrained. The first to third digits represent ROTZ, UY, UX, respectively. The number 1 (one) indicates the DOF is active, and the number 0 (zero) indicates the DOF is not active. For example, 011 means that UX and UY will be used in the multipoint constraint. Leading zeros may be omitted; for example, you can enter 1 to indicate that UX is the only active DOF. If KEYOPT(4) = 0 (which is the default) or 111, all DOF are constrained.

TARGE169 Output Data

The solution output associated with the element is shown in *Table 2, "TARGE169 Element Output Definitions"*. The following notation is used:

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 TARGE169 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, and K	Y	Y
ITRGET	Target surface number (assigned by ANSYS)	Y	Y
TSHAP	Segment shape type	Y	Y
ISEG	Segment numbering	*1*	*1*

1. Determined by ANSYS

TARGE169 Assumptions and Restrictions

- The 2-D segment element must be defined in an X-Y plane.
- For circular arcs, the third node defines the actual center of the circle and must be defined accurately when the element is generated and must be moved consistently with the other nodes during the deformation process. If the third node is not moved consistently with the other nodes, the arc shape will change with that node's movement. To ensure the correct behavior, apply all boundary conditions to a pilot node.
- For parabolic segments, the third point must lie at the middle of the parabola.

- For rigid surfaces, no external forces can be applied on target nodes except on a pilot node. If a pilot node is specified for a target surface, ANSYS will ignore the boundary conditions on any nodes of the target surface except for the pilot nodes. For each pilot node, ANSYS automatically defines an internal node and an internal constraint equation. The rotational DOF of the pilot node is connected to the translational DOF of the internal node by the internal constraint equation. You cannot use constraint equations or coupling on pilot nodes.
- Generally speaking, you should not change the R1 real constant between load steps or during restart stages; otherwise ANSYS assumes the radius of the circle varies between the load steps. When using direct generation, the real constant R1 for circles may be defined before the input of the element nodes. If multiple rigid circles are defined, each having a different radius, they must be defined by different target surfaces.

TARGE169 Product Restrictions

There are no product-specific restrictions for this element.

TARGE170

3-D Target Segment

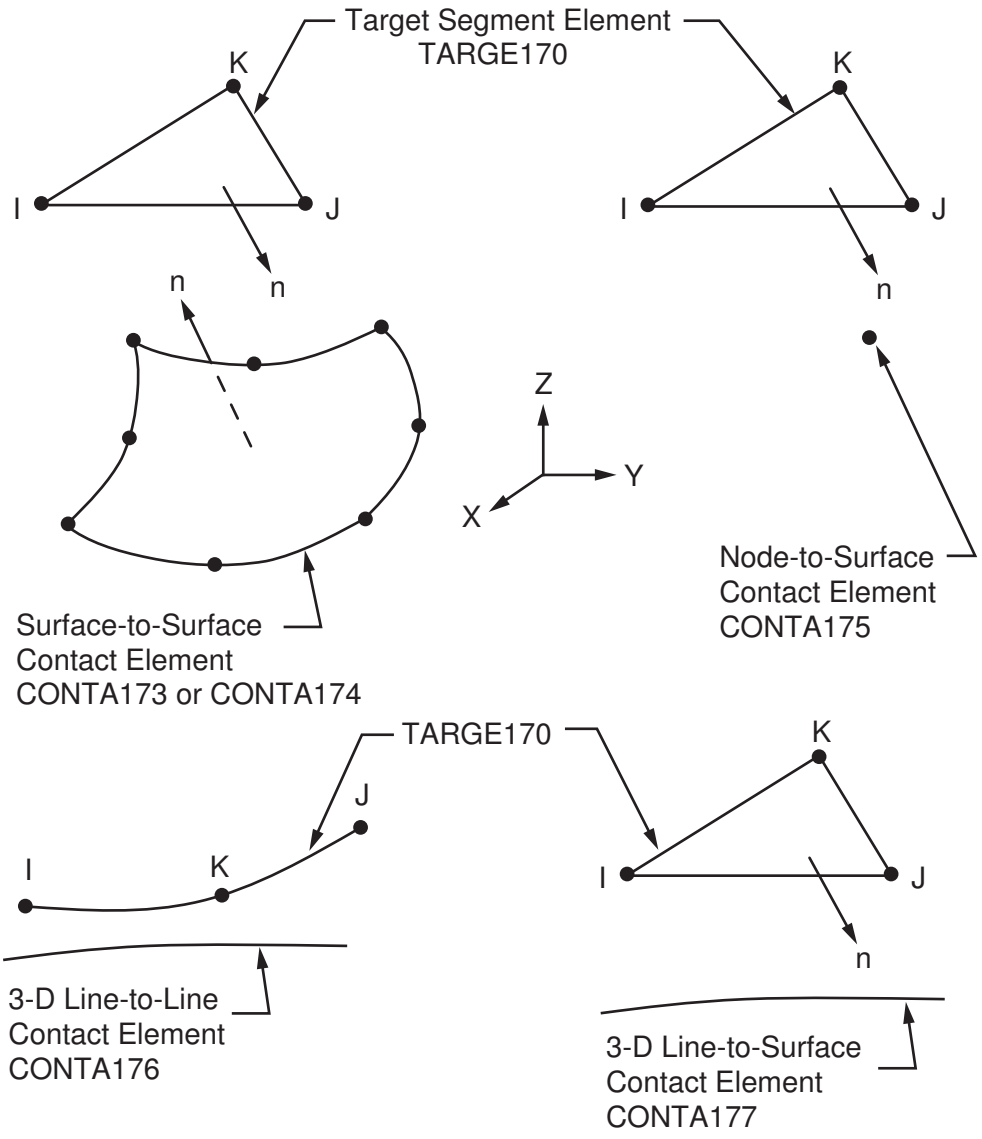
MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

TARGE170 Element Description

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements (CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177). The contact elements themselves overlay the solid, shell, or line elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements. See TARGE170 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. To represent 2-D target surfaces, use TARGE169, a 2-D target segment element.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid, shell, or line elements describing the boundary of the deformable target body.

Figure 1 TARGE170 Geometry



TARGE170 Input Data

The target surface is modeled through a set of *target segments,* typically, several target segments comprise one target surface.

The target surface can either be rigid or deformable. For modeling rigid-flexible contact, the rigid surface must be represented by a target surface. For flexible-flexible contact, one of the deformable surfaces must be overlayed by a target surface. See the *Contact Technology Guide* for more information about designating contact and target surfaces.

The target and associated contact surfaces are identified via a shared real constant set. This real constant set includes all real constants for both the target and contact elements.

Each target surface can be associated with only one contact surface, and vice-versa. However, several contact elements could make up the contact surface and thus come in contact with the same target surface. Likewise, several target elements could make up the target surface and thus come in contact with the same contact surface. For either the target or contact surfaces, you can put many elements in a single target or contact surface, but

doing so may increase computational cost. For a more efficient model, localize the contact and target surfaces by splitting the large surfaces into smaller target and contact surfaces, each of which contain fewer elements.

If a contact surface may contact more than one target surface, you must define duplicate contact surfaces that share the same geometry but relate to separate targets, that is, that have separate real constant set numbers.

Figure 2, "TARGE170 Segment Types" shows the available segment types for TARGE170. The general 3-D surface segments (3-node and 6-node triangles, and 4-node and 8-node quadrilaterals) and the primitive segments (cylinder, cone, and sphere) can be paired with 3-D surface-to-surface contact elements, CONTA173 and CONTA174, the 3-D node-to-surface contact element, CONTA175, and the 3-D line-to-surface contact element, CONTA177. The line segments (2-node line and 3-node parabola) can only be paired with the 3-D line-to-line contact element, CONTA176, to model 3-D beam-to-beam contact.

For any target surface definition, the node ordering of the target segment element is critical for proper detection of contact. For the general 3-D surface segments (triangle and quadrilateral segment types), the nodes must be ordered so that the outward normal to the target surface is defined by the right hand rule (see *Figure 2, "TARGE170 Segment Types"*). Therefore, for the surface target segments, the outward normal by the right hand rule is consistent to the external normal. For 3-D line segments (straight line and parabolic line), the nodes must be entered in a sequence that defines a continuous line. For a rigid cylinder, cone, or sphere, contact must occur on the outside of the elements; internal contacting of these segments is not allowed.

Considerations for Rigid Target Surfaces

Each target segment of a rigid surface is a single element with a specific shape, or *segment type*. The segment types are defined by several nodes and a target shape code, **TSHAP**, and are described in *Table 1, "TARGE170 3-D Segment Types, Target Shape Codes, and Nodes"*. The **TSHAP** command indicates the geometry (shape) of the element. The segment radii are defined by real constants (R1 and R2), and the segment location is determined by the nodes. ANSYS supports ten 3-D segment types; see *Table 1, "TARGE170 3-D Segment Types, Target Shape Codes, and Nodes"*.

Table 1 TARGE170 3-D Segment Types, Target Shape Codes, and Nodes

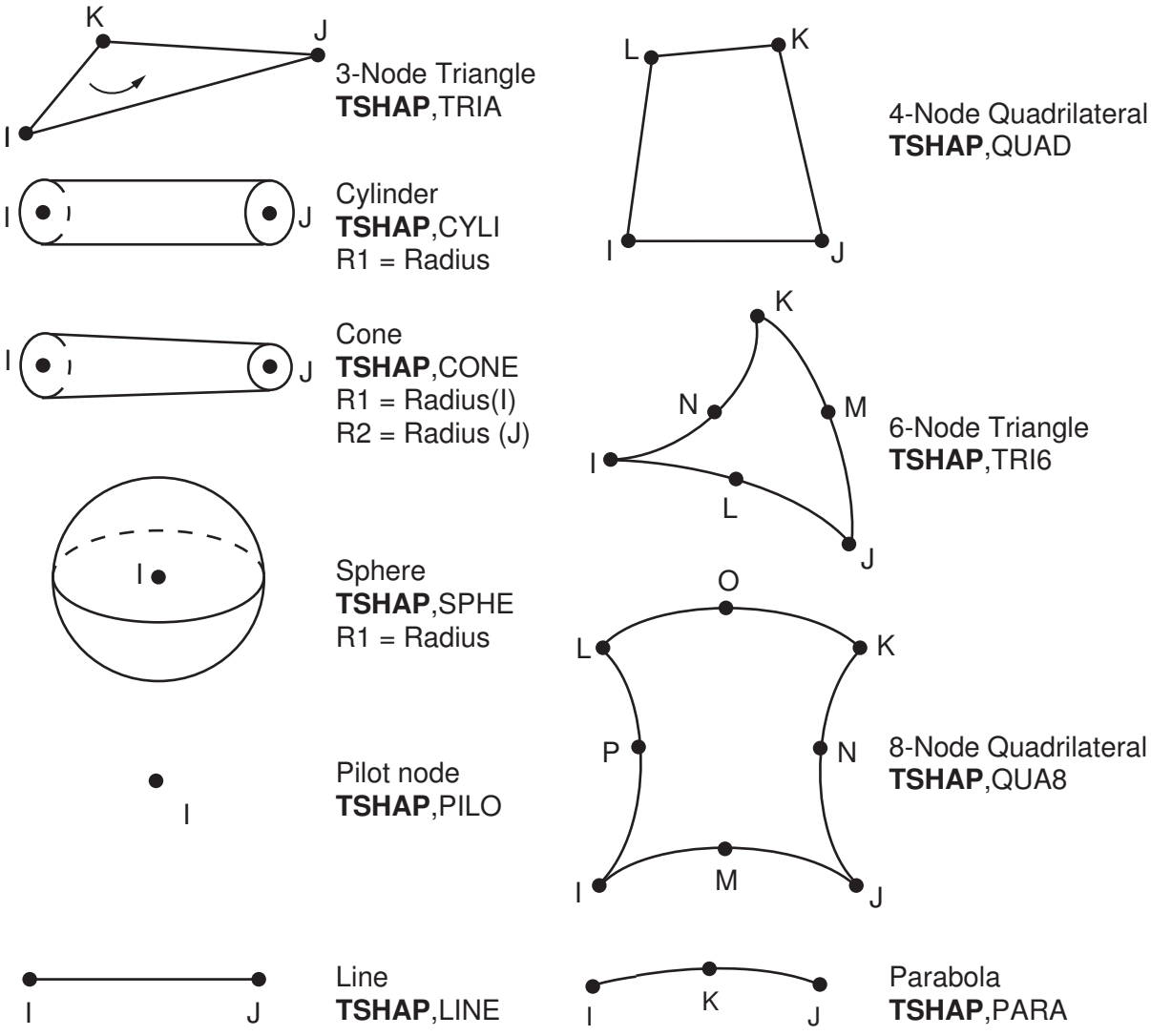
TSHAP	Segment Type	Nodes (DOF)[1]	R1	R2
TRIA	3-node triangle	1st - 3rd nodes are corner points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
QUAD	4-node quadrilateral	1st - 4th nodes are corner points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
TRI6	6-node triangle	1st - 3rd nodes are corner points, 4th - 6th are midside nodes (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
QUA8	8-node quadrilateral	1st - 4th nodes are corner points, 5th - 8th are midside nodes (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
LINE	2-node straight line	1st - 2nd nodes are line end points (UX, UY, UZ)	Target Radius[4]	Contact Radius[5]
PARA	3-node parabola	1st - 2nd nodes are line end points, 3rd is a midside node (UX, UY, UZ)	Target Radius[4]	Contact Radius[5]
CYLI	Cylinder[2]	1st - 2nd nodes are axial end points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius	None
CONE	Cone[2]	1st - 2nd nodes are axial end points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius at node 1	Radius at node 2

TSHAP	Segment Type	Nodes (DOF)[1]	R1	R2
SPHE	Sphere[2]	Sphere center point (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius	None
PILO	Pilot node[3]	1st point: (UX, UY, UZ, ROTX, ROTY, ROTZ) (TEMP) (VOLT) (MAG)	None	None

1. The DOF available depends on the setting of KEYOPT(1) of the associated contact element. Refer to the element documentation for either CONTA173, CONTA174, or CONTA175 for more details.
2. When creating a cylinder, cone, or sphere via direct generation, define the real constant set *before* creating the element.
3. Only pilot nodes have rotational degrees of freedom (ROTX, ROTY, ROTZ).
4. Input the target radius as a negative value when modeling internal pipe-to-pipe contact (a pipe contacting/sliding inside another pipe). Input a positive value to model external 3-D beam-to-beam contact.
5. Input a positive contact radius when modeling internal pipe-to-pipe contact or external 3-D beam-to-beam contact.

Figure 2, "TARGE170 Segment Types" shows the 3-D segment shapes.

Figure 2 TARGE170 Segment Types



For simple rigid target surfaces (including line segments), you can define the target segment elements individually by direct generation. You must first specify the *SHAPE* argument on the **TSHAP** command. When creating cylinders, cones, or spheres through direct generation, you must also define the real constant R1 (and R2 for cones) before creating the element.

For general 3-D rigid surfaces, target segment elements can be defined by area meshing (**AMESH**). Set KEYOPT(1) = 0 (the default) to generate low order target elements (3-node triangles and/or 4-node quadrilaterals) for rigid surfaces. Set KEYOPT(1) = 1 to generate target elements with midside nodes (6-node triangles and/or 8-node quadrilaterals).

For 3-D rigid lines, target segment elements can be defined by line meshing (**LMESH**). Set KEYOPT(1) = 0 (the default) to generate low order target elements (2-node straight lines). Set KEYOPT(1) = 1 to generate target elements with midside nodes (3-node parabolas).

You can also use keypoint meshing (**KMESH**) to generate the pilot node.

If the TARGE170 elements will be created via program meshing (**AMESH**, **LMESH**, or **KMESH** commands), then the **TSHAP** command is ignored and ANSYS chooses the correct shape automatically.

The pilot node provides a convenient, powerful way to assign boundary conditions such as rotations, translations, moments, temperature, and voltage on an entire rigid target surface. You assign the conditions only to the pilot node, eliminating the need to assign boundary conditions to individual nodes and reducing the chance of error. The pilot node, unlike the other segment types, is used to define the degrees of freedom for the entire target surface. This node can be any of the target surface nodes, but it does not have to be. All possible rigid motions of the target surface will be a combination of a translation and a rotation around the pilot node. The boundary conditions (including displacement, rotation, force, moment, temperature, voltage, and magnetic potential) of the entire target surface can be specified only on pilot nodes.

For rotation of a rigid body constrained only by a bonded, rigid-flexible contact pair with a pilot node, use the MPC algorithm or a surface-based constraint as described in Multipoint Constraints and Assemblies. Penalty-based algorithms can create undesirable rotational energies in this situation.

Real constants R1 and R2 (see *Table 1, "TARGE170 3-D Segment Types, Target Shape Codes, and Nodes"*) define the dimensions of the target shape.

By default, ANSYS automatically fixes the structural degree of freedom for rigid target nodes if they aren't explicitly constrained (KEYOPT(2) = 0). If you wish, you can override the automatic boundary condition settings by setting KEYOPT(2) = 1.

By default, the temperature is set to the value of TUNIF, and if this has no explicit value the temperature is set to zero. For thermal contact analysis, such as convection and radiation modeling, the behavior of a thermal contact surface (whether a "near-field" or "free" surface) is usually based on the contact status. Contact status affects the behavior of the contact surface as follows:

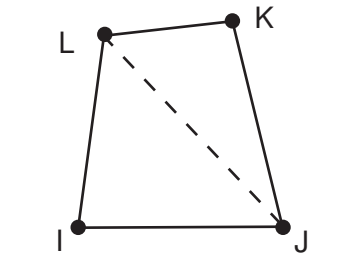
- If the contact surface is outside the pinball region, its behavior is as a far-field of free surface. In this instance, convection/radiation occurs with the ambient temperature.
- If the contact surface is inside the pinball region, the behavior is as a near-field surface.

However, the thermal contact surface status is ignored if KEYOPT(3) = 1 is set, and the surface is always treated as a free surface (see CONTA173, CONTA174, or CONTA175 for details).

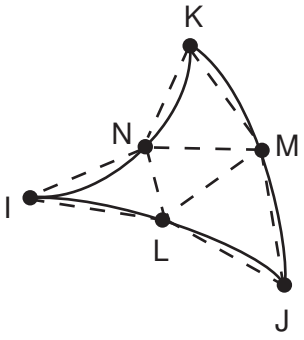
Considerations for Deformable Target Surfaces

For general deformable surfaces, use the **ESURF** command to overlay the target elements on the boundary of the existing mesh. By default, the command generates a target element with an external surface that has the same shape as the underlying element. While *not recommended*, you may select to split the external surface into triangle facet elements by issuing the **ESURF**,,,TRI command (see *Figure 3, "TARGE170 Triangle Facet Elements"*). The cylinder, cone, sphere, or pilot node target segments should not be used for deformable target surfaces.

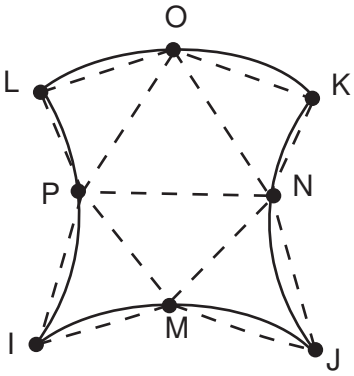
Figure 3 TARGE170 Triangle Facet Elements



4 Node Quadrilateral
TSHAP,QUAD



6 Node Triangle
TSHAP,TRI6



8 Node Quadrilateral
TSHAP,QUA8

Note

Segment types (**TSHAP** command) should not be used for this case

A summary of the element input is given in *TARGE170 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

TARGE170 Input Summary

Nodes
I, J, K, L, M, N, O, P (J - P are not required for all segment types)

Degrees of Freedom
UX, UY, UZ, TEMP, VOLT, MAG (ROTX, ROTY, ROTZ for pilot nodes only)

Real Constants
R1, R2, [the others are defined through the associated CONTA173, CONTA174, CONTA175, CONTA176, or CONTA177 elements]

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features

Nonlinear
Birth and death

KEYOPT(1)
Element order (used by **AMESH** and **LMESH** commands only):

0 --
Low order elements

1 --
High order elements

KEYOPT(2)
Boundary conditions for rigid target nodes:

0 --
Automatically constrained by ANSYS

1 --
Specified by user

KEYOPT(3)
Behavior of thermal contact surface:

0 --
Based on contact status

1 --
Treated as free-surface

KEYOPT(4)
DOF set to be constrained on dependent DOF for internally-generated multipoint constraints (MPCs). This option is used for solid-solid and shell-shell assemblies and for surface-based constraints that use a single pilot node for the target element:

n --
Enter a six digit value that represents the DOF set to be constrained. The first to sixth digits represent ROTZ, ROTY, ROTX, UZ, UY, UX, respectively. The number 1 (one) indicates the DOF is active, and the number 0 (zero) indicates the DOF is not active. For example, 100011 means that UX, UY, and ROTZ will be used in the multipoint constraint. Leading zeros may be omitted; for example, you can enter 11 to indicate that UX and UY are the only active DOF. If KEYOPT(4) = 0 (which is the default) or 111111, all DOF are constrained.

KEYOPT(5)
DOF set to be used in internally-generated multipoint constraints (MPCs), with the MPC algorithm and no separation or bonded behavior (KEYOPT(2) = 2 and KEYOPT(12) = 4, 5, or 6 on the contact element). Note that this key option is not used for surface-based constraints. (See *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information.):

0 --
Automatic constraint type detection (default)

1 --
Solid-solid constraint (no rotational DOFs are constrained)

2 --
Shell-shell constraint (both translational and rotational DOFs are constrained). Also used with penalty based shell-shell assembly (KEYOPT(2) = 0 or 1 and KEYOPT(12) = 5 or 6 on the contact element); see *Section 3.8.11.4: Bonded Contact for Shell-Shell Assemblies* in the *Contact Technology Guide* for more information.

3 --
Shell-solid constraint - contact normal direction (both translational and rotational DOFs are constrained on shell edges; only translational DOFs are constrained on solid surfaces)

4 --
Shell-solid constraint - all directions. This option acts the same as KEYOPT(5) = 3 if an intersection is found from the contact normal to the target surface. Otherwise, constraint equations are still built as long as contact node(s) and target segments are inside the pinball region.

Note

When the no separation option (KEYOPT(12) = 4 on the contact element) is used with the MPC approach, only the KEYOPT(5) = 0 and 1 options (auto detection or solid-solid constraint) described above are valid. If the auto detection option is set and the program finds a shell-shell or shell-solid constraint in this situation, the solution will terminate.

TARGE170 Output Data

The solution output associated with the element is shown in *Table 2, "TARGE170 Element Output Definitions"*.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 TARGE170 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, and K	Y	Y
ITRGET	Target surface number (assigned by ANSYS)	Y	Y
TSHAP	Segment shape type	Y	Y
ISEG	Segment numbering	*1*	*1*

1. Determined by ANSYS

TARGE170 Assumptions and Restrictions

- Generally speaking, you should not change real constants R1 or R2, either between load steps or during restart stages; otherwise ANSYS assumes the radii of the primitive segments varies between the load steps. When using direct generation, the real constants for cylinders, cones, and spheres may be defined before the input of the element nodes. If multiple rigid primitives are defined, each having different radii, they must be defined by different target surfaces.

- No external forces can be applied on target nodes except on a pilot node. To ensure the correct behavior, apply all boundary conditions to a pilot node.
- If a pilot node is specified for a target surface, ANSYS will ignore the boundary conditions on any nodes of the target surface except for the pilot nodes. For each pilot node, ANSYS automatically defines an internal node and an internal constraint equation. The rotational DOF of the pilot node is connected to the translational DOF of the internal node by the internal constraint equation.
- ANSYS recommends against using constraint equations or coupling on pilot nodes; if you do, conflicts may occur, yielding incorrect results. CE and CP commands do not apply to other nodes of a target surface because they do not have degrees of freedom (even when the pilot node is not present).

TARGE170 Product Restrictions

There are no product-specific restrictions for this element.

CONTA171

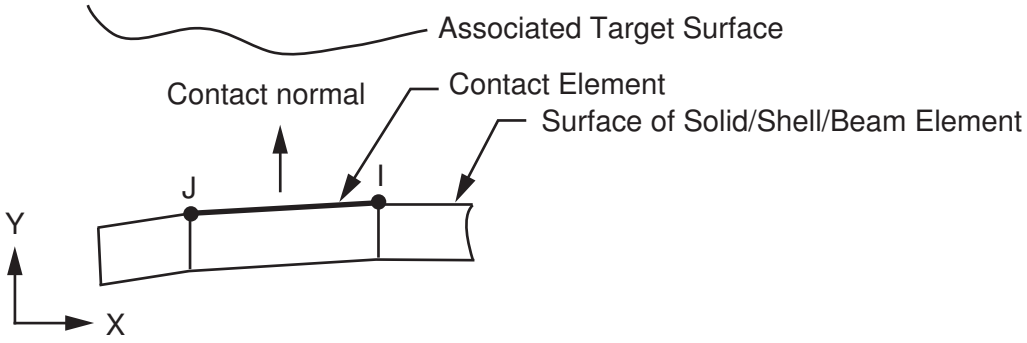
2-D 2-Node Surface-to-Surface Contact

MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

CONTA171 Element Description

CONTA171 is used to represent contact and sliding between 2-D "target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 2-D structural and coupled field contact analyses. This element is located on the surfaces of 2-D solid, shell, or beam elements without midside nodes (PLANE42, PLANE67, PLANE182, VISCO106, SHELL208, BEAM3, BEAM23, PLANE13, PLANE55, or MATRIX50). It has the same geometric characteristics as the solid, shell, or beam element face with which it is connected (see *Figure 1, "CONTA171 Geometry"*). Contact occurs when the element surface penetrates one of the target segment elements (TARGE169) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA171 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other surface-to-surface contact elements (CONTA172, CONTA173, CONTA174) are also available.

Figure 1 CONTA171 Geometry



CONTA171 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA171 Geometry"*. The element is defined by two nodes (the underlying solid, shell, or beam element has no midside nodes). If the underlying solid, shell, or beam elements do have midside nodes, use CONTA172. The element x-axis is along the I-J line of the element. The correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered such that the target must lie to the right side of the contact element when moving from the first contact element node to the second contact element node as in *Figure 1, "CONTA171 Geometry"*. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 2-D contact surface elements are associated with the 2-D target segment elements (TARGE169) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For modeling either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

This element supports various 2-D stress states, including plane stress, plane strain, and axisymmetric states. The stress state is automatically detected according to the stress state of the underlying element. However, if the underlying element is a superelement, you must use KEYOPT(3) to specify the stress state.

A summary of the element input is given in *CONTA171 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

CONTA171 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY (if KEYOPT(1) = 0)
UX, UY, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS
See *Table 1, "CONTA171 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU, EMIS (**MP** command)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Surface Loads

Convection, Face 1 (I-J)
Heat Flux, Face 1 (I-J)

Special Features

Nonlinear
Large deflection
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY

1 --
UX, UY, TEMP

2 --
TEMP

3 --
UX, UY, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, VOLT

6 --
VOLT

7 --
AZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Stress state when superelements are present:

0 --
Use with h-elements (no superelements)

1 --
Axisymmetric (use with superelements only)

2 --
Plane stress/Plane strain (use with superelements only)

3 --
Plane stress with thickness input (use with superelements only)

KEYOPT(4)

Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint, set KEYOPT(4) = 2 for a rigid surface constraint. See Surface-based Constraints for more information.

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. Activated only if **SOLCONTROL**,ON,ON at the procedure level.

KEYOPT(8)

Asymmetric contact selection:

0 --

No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)

Contact stiffness update:

0 --

Each load step if FKN is redefined during load step (pair based).

1 --

Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --

Each iteration based on current mean stress of underlying elements (pair based).

3 --

Each load step if FKN is redefined during load step (individual element based).

4 --

Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --

Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)

Beam/Shell thickness effect:

0 --
Exclude

1 --
Include

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA171 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target circle radius	Defining the Target Surface
2	R2	Superelement thickness	Defining the Target Surface
3	FKN	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection

CONTA171 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

- Additional element output as shown in *Table 2, "CONTA171 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2, "CONTA171 Element Output Definitions" gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 2 CONTA171 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J	Y	Y
XC, YC	Location where results are reported	Y	*5*
TEMP	Temperatures T(I), T(J)	Y	Y
LENGTH	Element length	Y	-
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid, shell, or beam element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*
NX, NY	Surface normal vector components	Y	-
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact pressure	Y	Y
CONT:SFRIC	Tangential contact stress	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU	Friction coefficient	Y	-
CONT:SLIDE	Total accumulated sliding (algebraic sum)	*3*	*3*
CONT:ASLIDE	Total accumulated sliding (absolute sum)	*3*	*3*

Name	Definition	O	R
TOLN	Penetration tolerance	Y	Y
CONT:STOTAL	Total stress SQRT (PRES**2+SFRIC**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX	Contact element force-x component	-	*4*
CNFY	Contact element force-Y component	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
FDDIS	Frictional energy dissipation	*6*	*6*
FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
CAREA	Contacting area	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system.
5. Available only at centroid as a ***GET** item.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Table 3, "CONTA171 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CONTA171 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTA171 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I, J

Table 3 CONTA171 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PRES	SMISC	5	1	2
SFRIC	SMISC	-	3	4
FLUX	SMISC	-	6	7
FDDIS	SMISC	-	8	9
FXCV	SMISC	-	10	11
FXRD	SMISC	-	12	13
FXCD	SMISC	-	14	15
FXNP	SMISC	-	16	17
JCONT	SMISC	-	18	19
CCONT	SMISC	-	18	19
HJOU	SMISC	-	20	21
STAT[*1*]	NMISC	19	1	2
OLDST	NMISC	-	3	4
PENE[*2*]	NMISC	-	5	6
DBA	NMISC	-	7	8
SLIDE	NMISC	-	9	10
KN	NMISC	-	11	12
KT	NMISC	-	13	14

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
TOLN	NMISC	-	15	16
IGAP	NMISC	-	17	18
PINB	NMISC	20	-	-
CNFX	NMISC	21	-	-
CNFY	NMISC	22	-	-
ISEG	NMISC	-	23	24
ASLIDE	NMISC	-	25	26
CAREA	NMISC	-	27	28
MU	NMISC	-	29	30
DTSTART	NMISC	-	31	32
DPARAM	NMISC	-	33	34
TEMPS	NMISC	-	37	38
TEMPT	NMISC	-	39	40
CONV	NMISC	-	41	42
RAC	NMISC	-	43	44
TCC	NMISC	-	45	46
CNFH	NMISC	47	-	-
ECURT	NMISC	48	-	-
ECHAR	NMISC	48	-	-
ECC	NMISC	-	49	50
VOLTS	NMISC	-	51	52
VOLTT	NMISC	-	53	54
CNOS	NMISC	-	55	56
TNOP	NMISC	-	57	58
SLTO	NMISC	-	59	60
ELSI	NMISC	-	67	68
DENERI	NMISC	-	69	70
DENERII	NMISC	-	71	72

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface

CNOS	Total number of contact status changes during substep

CONTA171 Assumptions and Restrictions

- The 2-D contact element must be defined in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- This 2-D contact element works with any 3-D elements in your model.
- Do not use this element in any model that contains axisymmetric harmonic elements.
- Node numbering must coincide with the external surface of the underlying solid, shell, or beam element, or with the original elements comprising the superelement.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which FTOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- When nodal detection is used and the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.
- This element allows birth and death and will follow the birth and death status of the underlying solid, shell, beam, or target elements.

CONTA171 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed.
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The AZ DOF (KEYOPT(1) = 7) is not allowed.

CONTA172

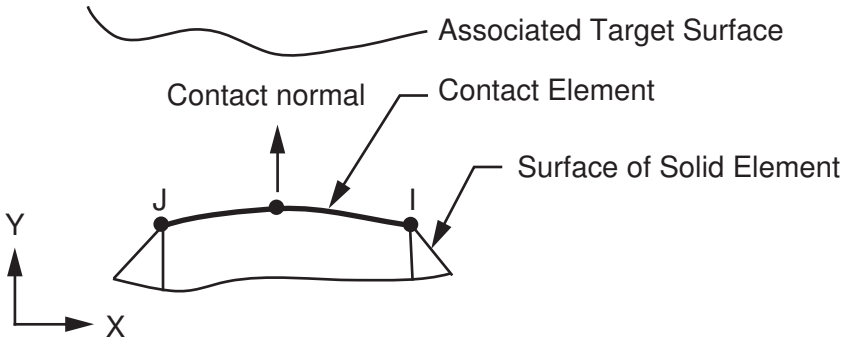
2-D 3-Node Surface-to-Surface Contact

MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

CONTA172 Element Description

CONTA172 is used to represent contact and sliding between 2-D "target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 2-D structural and coupled field contact analyses. This element is located on the surfaces of 2-D solid elements with midside nodes (PLANE121, PLANE183, SHELL209, PLANE82, VISCO88, VISCO108, PLANE35, PLANE77, PLANE53, PLANE223, PLANE230, or MATRIX50). It has the same geometric characteristics as the solid element face with which it is connected (see *Figure 1, "CONTA172 Geometry"*). Contact occurs when the element surface penetrates one of the target segment elements (TARGE169) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA172 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for a discussion concerning midside nodes. Other surface-to-surface contact elements (CONTA171, CONTA173, CONTA174) are also available.

Figure 1 CONTA172 Geometry



CONTA172 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA172 Geometry"*. The element is defined by three nodes (the underlying solid element has midside nodes). If the underlying solid elements do not have midside nodes, use CONTA171 (you may still use CONTA172 but you must drop the midside nodes). The element x-axis is along the I-J line of the element. The correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered such that the target must lie to the right side of the contact element when moving from the first contact element node to the second contact element node as in *Figure 1, "CONTA172 Geometry"*. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 2-D contact surface elements are associated with the 2-D target segment elements (TARGE169) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets with different real constant numbers), or you must combine the two target surfaces into one (both having the same real constant number).

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

This element supports various 2-D stress states, including plane stress, plane strain, and axisymmetric states. The stress state is automatically detected according to the stress state of the underlying element. However, if the underlying element is a superelement, you must use KEYOPT(3) to specify the stress state.

A summary of the element input is given in *CONTA172 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTA172 Input Summary

Nodes

I, J, K

Degrees of Freedom

UX, UY (if KEYOPT(1) = 0)
UX, UY, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS
See *Table 1, "CONTA172 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU, EMIS (**MP** command)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Surface Loads

Convection, Face 1 (I-J-K)
Heat Flux, Face 1 (I-J-K)

Special Features

Nonlinear
Large deflection
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY

1 --
UX, UY, TEMP

2 --
TEMP

3 --
UX, UY, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, VOLT

6 --
VOLT

7 --
AZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Stress state when superelements are present:

0 --
Use with h-elements (no superelements)

1 --
Axisymmetric (use with superelements only)

2 --
Plane stress/Plane strain (use with superelements only)

3 --
Plane stress with thickness input (use with superelements only)

KEYOPT(4)

Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint, set KEYOPT(4) = 2 for a rigid surface constraint. See Surface-based Constraints for more information.

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. Activated only if **SOLCONTROL**,ON,ON at the procedure level.

KEYOPT(8)

Asymmetric contact selection:

0 --

No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)

Contact stiffness update:

0 --

Each load step if FKN is redefined during load step (pair based).

1 --

Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --

Each iteration based on current mean stress of underlying elements (pair based).

3 --

Each load step if FKN is redefined during load step (individual element based).

4 --

Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --

Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)

Beam/Shell thickness effect:

0 --
Exclude

1 --
Include

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA172 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide . . .***
1	R1	Target circle radius	Defining the Target Surface
2	R2	Superelement thickness	Defining the Target Surface
3	FKN	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection

CONTA172 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

- Additional element output as shown in *Table 2, "CONTA172 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2, "CONTA172 Element Output Definitions" gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 2 CONTA172 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J	Y	Y
XC, YC	Location where results are reported	Y	*5*
TEMP	Temperatures T(I), T(J)	Y	Y
LENGTH	Element length	Y	-
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid, shell, or beam element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*
NX, NY	Surface normal vector components	Y	-
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact pressure	Y	Y
CONT:SFRIC	Tangential contact stress	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU	Friction coefficient	Y	-
CONT:SLIDE	Total accumulated sliding (algebraic sum)	*3*	*3*
CONT:ASLIDE	Total accumulated sliding (absolute sum)	*3*	*3*

Name	Definition	O	R
TOLN	Penetration tolerance	Y	Y
CONT:STOTAL	Total stress SQRT (PRES**2+SFRIC**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX	Contact element force-x component	-	*4*
CNFY	Contact element force-Y component	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
FDDIS	Frictional energy dissipation	*6*	*6*
FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
CAREA	Contacting area	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of STAT and OLDST are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system.
5. Available only at centroid as a ***GET** item.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Table 3, "CONTA172 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CONTA172 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTA172 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K
sequence number for data at nodes I, J, K

Table 3 CONTA172 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PRES	SMISC	5	1	2
SFRIC	SMISC	-	3	4
FLUX	SMISC	-	6	7
FDDIS	SMISC	-	8	9
FXCV	SMISC	-	10	11
FXRD	SMISC	-	12	13
FXCD	SMISC	-	14	15
FXNP	SMISC	-	16	17
JCONT	SMISC	-	18	19
CCONT	SMISC	-	18	19
HJOU	SMISC	-	20	21
STAT[*1*]	NMISC	19	1	2
OLDST	NMISC	-	3	4
PENE[*2*]	NMISC	-	5	6
DBA	NMISC	-	7	8
SLIDE	NMISC	-	9	10
KN	NMISC	-	11	12
KT	NMISC	-	13	14

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
TOLN	NMISC	-	15	16
IGAP	NMISC	-	17	18
PINB	NMISC	20	-	-
CNFX	NMISC	21	-	-
CNFY	NMISC	22	-	-
ISEG	NMISC	-	23	24
CAREA	NMISC	-	27	28
MU	NMISC	-	29	30
DTSTART	NMISC	-	31	32
DPARAM	NMISC	-	33	34
TEMPS	NMISC	-	37	38
TEMPT	NMISC	-	39	40
CONV	NMISC	-	41	42
RAC	NMISC	-	43	44
TCC	NMISC	-	45	46
CNFH	NMISC	47	-	-
ECURT	NMISC	48	-	-
ECHAR	NMISC	48	-	-
ECC	NMISC	-	49	50
VOLTS	NMISC	-	51	52
VOLTT	NMISC	-	53	54
CNOS	NMISC	-	55	56
TNOP	NMISC	-	57	58
SLTO	NMISC	-	59	60
ELSI	NMISC	-	67	68
DENERI	NMISC	-	69	70
DENERII	NMISC	-	71	72

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value
3. Contact element forces are defined in the global Cartesian system

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface

CNOS	Total number of contact status changes during substep

CONTA172 Assumptions and Restrictions

- The 2-D contact element must be defined in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- This 2-D contact element works with any 3-D elements in your model.
- Do not use this element in any model that contains axisymmetric harmonic elements.
- Node numbering must coincide with the external surface of the underlying solid element or with the original elements comprising the superelement.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- When nodal detection is used and the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.
- This element allows birth and death and will follow the birth and death status of the underlying solid or target elements.

CONTA172 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed.
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The AZ DOF (KEYOPT(1) = 7) is not allowed.

CONTA173

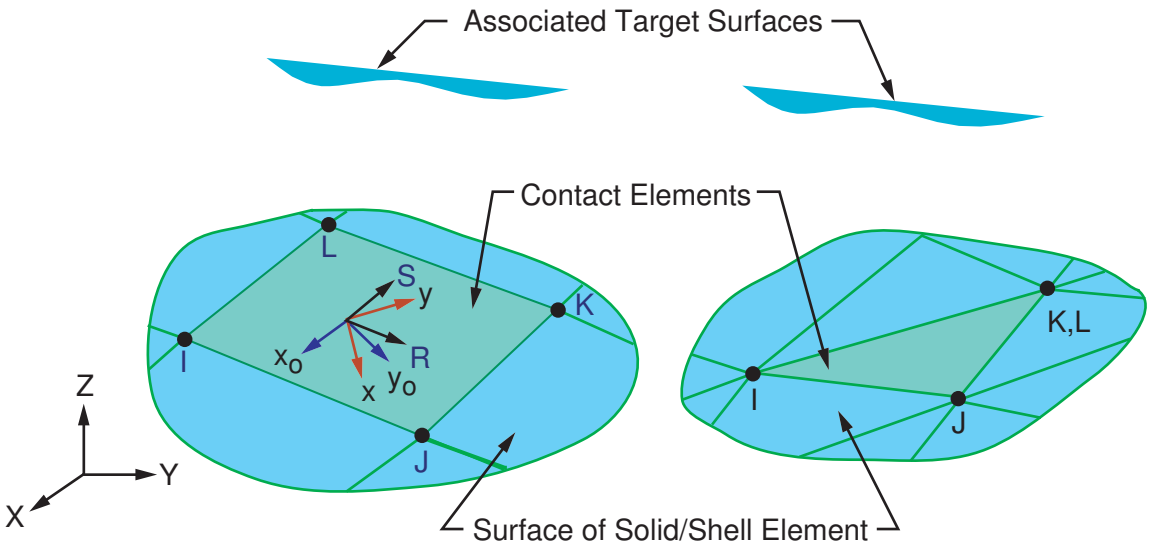
3-D 4-Node Surface-to-Surface Contact

MP ME ST PR PRN DS <> <> <> <> <> PP <>
Product Restrictions

CONTA173 Element Description

CONTA173 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on the surfaces of 3-D solid or shell elements without midside nodes (SOLID5, SOLID45, SOLID46, SOLID65, SOLID69, SOLID70, SOLID96, SOLID185, SOLSH190, VISCO107, SHELL28, SHELL41, SHELL43, SHELL57, SHELL63, SHELL131, SHELL157, SHELL181, and MATRIX50). It has the same geometric characteristics as the solid or shell element face with which it is connected (see *Figure 1, "CONTA173 Geometry"*). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA173 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other surface-to-surface contact elements (CONTA171, CONTA172, CONTA174) are also available.

Figure 1 CONTA173 Geometry



R = Element x-axis for isotropic friction

x_o = Element axis for orthotropic friction if **ESYS** is not supplied (parallel to global X-axis)

x = Element axis for orthotropic friction if **ESYS** is supplied

CONTA173 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA173 Geometry"*. The element is defined by four nodes (the underlying solid or shell element has no midside nodes). If the underlying solid or shell elements do have midside nodes, use CONTA174. The node ordering is consistent with the node ordering for the underlying solid or shell element. The positive normal is given by the right-hand rule going around the nodes of the element and is identical to the external normal direction of the underlying solid or shell element surface. For shell elements, the same nodal ordering between shell and contact elements defines upper surface contact; otherwise, it repres-

ents bottom surface contact. Remember the target surfaces must always be on its outward normal direction. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

CONTA173 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See *Section 2.5.17: Contact Friction* for more information.)

For isotropic friction, the applicable coordinate system is the default element coordinate system (noted by the R and S axes in the above figure).

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. (These are depicted by the x_o and x axes in the above figure.) The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact surface. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

The 3-D contact surface elements are associated with the 3-D target segment elements (TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

A summary of the element input is given in *CONTA173 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTA173 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ (if KEYOPT(1) = 0)
UX, UY, UZ, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, UZ, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, UZ, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
MAG (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC
See *Table 1, "CONTA173 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU, EMIS (**MP** command)
FRIC (**TB** command; see *Section 2.5.17: Contact Friction*)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Surface Loads

Convection, Face 1 (I-J-K-L)
Heat Flux, Face 1 (I-J-K-L)

Special Features

Nonlinear
Large deflection
Isotropic or orthotropic friction
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
MAG

KEYOPT(2)

Contact algorithm:

0 --

Augmented Lagrangian (default)

1 --

Penalty function

2 --

Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --

Lagrange multiplier on contact normal and penalty on tangent

4 --

Pure Lagrange multiplier on contact normal and tangent

KEYOPT(4)

Location of contact detection point:

0 --

On Gauss point (for general cases)

1 --

On nodal point - normal from contact surface

2 --

On nodal point - normal to target surface

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint, set KEYOPT(4) = 2 for a rigid surface constraint. See Surface-based Constraints for more information.

KEYOPT(5)

CNOF/ICONT automated adjustment:

0 --

No automated adjustment

1 --

Close gap with auto CNOF

2 --

Reduce penetration with auto CNOF

3 --

Close gap/reduce penetration with auto CNOF

4 --

Auto ICONT

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --

Use default range for stiffness updating

1 --

Make a nominal refinement to the allowable stiffness range

2 --

Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control:

0 --

No control

1 --

Automatic bisection of increment

2 --

Change in contact predictions made to maintain a reasonable time/load increment

3 --

Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. Activated only if **SOLCONTROL**,ON,ON at the procedure level.

KEYOPT(8)

Asymmetric contact selection:

0 --

No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)

Contact stiffness update:

0 --
Each load step if FKN is redefined during load step (pair based).

1 --
Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

3 --
Each load step if FKN is redefined during load step (individual element based).

4 --
Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --
Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)
Shell thickness effect:

0 --
Exclude

1 --
Include

KEYOPT(12)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA173 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance	Modeling Surface Interaction

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC	Magnetic contact permeance	Modeling Magnetic Contact

CONTA173 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTA173 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2, "CONTA173 Element Output Definitions" gives element output at the element level. In the results file, the nodal results are obtained from its closest integration point.

Table 2 CONTA173 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K, L	Y	Y
XC, YC, ZC	Location where results are reported	Y	*5*
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*

Name	Definition	O	R
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact pressure	Y	Y
TAUR/TAUS[*7*]	Tangential contact stresses	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU[*8*]	Friction coefficient	Y	-
TASS/TASR[*7*]	Total (algebraic sum) sliding in S and R directions	*3*	*3*
AASS/AASR[*7*]	Total (absolute sum) sliding in S and R directions	*3*	*3*
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress SQRT (TAUR**2+TAUS**2)	Y	Y
CONT:STOTAL	Total stress SQRT (PRES**2+TAUR**2+TAUS**2)	Y	Y
CONT:SLIDE	Total sliding SQRT (TASS**2+TASR**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX[*4*]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
FDDIS	Frictional energy dissipation	*6*	*6*
FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
CAREA	Contacting area	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y

Name	Definition	O	R
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
MCC	Magnetic contact permeance	Y	Y
MFLUX	Magnetic flux density	Y	Y
MAGS	Magnetic potential on contact node	Y	Y
MAGT	Magnetic potential on associated target	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system
5. Available only at centroid as a ***GET** item.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
7. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.
8. For orthotropic friction, an equivalent coefficient of friction is output.

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Table 3, "CONTA173 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CONTA173 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTA173 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 3 CONTA173 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	13	1	2	3	4
TAUR	SMISC	-	5	6	7	8
TAUS	SMISC	-	9	10	11	12
FLUX	SMISC	-	14	15	16	17
FDDIS	SMISC	-	18	19	20	21
FXCV	SMISC		22	23	24	25
FXRD	SMISC	-	26	27	28	29
FXCD	SMISC	-	30	31	32	33
FXNP	SMISC	-	34	35	36	37
JCONT	SMISC	-	38	39	40	41
CCONT	SMISC	-	38	39	40	41
HJOU	SMISC	-	42	43	44	45
MFLUX	SMISC	-	46	47	48	49
STAT[*1*]	NMISC	41	1	2	3	4
OLDST	NMISC	-	5	6	7	8
PENE[*2*]	NMISC	-	9	10	11	12
DBA	NMISC	-	13	14	15	16
TASR	NMISC	-	17	18	19	20
TASS	NMISC	-	21	22	23	24
KN	NMISC	-	25	26	27	28
KT	NMISC	-	29	30	31	32
TOLN	NMISC	-	33	34	35	36
IGAP	NMISC	-	37	38	39	40
PINB	NMISC	42	-	-	-	-
CNFX	NMISC	43	-	-	-	-
CNFY	NMISC	44	-	-	-	-
CNFZ	NMISC	45	-	-	-	-
ISEG	NMISC	-	46	47	48	49
AASR	NMISC	-	50	51	52	53
AASS	NMISC	-	54	55	56	57
CAREA	NMISC	-	58	59	60	61
MU	NMISC	-	62	63	64	65
DTSTART	NMISC	-	66	67	68	69
DPARAM	NMISC	-	70	71	72	73

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
TEMPS	NMISC	-	78	79	80	81
TEMPT	NMISC	-	82	83	84	85
CONV	NMISC	-	86	87	88	89
RAC	NMISC	-	90	91	92	93
TCC	NMISC	-	94	95	96	97
CNFH	NMISC	98	-	-	-	-
ECURT	NMISC	99	-	-	-	-
ECHAR	NMISC	99	-	-	-	-
ECC	NMISC	-	100	101	102	103
VOLTS	NMISC	-	104	105	106	107
VOLTT	NMISC	-	108	109	110	111
CNOS	NMISC	-	112	113	114	115
TNOP	NMISC	-	116	117	118	119
SLTO	NMISC	-	120	121	122	123
MCC	NMISC	-	124	125	126	127
MAGS	NMISC	-	128	129	130	131
MAGT	NMISC	-	132	133	134	135
ELSI	NMISC	-	136	137	138	139
DENERI	NMISC	-	140	141	142	143
DENERII	NMISC	-	144	145	146	147

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep

CONTA173 Assumptions and Restrictions

- The 3-D contact element must coincide with the external surface of the underlying solid or shell element or with the original elements comprising the superelement.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.

- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- This element allows birth and death and will follow the birth and death status of the underlying solid, shell, beam, or target elements.

CONTA173 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The MAG DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The MAG DOF (KEYOPT(1) = 7) is not allowed.

CONTA174

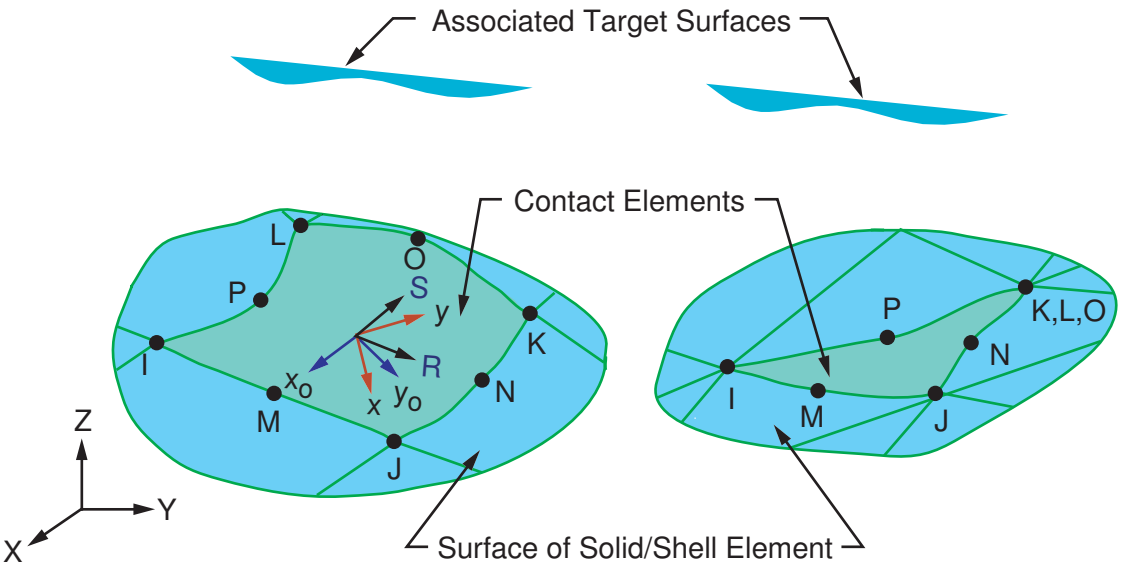
3-D 8-Node Surface-to-Surface Contact

MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

CONTA174 Element Description

CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on the surfaces of 3-D solid or shell elements with midside nodes (SOLID87, SOLID90, SOLID92, SOLID95, SOLID98, SOLID122, SOLID123, SOLID186, SOLID187, SOLID191, SOLID226, SOLID227, SOLID231, SOLID232, VISCO89, SHELL91, SHELL93, SHELL99, SHELL132, SHELL281, and MATRIX50). It has the same geometric characteristics as the solid or shell element face with which it is connected (see *Figure 1, "CONTA174 Geometry"* below). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA174 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other surface-to-surface contact elements (CONTA171, CONTA172, CONTA173) are also available.

Figure 1 CONTA174 Geometry



R = Element x-axis for isotropic friction

x_o = Element axis for orthotropic friction if **ESYS** is not supplied (parallel to global X-axis)

x = Element axis for orthotropic friction if **ESYS** is supplied

CONTA174 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA174 Geometry"*. The element is defined by eight nodes (the underlying solid or shell element has midside nodes). It can degenerate to a six node element depending on the shape of the underlying solid or shell elements. If the underlying solid or shell elements do not have midside nodes, use CONTA173 (you may still use CONTA174 but you must drop all midside nodes). See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes. The node ordering is consistent with the node ordering for the underlying solid or shell element. The positive

normal is given by the right-hand rule going around the nodes of the element and is identical to the external normal direction of the underlying solid or shell element surface. For shell elements, the same nodal ordering between shell and contact elements defines upper surface contact; otherwise, it represents bottom surface contact. Remember the target surfaces must always be on its outward normal direction. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

CONTA174 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See *Section 2.5.17: Contact Friction* for more information.)

For isotropic friction, the applicable coordinate system is the default element coordinate system (noted by the R and S axes in the above figure).

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. (These are depicted by the x_o and x axes in the above figure.) The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact surface. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

The 3-D contact surface elements (CONTA173 and CONTA174) are associated with the 3-D target segment elements (TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine two target surfaces into one (targets that share the same real constant numbers).

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

A summary of the element input is given in *CONTA174 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

CONTA174 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ (if KEYOPT(1) = 0)
UX, UY, UZ, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, UZ, TEMP, VOLT (if KEYOPT(1) = 3)

TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, UZ, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
MAG (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC
See *Table 1, "CONTA174 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU, EMIS (**MP** command)
FRIC (**TB** command; see *Section 2.5.17: Contact Friction*)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Surface Loads

Convection, Face 1 (I-J-K-L)
Heat Flux, Face 1 (I-J-K-L)

Special Features

Nonlinear
Large deflection
Isotropic or orthotropic friction
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
MAG

KEYOPT(2)
Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(4)
Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint, set KEYOPT(4) = 2 for a rigid surface constraint. See Surface-based Constraints for more information.

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --

Make a nominal refinement to the allowable stiffness range

2 --

Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control:

0 --

No control

1 --

Automatic bisection of increment

2 --

Change in contact predictions made to maintain a reasonable time/load increment

3 --

Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. Activated only if **SOLCONTROL**,ON,ON at the procedure level.

KEYOPT(8)

Asymmetric contact selection:

0 --

No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)

Contact stiffness update:

0 --

Each load step if FKN is redefined during load step (pair based).

1 --

Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --

Each iteration based on current mean stress of underlying elements (pair based).

3 --

Each load step if FKN is redefined during load step (individual element based).

4 --

Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --

Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)

Shell thickness effect:

0 --

Exclude

1 --

Include

KEYOPT(12)

Behavior of contact surface:

0 --

Standard

1 --

Rough

2 --

No separation (sliding permitted)

3 --

Bonded

4 --

No separation (always)

5 --

Bonded (always)

6 --

Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA174 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance	Modeling Surface Interaction

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC	Magnetic contact permeance	Modeling Magnetic Contact

CONTA174 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTA174 Element Output Definitions"*

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2, "CONTA174 Element Output Definitions" gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 2 CONTA174 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K, L, M, N, O, P	Y	Y
XC, YC, ZC	Location where results are reported	Y	*5*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*

Name	Definition	O	R
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap from previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact pressure	Y	Y
TAUR/TAUS[*7*]	Tangential contact stresses	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU[*8*]	Friction coefficient	Y	-
TASS/TASR[*7*]	Total (algebraic sum) sliding in S and R directions	*3*	*3*
AASS/AASR[*7*]	Total (absolute sum) sliding in S and R directions	*3*	*3*
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress SQRT (TAUR**2+TAUS**2)	Y	Y
CONT:STOTAL	Total stress SQRT (PRES**2+TAUR**2+TAUS**2)	Y	Y
CONT:SLIDE	Total sliding SQRT (TASS**2 + TASR**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX	Contact element force-X component	-	*4*
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
FDDIS	Frictional energy dissipation	*6*	*6*
FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
CAREA	Contacting area	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y

Name	Definition	O	R
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
MCC	Magnetic contact permeance	Y	Y
MFLUX	Magnetic flux density	Y	Y
MAGS	Magnetic potential on contact node	Y	Y
MAGT	Magnetic potential on associated target	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system.
5. Available only at centroid as a ***GET** item.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
7. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.
8. For orthotropic friction, an equivalent coefficient of friction is output.

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Table 3, "CONTA174 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "CONTA174 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "CONTA174 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L,M,N,O,P
sequence number for data at nodes I,J,K,L,M,N,O,P

Table 3 CONTA174 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	13	1	2	3	4
TAUR	SMISC	-	5	6	7	8
TAUS	SMISC	-	9	10	11	12
FLUX	SMISC	-	14	15	16	17
FDDIS	SMISC	-	18	19	20	21
FXCV	SMISC		22	23	24	25
FXRD	SMISC	-	26	27	28	29
FXCD	SMISC	-	30	31	32	33
FXNP	SMISC	-	34	35	36	37
JCONT	SMISC	-	38	39	40	41
CCONT	SMISC	-	38	39	40	41
HJOU	SMISC	-	42	43	44	45
MFLUX	SMISC	-	46	47	48	49
STAT[*1*]	NMISC	41	1	2	3	4
OLDST	NMISC	-	5	6	7	8
PENE[*2*]	NMISC	-	9	10	11	12
DBA	NMISC	-	13	14	15	16
TASR	NMISC	-	17	18	19	20
TASS	NMISC	-	21	22	23	24
KN	NMISC	-	25	26	27	28
KT	NMISC	-	29	30	31	32
TOLN	NMISC	-	33	34	35	36
IGAP	NMISC	-	37	38	39	40
PINB	NMISC	42	-	-	-	-
CNFX	NMISC	43	-	-	-	-
CNFY	NMISC	44	-	-	-	-
CNFZ	NMISC	45	-	-	-	-
ISEG	NMISC	-	46	47	48	49
AASR	NMISC	-	50	51	52	53
AASS	NMISC	-	54	55	56	57
CAREA	NMISC	-	58	59	60	61
MU	NMISC	-	62	63	64	65
DTSTART	NMISC	-	66	67	68	69
DPARAM	NMISC	-	70	71	72	73

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
TEMPS	NMISC	-	78	79	80	81
TEMPT	NMISC	-	82	83	84	85
CONV	NMISC	-	86	87	88	89
RAC	NMISC	-	90	91	92	93
TCC	NMISC	-	94	95	96	97
CNFH	NMISC	98	-	-	-	-
ECURT	NMISC	99	-	-	-	-
ECHAR	NMISC	99	-	-	-	-
ECC	NMISC	-	100	101	102	103
VOLTS	NMISC	-	104	105	106	107
VOLTT	NMISC	-	108	109	110	111
CNOS	NMISC	-	112	113	114	115
TNOP	NMISC	-	116	117	118	119
SLTO	NMISC	-	120	121	122	123
MCC	NMISC	-	124	125	126	127
MAGS	NMISC	-	128	129	130	131
MAGT	NMISC	-	132	133	134	135
ELSI	NMISC	-	136	137	138	139
DENERI	NMISC	-	140	141	142	143
DENERII	NMISC	-	144	145	146	147

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep

CONTA174 Assumptions and Restrictions

- The 3-D contact element must coincide with the external surface of the underlying solid or shell element.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.

- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- This element allows birth and death and will follow the birth and death status of the underlying solid, shell, beam or target elements.

CONTA174 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The MAG DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The MAG DOF (KEYOPT(1) = 7) is not allowed.

CONTA175

2-D/3-D Node-to-Surface Contact

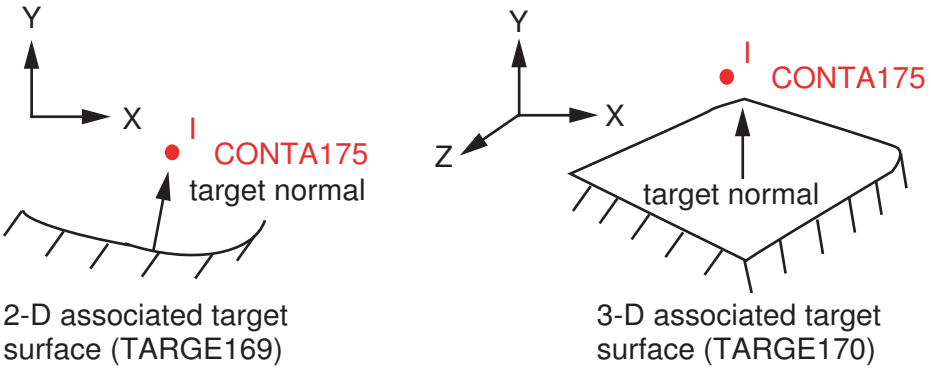
MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

CONTA175 Element Description

CONTA175 may be used to represent contact and sliding between two surfaces (or between a node and a surface, or between a line and a surface) in 2-D or 3-D. The element is applicable to 2-D or 3-D structural and coupled field contact analyses. This element is located on the surfaces of solid, beam, and shell elements. 3-D solid and shell elements with midside nodes are supported for bonded and no separation contact. For other contact types, lower order solid and shell elements are recommended.

Contact occurs when the element surface penetrates one of the target segment elements (TARGE169, TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA175 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 CONTA175 Geometry



CONTA175 Input Data

The geometry is shown in *Figure 1, "CONTA175 Geometry"*. The element is defined by one node. The underlying elements can be 2-D or 3-D solid, shell, or beam elements. The 3-D underlying solid or shell elements must have no midside nodes. CONTA175 represents 2-D or 3-D contact depending on whether the associated 2-D (TARGE169) or 3-D (TARGE170) segments are used. Remember, contact can occur only when the outward normal direction of the 2-D or 3-D target surface points to the contact surface. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

CONTA175 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See *Section 2.5.17: Contact Friction* for more information.)

For isotropic friction, the default element coordinate system (based on node connectivity of the underlying elements) is used. For orthotropic friction, the global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. The principal directions are computed on the target surface and then projected onto the contact element (node). The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the target surface. The second principal direction is

defined by taking a cross product of the first principal direction and the target normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

The contact surface elements are associated with the target segment elements (TARGE169, TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information. If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. *Chapter 4: Node-to-Surface Contact* discusses CONTA175 specifically, including the use of real constants and KEYOPTs.

A summary of the element input is given in *CONTAC175 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTAC175 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, (UZ) (if KEYOPT(1) = 0
UX, UY, (UZ), TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, (UZ), TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, (UZ), VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (2-D), MAG (3-D) (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC
See *Table 1, "CONTA175 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU, EMIS (**MP** command)
FRIC (**TB** command; see *Section 2.5.17: Contact Friction*)

CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Special Features

Nonlinear
Large deflection
Isotropic or orthotropic friction
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
AZ (2-D) or MAG (3-D)

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Contact model:

0 --
Contact force based model (default)

1 --
Contact traction model

KEYOPT(4)
Contact normal direction:

0 --
Normal to target surface (default)

1 --
Normal from contact nodes

2 --
Normal from contact nodes (used for shell/beam bottom surface contact when shell/beam thickness is accounted for)

3 --
Normal to target surface (used for shell/beam bottom surface contact when shell/beam thickness is accounted for)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 0 for a rigid surface constraint, set KEYOPT(4) = 1 for a force-distributed constraint. See Surface-based Constraints for more information.

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. It is activated only if **SOLCONTROL**,ON,ON at the procedure level.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)
Effect of initial penetration or gap:

0 --
Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)
Contact Stiffness Update:

0 --
Each load step if FKN is redefined during load step (pair based).

1 --
Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

3 --
Each load step if FKN is redefined during load step (individual element based).

4 --
Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --
Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)
Shell Thickness Effect (only for real constant based thickness input):

0 --
Exclude

1 --
Include

KEYOPT(12)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA175 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure [*1*]	Chattering Control Parameters

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC	Magnetic contact permeance	Modeling Magnetic Contact

1. For the force-based model (KEYOPT(3) = 0), TNOP is the allowable tensile contact force.

CONTA175 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTA175 Element Output Definitions"*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTA175 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I	Y	Y
XC, YC, (ZC)	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	AREA for 3-D, Length for 2-D	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-

Name	Definition	O	R
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact pressure	*2*	*2*
TAUR/TAUS[*8*]	Tangential contact stresses	*2*	*2*
KN	Current normal contact stiffness (units: Force/Length for contact force model, units: Force/Length^3 for contract traction model)	*5*	*5*
KT	Current tangent contact stiffness (same units as KN)	*5*	*5*
MU[*9*]	Friction coefficient	Y	-
TASS/TASR[*8*]	Total (algebraic sum) sliding in S and R directions (3-D only)	*3*	*3*
AASS/AASR[*8*]	Total (absolute sum) sliding in S and R directions (3-D only)	*3*	*3*
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress SQRT (TAUR**2+TAUS**2) (3-D only)	*2*	*2*
CONT:STOTAL	Total stress SQRT (PRES**2+TAUR**2+TAUS**2) (3-D only)	*2*	*2*
CONT:SLIDE	Total sliding SQRT (TASS**2+TASR**2) (3-D only)	Y	Y
NX, NY	Surface normal vector components (2-D only)	Y	-
CONT:SFRIC	Tangential contact stress (2-D only)	*2*	*2*
CONT:SLIDE	Total accumulated sliding (algebraic sum) (2-D only)	*3*	*3*
CONT:ASLIDE	Total accumulated sliding (absolute sum) (2-D only)	*3*	*3*
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX[*4*]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component (3-D only)	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	*6*	*6*
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
FDDIS	Frictional energy dissipation	*7*	*7*
FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
CAREA	Contacting area	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	*6*	*6*

Name	Definition	O	R
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	*2*	*2*
SLTO	Allowable elastic slip	Y	Y
MCC	Magnetic contact permeance	*6*	*6*
MFLUX	Magnetic flux density	Y	Y
AZS/MAGS	2-D/3-D Magnetic potential on contact node	Y	Y
AZT/MAGT	2-D/3-D Magnetic potential on associated target	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. For the force-based model (KEYOPT(3) = 0), the unit of the quantities is FORCE. For the traction-based model (KEYOPT(3) = 1), the unit is FORCE/AREA.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system
5. For the force-based model, the unit of stiffness is FORCE/LENGTH. For the traction-ased model, the unit is FORCE/LENGTH^3.
6. The units of TCC, ECC, and MCC in the traction-based model should be the units of TCC, ECC, and MCC of the force-based model per area.
7. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
8. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.
9. For orthotropic friction, an equivalent coefficient of friction is output.

Table 3, "CONTA175 (3-D) Item and Sequence Numbers" and *Table 4, " CONTA175 (2-D) Item and Sequence Numbers"* list outputs available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in the tables below:

Name
output quantity as defined in *Table 2, "CONTA175 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

I

sequence number for data at nodes I

Table 3 CONTA175 (3-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
PRES	SMISC	13	1
TAUR	SMISC	-	5
TAUS	SMISC	-	9
FLUX	SMISC	-	14
FDDIS	SMISC	-	18
FXCV	SMISC		22
FXRD	SMISC	-	26
FXCD	SMISC	-	30
FXNP	SMISC	-	34
JCONT	SMISC	-	38
CCONT	SMISC	-	38
HJOU	SMISC	-	42
MFLUX	SMISC	-	46
STAT[*1*]	NMISC	41	1
OLDST	NMISC	-	5
PENE[*2*]	NMISC	-	9
DBA	NMISC	-	13
TASR	NMISC	-	17
TASS	NMISC	-	21
KN	NMISC	-	25
KT	NMISC	-	29
TOLN	NMISC	-	33
IGAP	NMISC	-	37
PINB	NMISC	42	-
CNFX	NMISC	43	-
CNFY	NMISC	44	-
CNFZ	NMISC	45	-
ISEG	NMISC	-	46
AASR	NMISC	-	50
AASS	NMISC	-	54
CAREA	NMISC	-	58
MU	NMISC	-	62
DTSTART	NMISC	-	66
DPARAM	NMISC	-	70
TEMPS	NMISC	-	78
TEMPT	NMISC	-	82

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
CONV	NMISC	-	86
RAC	NMISC	-	90
TCC	NMISC	-	94
CNFH	NMISC	98	-
ECURT	NMISC	99	-
ECHAR	NMISC	99	-
ECC	NMISC	-	100
VOLTS	NMISC	-	104
VOLTT	NMISC	-	108
CNOS	NMISC	-	112
TNOP	NMISC	-	116
SLTO	NMISC	-	120
MCC	NMISC	-	124
MAGS	NMISC	-	128
MAGT	NMISC	-	132
ELSI	NMISC	-	136
DENERI	NMISC	-	140
DENERII	NMISC	-	144

Table 4 CONTA175 (2-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
PRES	SMISC	5	1
SFRIC	SMISC	-	3
FLUX	SMISC	-	6
FDDIS	SMISC	-	8
FXCV	SMISC	-	10
FXRD	SMISC	-	12
FXCD	SMISC	-	14
FXNP	SMISC	-	16
JCONT	SMISC	-	18
CCONT	SMISC	-	18
HJOU	SMISC	-	20
MFLUX	SMISC	-	22
STAT[*1*]	NMISC	19	1
OLDST	NMISC	-	3
PENE[*2*]	NMISC	-	5
DBA	NMISC	-	7
SLIDE	NMISC	-	9
KN	NMISC	-	11

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
KT	NMISC	-	13
TOLN	NMISC	-	15
IPENE	NMISC	-	17
PINB	NMISC	20	-
CNFX	NMISC	21	-
CNFY	NMISC	22	-
ISEG	NMISC	-	23
CAREA	NMISC	-	27
MU	NMISC	-	29
DTSTART	NMISC	-	31
DPARAM	NMISC	-	33
TEMPS	NMISC	-	37
TEMPT	NMISC	-	39
CONV	NMISC	-	41
RAC	NMISC	-	43
TCC	NMISC	-	45
CNFH	NMISC	47	-
ECURT	NMISC	48	-
ECHAR	NMISC	48	-
ECC	NMISC	-	49
VOLTS	NMISC	-	51
VOLTT	NMISC	-	53
CNOS	NMISC	-	55
TNOP	NMISC	-	57
SLTO	NMISC	-	59
MCC	NMISC	-	61
AZS	NMISC	-	63
AZT	NMISC	-	65
ELSI	NMISC	-	67
DENERI	NMISC	-	69
DENERII	NMISC	-	71

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure for the traction-based model. Contact normal force for the force-based model.

SFRIC	Contact friction stress for the traction-based model. Friction force for the force-based model.
STOT	Contact total stress (pressure plus friction) for the traction-based model. Total contact force for the force-based model.
SLIDE	Contact sliding distance
GAP	Contact gap distance
CNOS	Total number of contact status changes during substep

CONTA175 Assumptions and Restrictions

- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- This element allows birth and death and will follow the birth and death status of the underlying solid, shell, beam, or target elements.
- When the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.

CONTA175 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed.
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ (2-D) and MAG (3-D) DOFs (KEYOPT(1) = 7) are not allowed.

ANSYS Mechanical

- The AZ (2-D) and MAG (3-D) DOFs (KEYOPT(1) = 7) are not allowed.

CONTA176

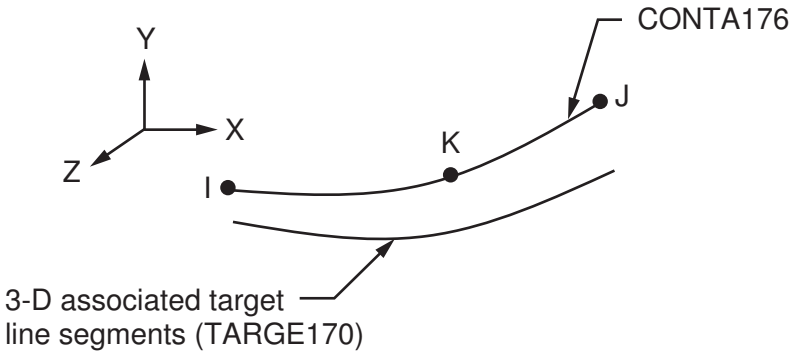
3-D Line-to-Line Contact

MP ME ST PR PRN DS DSS <> EM <> <> PP <>
Product Restrictions

CONTA176 Element Description

CONTA176 is used to represent contact and sliding between 3-D line segments (TARGE170) and a deformable line segment, defined by this element. The element is applicable to 3-D beam-beam structural contact analyses. This element is located on the surfaces of 3-D beam or pipe elements with or without midside nodes (BEAM4, BEAM24, BEAM188, BEAM189, PIPE16, PIPE20). Contact occurs when the element surface penetrates one of the 3-D straight line or parabolic line segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA176 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. To model beam-to-surface contact, use the line-to-surface contact element, CONTA177.

Figure 1 CONTA176 Geometry



CONTA176 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA176 Geometry"*. The element is defined by two nodes (if the underlying beam element does not have a midside node) or three nodes (if the underlying beam element has a midside node). The element x-axis is along the I-J line of the element. Correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered in a sequence that defines a continuous line. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

Three different scenarios can be modeled by CONTA176:

- Internal contact where one beam (or pipe) slides inside another hollow beam (or pipe) (see *Figure 2, "Beam Sliding Inside a Hollow Beam"*)
- External contact between two beams that lie next to each other and are roughly parallel (see *Figure 3, "Parallel Beams in Contact"*)
- External contact between two beams that cross (see *Figure 4, "Crossing Beams in Contact"*)

Use KEYOPT(3) = 0 for the first two scenarios (internal contact and parallel beams). In both cases, the contact condition is only checked at contact nodes.

Use KEYOPT(3) = 1 for the third scenario (beams that cross). In this case, the contact condition is checked along the entire length of the beams. The beams with circular cross sections are assumed to come in contact in a point-wise manner. Each contact element can potentially contact no more than one target element.

Figure 2 Beam Sliding Inside a Hollow Beam

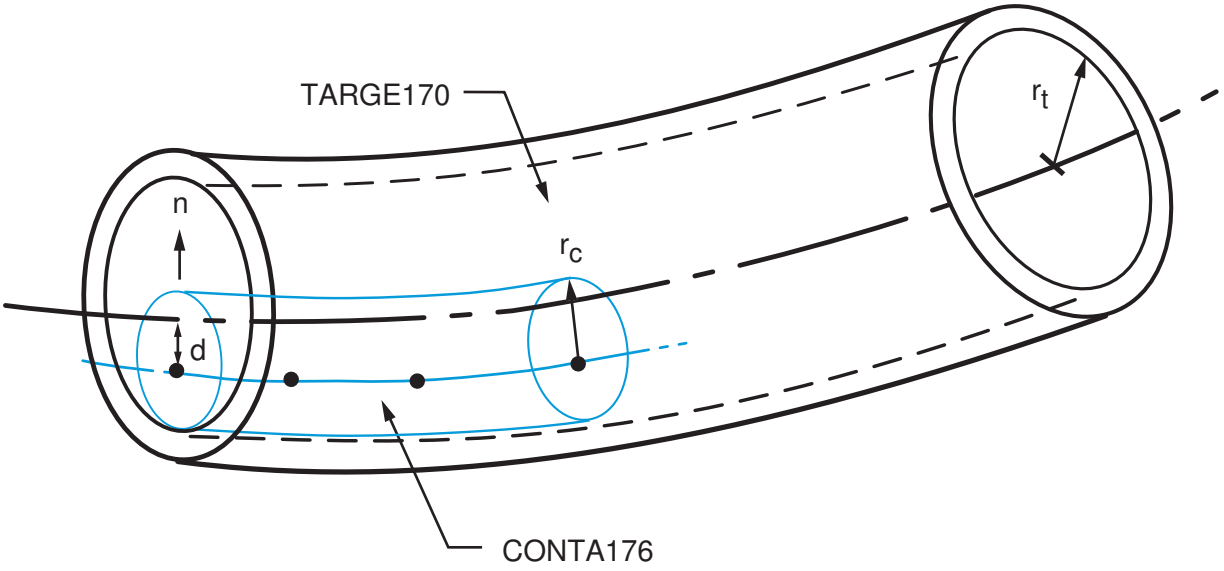


Figure 3 Parallel Beams in Contact

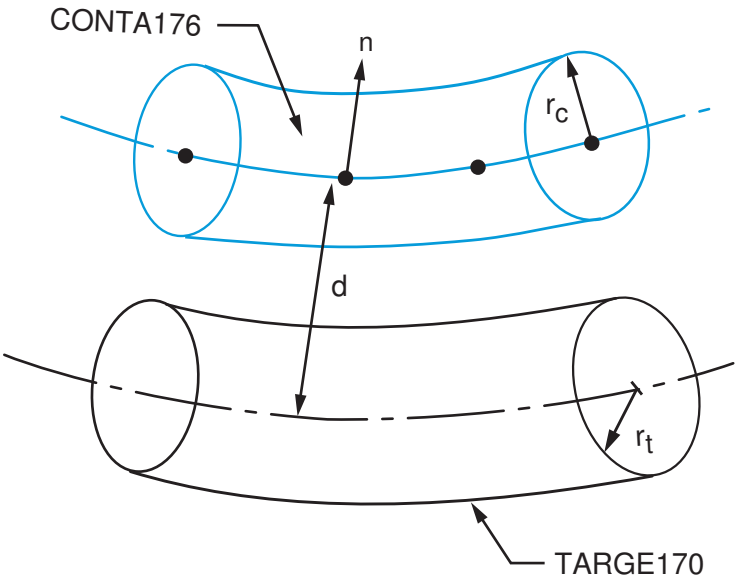
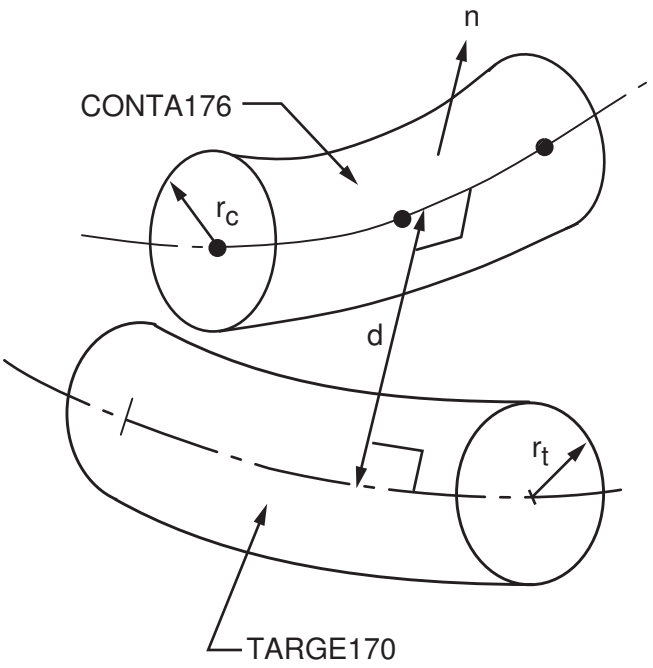


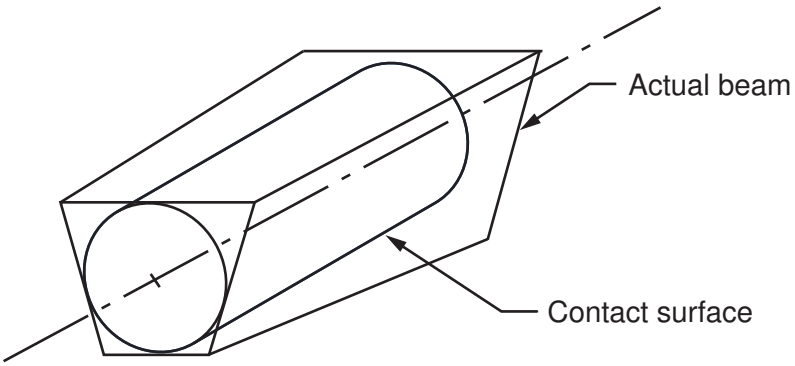
Figure 4 Crossing Beams in Contact



The 3-D line-to-line contact elements are associated with the target line segment elements (LINE or PARA segment types for TARGE170) via a shared real constant set. The contact/target surface is assumed to be the surface of a cylinder. For a general beam cross section, use an equivalent circular beam (see *Figure 5, "Equivalent Circular Cross Section"*). Use the first real constant, R1, to define the radius on the target side (target radius r_t). Use the second real constant, R2, to define the radius on the contact side (contact radius r_c). Follow these guidelines to define the equivalent circular cross section:

- Determine the smallest cross section along the beam axis.
- Determine the largest circle embedded in that cross section.

Figure 5 Equivalent Circular Cross Section



The target radius can be entered as either a negative or positive value. Use a negative value when modeling internal contact (a beam sliding inside a hollow beam, or pipe sliding inside another pipe), with the input value equal to the inner radius of the outer beam (see *Figure 2, "Beam Sliding Inside a Hollow Beam"*). Use a positive value when modeling contact between the exterior surfaces of two cylindrical beams.

For the case of internal contact, the inner beam should usually be considered the contact surface and the outer beam should be the target surface. The inner beam can be considered as the target surface only when the inner beam is much stiffer than the outer beam.

Contact is detected when two circular beams touch or overlap each other. The non-penetration condition for beams with a circular cross section can be defined as follows.

For internal contact:

$$g = \left| r_t - r_c \right| - d \le 0$$

and for external contact:

$$g = d - (r_c + r_t) \le 0$$

where r_c and r_t are the radii of the cross sections of the beams on the contact and target sides, respectively; and d is the minimal distance between the two beams which also determines the contact normal direction (see *Figure 4, "Crossing Beams in Contact"*). Contact occurs for negative values of g.

ANSYS looks for contact only between contact and target surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information. If more than one target surface will make contact with the same boundary of beam elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA176 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See *Section 2.5.17: Contact Friction* for more information.) The local element coordinates based on the nodal connectivity are used as principal directions. Local element coordinates defined using the **ESYS** command are ignored.

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. *Chapter 5: 3-D Beam-to-Beam Contact* discusses CONTA176 specifically, including the use of real constants and KEYOPTs.

The following table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

CONTA176 Input Summary

Nodes

I, J, (K)

Degrees of Freedom

UX, UY, UZ

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,

PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, (Blank), (Blank), (Blank), (Blank), (Blank),
(Blank), (Blank), FACT, DC, SLTO, TNOP,
TOLS, (Blank)
See *Table 1, "CONTA176 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU (**MP** command)
FRIC (**TB** command; see *Section 2.5.17: Contact Friction*)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Special Features

Nonlinear
Large deflection
Isotropic or orthotropic friction
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom. Currently, the default (UX, UY, UZ) is the only valid option:

0 --
UX, UY, UZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Beam contact type:

0 --
Parallel beams or beam inside beam

1 --
Crossing beams

KEYOPT(4)

Type of surface-based constraint (see Surface-based Constraints for more information):

0 --
Rigid surface constraint

1 --
Force-distributed constraint

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. It is activated only if **SOLCONTROL**,ON,ON is issued at the procedure level.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)

Contact Stiffness Update:

0 --

Each load step if FKN is redefined during load step (pair based).

1 --

Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --

Each iteration based on current mean stress of underlying elements (pair based).

3 --

Each load step if FKN is redefined during load step (individual element based).

4 --

Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --

Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(12)

Behavior of contact surface:

0 --

Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA176 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide . . .***
1	R1	Target radius	Real Constants R1, R2
2	R2	Contact radius	Real Constants R1, R2
3	FKN[*1*]	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT[*1*]	Tangent penalty stiffness factor	Determining Contact Stiffness

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
13	COHE	Contact cohesion	Choosing a Friction Model
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact force	Chattering Control Parameters
25	TOLS	Target edge extension factor	Real Constant TOLS

1. The units of real constants FKN and FKT have a factor of AREA with respect to those used in the surface-to-surface contact elements. See Performing a 3-D Beam-to-Beam Contact Analysis for more information.

CONTA176 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTA176 Element Output Definitions"*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTA176 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K	Y	Y
XC, YC, ZC	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	Length	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying beam element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y

Name	Definition	O	R
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact force	*2*	*2*
TAUR/TAUS[*7*]	Tangential contact stresses	*2*	*2*
KN	Current normal contact stiffness (units: Force/Length)	*5*	*5*
KT	Current tangent contact stiffness (same units as KN)	*5*	*5*
MU[*8*]	Friction coefficient	Y	-
TASS/TASR[*7*]	Total (algebraic sum) sliding in S and R directions	*3*	*3*
AASS/AASR[*7*]	Total (absolute sum) sliding in S and R directions	*3*	*3*
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress SQRT (TAUR**2+TAUS**2)	*2*	*2*
CONT:STOTAL	Total stress SQRT (PRES**2+TAUR**2+TAUS**2)	*2*	*2*
CONT:SLIDE	Total sliding SQRT (TASS**2+TASR**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX[*4*]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
CAREA	Contacting area	-	Y
FDDIS	Frictional energy dissipation	*6*	*6*
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact force	*2*	*2*
SLTO	Allowable elastic slip	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. The unit of the quantities is FORCE.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system

5. The unit of stiffness is FORCE/LENGTH.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
7. For the case of orthotropic friction in contact between beams, components are defined in the global Cartesian system.
8. For orthotropic friction, an equivalent coefficient of friction is output.

The following table lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information.

Name
output quantity as defined in *Table 2, "CONTA176 Element Output Definitions"*

Item
predetermined item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J, K
sequence number for data at nodes I, J, K

Table 3 CONTA176 (3-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
PRES	SMISC	13	1	2	3
TAUR	SMISC	-	5	6	7
TAUS	SMISC	-	9	10	11
FDDIS	SMISC	-	18	19	20
STAT[*1*]	NMISC	41	1	2	3
OLDST	NMISC	-	5	6	7
PENE[*2*]	NMISC	-	9	10	11
DBA	NMISC	-	13	14	15
TASR	NMISC	-	17	18	19
TASS	NMISC	-	21	22	23
KN	NMISC	-	25	26	27
KT	NMISC	-	29	30	31
TOLN	NMISC	-	33	34	35
IGAP	NMISC	-	37	38	39
PINB	NMISC	42	-	-	-
CNFX	NMISC	43	-	-	-
CNFY	NMISC	44	-	-	-
CNFZ	NMISC	45	-	-	-
ISEG	NMISC	-	46	47	48
AASR	NMISC	-	50	51	52
AASS	NMISC	-	54	55	56
CAREA	NMISC	-	58	59	60

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
MU	NMISC	-	62	63	64
DTSTART	NMISC	-	66	67	68
DPARAM	NMISC	-	70	71	72
CNOS	NMISC	-	112	113	114
TNOP	NMISC	-	116	117	118
SLTO	NMISC	-	120	121	122
ELSI	NMISC	-	136	137	138
DENERI	NMISC	-	140	141	142
DENERII	NMISC	-	144	145	146

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

CONTA176 Assumptions and Restrictions

- The main restriction is the assumption of constant circular beam cross section. The contact radius is assumed to be the same for all elements in the contact pair.
- For KEYOPT(3) = 1 (crossing beams), contact between the beams is pointwise, and each contact element contacts no more than one target element.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Augmented Lagrangian method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- This element allows birth and death and will follow the birth and death status of the underlying beam, pipe, or target elements.

CONTA176 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property (input via **MP**,MU or **TB**,FRIC) is not allowed.
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

CONTA177

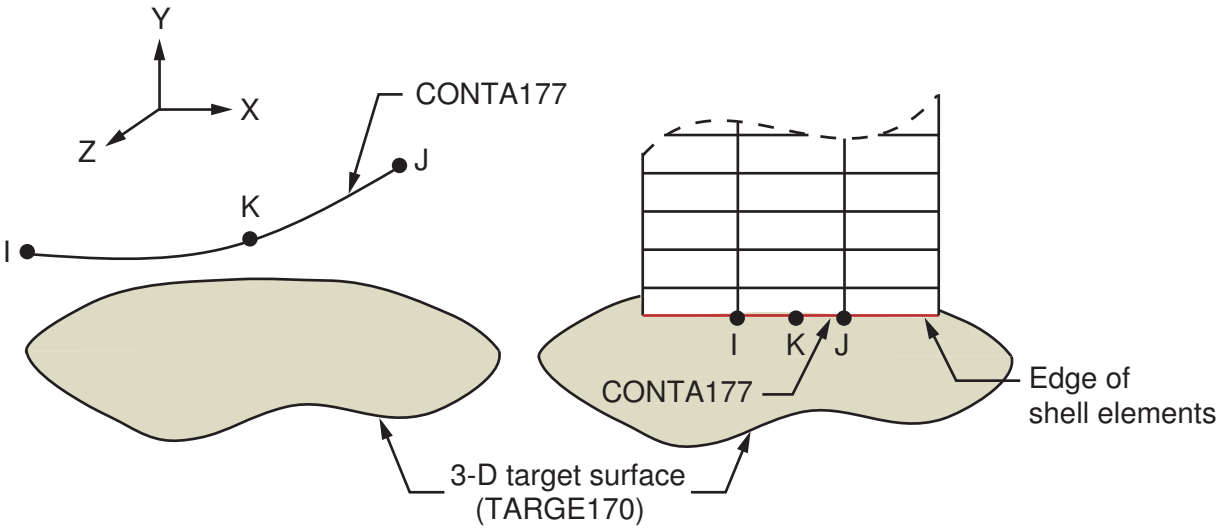
3-D Line-to-Surface Contact

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

CONTA177 Element Description

CONTA177 is used to represent contact and sliding between 3-D surface segments (TARGE170) and a deformable line segment, defined by this element. The element is applicable to 3-D beam-to-solid and 3-D shell edge-to-solid structural contact analyses. This element is located on the surfaces of 3-D beam or pipe elements with or without midside nodes (BEAM4, BEAM24, BEAM188, BEAM189, PIPE16, PIPE20). It can also be located on the edge of 3-D shell elements with or without midside nodes, such as SHELL181 and SHELL281. Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA177 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. To model beam-to-beam contact, use the line-to-line contact element, CONTA176.

Figure 1 CONTA177 Geometry



CONTA177 Input Data

The geometry and node locations are shown in *Figure 1, "CONTA177 Geometry"*. The element is defined by two nodes (if the underlying beam or shell element does not have a midside node) or three nodes (if the underlying beam or shell element has a midside node). The element x-axis is along the I-J line of the element. Correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered in a sequence that defines a continuous line. See *Section 3.7.3: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

ANSYS looks for contact only between contact and target surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable "surfaces" (beam or shell edge) must be represented by a contact surface. See *Section 3.5: Designating Contact and Target Surfaces* in the *Contact Technology Guide* for more information. If more than one target surface will make contact with the same boundary of line elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA177 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See *Section 2.5.17: Contact Friction* for more information.) The local element coordinates based on the nodal connectivity are used as principal directions. Local element coordinates defined using the **ESYS** command are ignored.

To model separation of bonded contact with KEYOPT(12) = 4, 5, or 6, use the **TB** command with the CZM label. See *Chapter 11: Debonding* in the *Contact Technology Guide* for more information.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. *Chapter 6: Line-to-Surface Contact* discusses CONTA177 specifically, including the use of real constants and KEYOPTs.

The following table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

CONTA177 Input Summary

Nodes

I, J, (K)

Degrees of Freedom

UX, UY, UZ

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, (Blank), (Blank), (Blank), (Blank), (Blank),
(Blank), (Blank), FACT, DC, SLTO, TNOP,
TOLS, (Blank)
See *Table 1, "CONTA177 Real Constants"* for descriptions of the real constants.

Material Properties

DAMP, MU (**MP** command)
FRIC (**TB** command; see *Section 2.5.17: Contact Friction*)
CZM (**TB** command; see *Section 11.1.1: Cohesive Zone Materials Used for Debonding* in the *Contact Technology Guide*)

Special Features

Nonlinear
Large deflection
Isotropic or orthotropic friction
Debonding
Birth and death

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom. Currently, the default (UX, UY, UZ) is the only valid option:

0 --
UX, UY, UZ

KEYOPT(2)

Contact algorithm:

0 --

Augmented Lagrangian (default)

1 --

Penalty function

2 --

Multipoint constraint (MPC); see *Chapter 9: Multipoint Constraints and Assemblies* in the *Contact Technology Guide* for more information

3 --

Lagrange multiplier on contact normal and penalty on tangent

4 --

Pure Lagrange multiplier on contact normal and tangent

KEYOPT(4)

Type of surface-based constraint (see Surface-based Constraints for more information):

0 --

Rigid surface constraint

1 --

Force-distributed constraint

KEYOPT(5)

CNOF/ICONT Automated adjustment:

0 --

No automated adjustment

1 --

Close gap with auto CNOF

2 --

Reduce penetration with auto CNOF

3 --

Close gap/reduce penetration with auto CNOF

4 --

Auto ICONT

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --

Use default range for stiffness updating

1 --

Make a nominal refinement to the allowable stiffness range

2 --

Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control:

0 --

No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

Note

For KEYOPT(7) = 2 or 3, includes automatic bisection of increment. It is activated only if **SOLCONTROL**,ON,ON is issued at the procedure level.

KEYOPT(9)
Effect of initial penetration or gap:

0 --
Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Note

For KEYOPT(9) = 1, 3, or 4, the indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5.

KEYOPT(10)
Contact Stiffness Update:

0 --
Each load step if FKN is redefined during load step (pair based).

1 --
Each substep based on mean stress of underlying elements from the previous substep (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

3 --
Each load step if FKN is redefined during load step (individual element based).

4 --
Each substep based on mean stress of underlying elements from the previous substep (individual element based).

5 --
Each iteration based on current mean stress of underlying elements (individual element based).

Note

KEYOPT(10) = 0, 1, and 2 are pair based, meaning that the stiffness and settings for ICONT, FTOLN, PINB, PMAX, and PMIN are averaged across all the contact elements in a contact pair. For KEYOPT(10) = 3, 4, and 5, the stiffness and settings are based on each individual contact element (geometry and material behaviors).

KEYOPT(11)

Shell thickness effect (target side only):

0 --
Exclude

1 --
Include

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See *Section 9.3.4: Specifying a Local Coordinate System* in the *Contact Technology Guide* for more information.

Table 1 CONTA177 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN[*1*]	Normal penalty stiffness factor	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration

No.	Name	Description	For more information, see this section in the *Contact Technology Guide . . .*
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress	Choosing a Friction Model
10	CNOF	Contact surface offset	Accounting for Thickness Effect (CNOF and KEYOPT(11))
11	FKOP	Contact opening stiffness or contact damping	Selecting Surface Interaction Models
12	FKT[*1*]	Tangent penalty stiffness factor	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact force	Chattering Control Parameters
25	TOLS	Target edge extension factor	Real Constant TOLS

1. The units of real constants FKN and FKT have a factor of AREA with respect to those used in the surface-to-surface contact elements. See *Section 6.2.1.1: Real Constants FKN and FKT* for more information.

CONTA177 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "CONTA177 Element Output Definitions"*.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTA177 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K	Y	Y
XC, YC, ZC	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	Length	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying beam or shell element number	Y	-
CONT:STAT	Current contact statuses	*1*	*1*
OLDST	Old contact statuses	*1*	*1*
ISEG	Underlying current target number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
CONT:PRES	Normal contact force	*2*	*2*
TAUR/TAUS[*7*]	Tangential contact stresses	*2*	*2*
KN	Current normal contact stiffness (units: Force/Length)	*5*	*5*
KT	Current tangent contact stiffness (same units as KN)	*5*	*5*
MU[*8*]	Friction coefficient	Y	-
TASS/TASR[*7*]	Total (algebraic sum) sliding in S and R directions	*3*	*3*
AASS/AASR[*7*]	Total (absolute sum) sliding in S and R directions	*3*	*3*
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress SQRT (TAUR**2+TAUS**2)	*2*	*2*
CONT:STOTAL	Total stress SQRT (PRES**2+TAUR**2+TAUS**2)	*2*	*2*
CONT:SLIDE	Total sliding SQRT (TASS**2+TASR**2)	Y	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CNFX[*4*]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
CAREA	Contacting area	-	Y
FDDIS	Frictional energy dissipation	*6*	*6*
CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact force	*2*	*2*
SLTO	Allowable elastic slip	Y	Y
ELSI	Elastic slip distance for sticking contact within a substep	-	Y
DTSTART	Load step time during debonding	Y	Y

Name	Definition	O	R
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. The unit of the quantities is FORCE.
3. Only accumulates the sliding when contact occurs.
4. Contact element forces are defined in the global Cartesian system
5. The unit of stiffness is FORCE/LENGTH.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
7. For the case of orthotropic friction in contact between beams (or shell edges) and a 3-D surface, components are defined in the global Cartesian system.
8. For orthotropic friction, an equivalent coefficient of friction is output.

The following table lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information.

Name
output quantity as defined in *Table 2, "CONTA177 Element Output Definitions"*

Item
predetermined item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J, K
sequence number for data at nodes I, J, K

Table 3 CONTA177 (3-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
PRES	SMISC	13	1	2	3
TAUR	SMISC	-	5	6	7
TAUS	SMISC	-	9	10	11
FDDIS	SMISC	-	18	19	20
STAT[*1*]	NMISC	41	1	2	3
OLDST	NMISC	-	5	6	7
PENE[*2*]	NMISC	-	9	10	11

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
DBA	NMISC	-	13	14	15
TASR	NMISC	-	17	18	19
TASS	NMISC	-	21	22	23
KN	NMISC	-	25	26	27
KT	NMISC	-	29	30	31
TOLN	NMISC	-	33	34	35
IGAP	NMISC	-	37	38	39
PINB	NMISC	42	-	-	-
CNFX	NMISC	43	-	-	-
CNFY	NMISC	44	-	-	-
CNFZ	NMISC	45	-	-	-
ISEG	NMISC	-	46	47	48
AASR	NMISC	-	50	51	52
AASS	NMISC	-	54	55	56
CAREA	NMISC	-	58	59	60
MU	NMISC	-	62	63	64
DTSTART	NMISC	-	66	67	68
DPARAM	NMISC	-	70	71	72
CNOS	NMISC	-	112	113	114
TNOP	NMISC	-	116	117	118
SLTO	NMISC	-	120	121	122
ELSI	NMISC	-	136	137	138
DENERI	NMISC	-	140	141	142
DENERII	NMISC	-	144	145	146

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value

CONTA177 Assumptions and Restrictions

- The thickness effects of underlying beam elements on the contact side can be taken into account by specifying the contact surface offset CNOF.
- The thickness effects of underlying shell elements on the target side can be taken into account by setting KEYOPT(11) = 1.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Augmented Lagrangian method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses.

- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- This element allows birth and death and will follow the birth and death status of the underlying beam, pipe, shell, or target elements.

CONTA177 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property (input via **MP**,MU or **TB**,FRIC) is not allowed.
- The birth and death special feature is not allowed.
- The DAMP material property is not allowed.

CONTA178

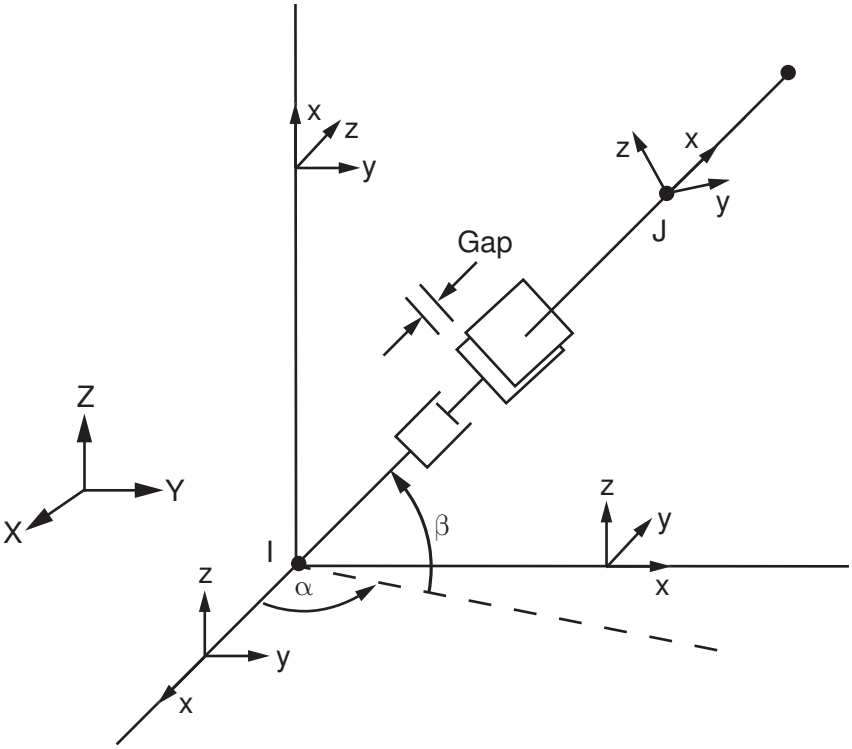
3-D Node-to-Node Contact

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

CONTA178 Element Description

CONTA178 represents contact and sliding between any two nodes of any types of elements. The element has two nodes with three degrees of freedom at each node with translations in the X, Y, and Z directions. It can also be used in 2-D and axisymmetric models by constraining the UZ degree of freedom. The element is capable of supporting compression in the contact normal direction and Coulomb friction in the tangential direction. The element may be initially preloaded in the normal direction or it may be given a gap specification. A longitudinal damper option can also be included. See CONTA178 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. Other contact elements, such as CONTAC12, COMBIN40, CONTAC52, are also available.

Figure 1 CONTA178 Geometry



CONTA178 Input Data

The geometry, node locations, and the coordinate system for this element are shown in the CONTA178 figure above. The element is defined by two nodes, an initial gap or interference (GAP), an initial element status (START), and damping coefficients CV1 and CV2. The orientation of the interface is defined by the node locations (I and J) or by a user specified contact normal direction. The interface is assumed to be perpendicular to the I-J line or to the specified gap direction. The element coordinate system has its origin at node I and the x-axis is directed toward node J or in the user specified gap direction. The interface is parallel to the element y-z plane. See *Section 8.2.2: Generating Contact Elements* in the *Contact Technology Guide* for more information on generating elements automatically using the EINTF command.

Contact Algorithms

Four different contact algorithms can be selected:

- Pure Lagrange multiplier method (KEYOPT(2) = 4)
- Lagrange multiplier on contact normal and penalty on frictional (tangential) direction (KEYOPT(2) = 3)
- Augmented Lagrange method (KEYOPT(2) = 0)
- Pure Penalty method (KEYOPT(2) = 1)

The following sections outline these four algorithms.

Pure Lagrange Multiplier

The pure Lagrange multiplier method does not require contact stiffness FKN, FKS. Instead it requires chattering control parameters TOLN, FTOL, by which ANSYS assumes that the contact status remains unchanged. TOLN is the maximum allowable penetration and FTOL is the maximum allowable tensile contact force.

Note

A negative contact force occurs when the contact status is closed. A tensile contact force (positive) refers to a separation between the contact surfaces, but not necessarily and open contact status.

The behavior can be described as follows:

- If the contact status from the previous iteration is open and the current calculated penetration is smaller than TOLN, then contact remains open. Otherwise the contact status switches to closed and another iteration is processed.
- If the contact status from the previous iteration is closed and the current calculated contact force is positive, but smaller than FTOL, then contact remains closed. If the tensile contact force is larger than FTOL, then the contact status changes from closed to open and ANSYS continues to the next iteration.

ANSYS will provide reasonable defaults for TOLN and FTOL. Keep in mind the following when providing values for TOLN and FTOL:

- A positive value is a scaling factor applied to the default values.
- A negative value is used as an absolute value (which overrides the default).

The objective of TOLN and FTOL is to provide stability to models which exhibit contact chattering due to changing contact status. If the values you use for these tolerances are too small, the solution will require more iterations. However, if the values are too big it will affect the accuracy of the solution, since a certain amount of penetration or tensile contact force are allowed.

Theoretically, the pure Lagrange multiplier method enforces zero penetration when contact is closed and "zero slip" when sticking contact occurs. However the pure Lagrange multiplier method adds additional degrees of freedom to the model and requires additional iterations to stabilize contact conditions. This will increase the computational cost and may even lead to solution divergence if many contact points are oscillating between sticking and sliding conditions during iterations.

Lagrange Multiplier on Normal and Penalty on Tangent Plane

An alternative algorithm is the Lagrange multiplier method applied on the contact normal and the penalty method (tangential contact stiffness) on the frictional plane. This method only allows a very small amount of slip for a sticking contact condition. It requires chattering control parameters TOLN, FTOL as well as the maximum

allowable elastic slip parameter SLTOL. Again, ANSYS provides default tolerance values which work well in most cases. You can override the default value for SLTOL by defining a scaling factor (positive value) or an absolute value (negative value). Based on the tolerance, current normal contact force, and friction coefficient, the tangential contact stiffness FKS can be obtained automatically. In a few cases, you can override FKS by defining a scaling factor (positive input) or absolute value (negative input). Use care when specifying values for SLTOL and FKS. If the value for SLTOL is too large and the value for FKS too small, too much elastic slip can occur. If the value for SLTOL is too small or the value for FKS too large, the problem may not converge.

Augmented Lagrange Method

The third contact algorithm is the augmented Lagrange method, which is basically the penalty method with additional penetration control. This method requires contact normal stiffness FKN, maximum allowable penetration TOLN, and maximum allowable slip SLTOL. FKS can be derived based on the maximum allowable slip SLTOL and the current normal contact force. ANSYS provides a default normal contact stiffness FKN which is based on the Young's modulus E and the size of the underlying elements. If Young's modulus E is not found, E = $1x10^9$ will be assumed.

You can override the default normal contact stiffness FKN by defining a scaling factor (positive input) or absolute value (negative input with unit force/length). If you specify a large value for TOLN, the augmented Lagrange method works as the penalty method. Use care when specifying values for FKN and TOLN. If the value for FKN is too small and the value for TOLN too large, too much penetration can occur. If the value for FKN is too large or the value for TOLN too small, the problem may not converge.

Penalty Method

The last algorithm is the pure penalty method. This method requires both contact normal and tangential stiffness values FKN, FKS. Real constants TOLN, FTOLN, and SLTOL are not used and penetration is no longer controlled in this method. Default FKN is provided as the one used in the augmented Lagrange method. The default FKS is given by MU x FKN. When FKN, FKS are defined as absolute values (negative input), the method works as the penalty method used in element CONTAC52.

Contact Normal Definition

The contact normal direction is of primary importance in a contact analysis. By default [KEYOPT(5) = 0 and NX, NY, NZ = 0], ANSYS will calculate the contact normal direction based on the initial positions of the I and J nodes, such that a positive displacement (in the element coordinate system) of node J relative to node I opens the gap. However, you must specify the contact normal direction for any of the following conditions:

- If nodes I and J have the same initial coordinates.
- If the model has an initial interference condition in which the underlying elements' geometry overlaps.
- If the initial open gap distance is very small.

In the above cases, the ordering of nodes I and J is critical. The correct contact normal usually points from node I toward node J unless contact is initially overlapped.

You can specify the contact normal by means of real constants NX, NY, NZ (direction cosines related to the global Cartesian system) or element KEYOPT(5). The following lists the various options for KEYOPT(5):

KEYOPT(5) = 0

The contact normal is either based on the real constant values of NX, NY, NZ or on node locations when NX, NY, NZ are not defined. For 2-D contact, NZ = 0.

KEYOPT(5) = 1 (2,3)

The contact normal points in a direction which averages the direction cosines of the X (Y, Z) axis of the nodal coordinates on both nodes I and J. The direction cosines on nodes I and J should be very close. This option may be supported by the **NORA** and **NORL** commands, which rotate the X axis of the nodal coordinate system to point to the surface normal of solid models.

KEYOPT(5) = 4 (5,6)

The contact normal points to X (Y, Z) of the element coordinate system issued by the **ESYS** command. If you use this option, make sure that the element coordinate system specified by **ESYS** is the Cartesian system. Otherwise, the global Cartesian system is assumed.

Contact Status

The initial gap defines the gap size (if positive) or the displacement interference (if negative). If KEYOPT(4) = 0, the default, the gap size can be automatically calculated from the GAP real constant and the node locations (projection of vector points from node I to J on the contact normal), that is, the gap size is determined from the additive effect of the geometric gap and the value of GAP.

If KEYOPT(4) = 1, the initial gap size is only based on real constant GAP (node locations are ignored).

By default KEYOPT(9) is set to 0, which means the initial gap size is applied in the first load step. To ramp the initial gap size with the first load step (to model initial interference problems, for example), set KEYOPT(9) = 1. Also, set **KBC**,0 and do not specify any external loads over the first load step.

The force deflection relationships for the contact element can be separated into the normal and tangential (sliding) directions. In the normal direction, when the normal force (FN) is negative, the contact status remains closed (STAT = 3 or 2). In the tangential direction, for FN < 0 and the absolute value of the tangential force (FS) less than μ|FN|, contact "sticks" (STAT = 3). For FN < 0 and FS = μ|FN|, sliding occurs (STAT = 2). As FN becomes positive, contact is broken (STAT = 1) and no force is transmitted (FN = 0, FS = 0).

The contact condition at the beginning of the first substep can be determined from the START parameter. The initial element status (START) is used to define the "previous" condition of the interface at the start of the first substep. This value overrides the condition implied by the interference specification and can be useful in anticipating the final interface configuration and reducing the number of iterations required for convergence. However, specifying unrealistic START values can sometimes degrade the convergence behavior.

If START = 0.0 or blank, the initial status of the element is determined from either the GAP value or the KEYOPT(4) setting. If START = 3.0, contact is initially closed and not sliding (μ ≠ 0), or sliding (if μ = 0.0). If START = 2.0, contact is initially closed and sliding. If START = 1.0, contact is initially open.

Friction

The only material property used is the interface coefficient of friction μ (MU). A zero value should be used for frictionless surfaces. Temperatures may be specified at the element nodes (for material property evaluation only). The coefficient of friction μ is evaluated at the average of the two node temperatures. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I).

For analyses involving friction, using **NROPT**,UNSYM is useful (and, in fact, sometimes required if the coefficient of friction μ is > 0.2) for problems where the normal and tangential (sliding) motions are strongly coupled.

Weak Spring

KEYOPT(3) can be used to specify a "weak spring" across an open or free sliding interface, which is useful for preventing rigid body motion that could occur in a static analysis. The weak spring stiffness is computed by multiplying the normal stiffness KN by a reduction factor if the real constant REDFACT is positive (which defaults

to 1 x 10^{-6}). The weak spring stiffness can be overridden if REDFACT has a negative value. Set KEYOPT(3) = 1 to add weak spring stiffness only to the contact normal direction when contact is open. Set KEYOPT(3) = 2 to add weak spring stiffness to the contact normal direction for open contact and tangent plane for frictionless or open contact.

Just as for CONTAC52, the weak spring only contributes to global stiffness, which prevents a "singularity" condition from occurring during the solution phase if KEYOPT(3) = 1,2. By setting KEYOPT(3) = 3,4, the weak spring will contribute both to the global stiffness and the internal nodal force which holds two separated nodes.

Note

The weak spring option should *never* be used in conjunction with either the no-separation or bonded contact options defined by KEYOPT(10).

Contact Behavior

Use KEYOPT(10) to model the following different contact surface behaviors:

KEYOPT(10) = 0
: Models standard unilateral contact; that is, normal pressure equals zero if separation occurs.

KEYOPT(10) = 1
: Models rough frictional contact where there is no sliding. This case corresponds to an infinite friction coefficient and ignores the material property input MU.

KEYOPT(10) = 2
: Models no separation contact, in which two gap nodes are tied (although sliding is permitted) for the remainder of the analysis once contact is established.

KEYOPT(10) = 3
: Models bonded contact, in which two gap nodes are bonded in all directions (once contact is established) for the remainder of the analysis.

KEYOPT(10) = 4
: Models no separation contact, in which two gap nodes are always tied (sliding is permitted) throughout the analysis.

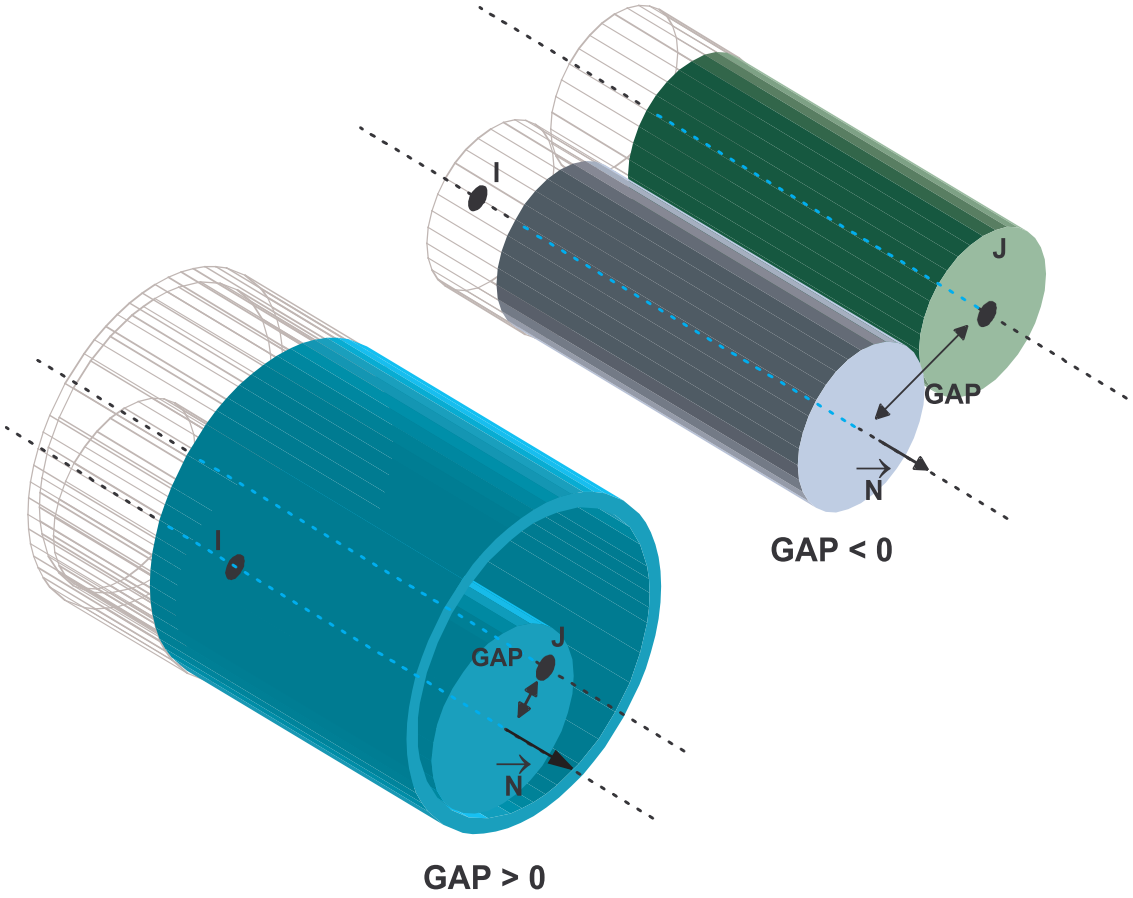
KEYOPT(10) = 5
: Models bonded contact, in which two gap nodes are bonded in all directions throughout the analysis.

KEYOPT(10) = 6
: Models bonded contact, in which two gap nodes that are initially in a closed state will remain closed and two gap nodes that are initially in an open state will remain open throughout the analysis.

Cylindrical Gap

The cylindrical gap option (KEYOPT(1) = 1) is useful where the final contact normal is not fixed during the analysis, such as in the interaction between concentric pipes. With this option, you define the real constants NX, NY, NZ as the direction cosines of the cylindrical axis ($\vec{N}$) in the global Cartesian coordinate system. The contact normal direction lies in a cross section that is perpendicular to the cylindrical axis. The program measures the relative distance |XJ - XI| between the current position of node I and the current position of node J projected onto the cross section. NX, NY, NZ defaults to (0,0,1), which is the case for a 2-D circular gap. With the cylindrical gap option, KEYOPT(4) and KEYOPT(5) are ignored and node ordering can be arbitrary. Real constant GAP is no long referred as the initial gap size and a zero value is not allowed. The following explanation defines the model based on the sign of the GAP value.

Figure 2 CONTA178 Gap and Nodes



- A positive GAP value models contact when one smaller cylinder inserted into another parallel larger cylinder. GAP is equal to the difference between the radii of the cylinders (|RJ - RI|) and it represents the maximum allowable distance projected on the cross-section. The contact constraint condition can be written as : $|XJ - XI| \le |GAP|$
- A negative GAP value models external contact between two parallel cylinders. GAP is equal to the sum of the radii of the cylinders (|RJ + RI|) and it represents the minimum allowable distance projected on the cross-section. The contact constraint condition can be written as: $|XJ - XI| \ge |GAP|$

Damper

The damping capability is only used for modal and transient analyses. By default, the damping capability is removed from the element. Damping is only active in the contact normal direction when contact is closed. The damping coefficient units are Force (Time/Length). For a 2-D axisymmetric analysis, the coefficient should be on a full 360° basis. The damping force is computed as $F_x = -Cv\frac{du_x}{dt}$, where Cv is the damping coefficient given by $C_v = C_{v1} + C_{v2}xV$. V is the velocity calculated in the previous substep. The second damping coefficient (C_{v2}) is available to produce a nonlinear damping effect.

Monitoring Contact Status

By default, ANSYS will not print out contact status and contact stiffness for each individual element. Use KEYOPT(12) = 1 to print out such information, which may help in solving problems that are difficult to converge.

A summary of the element input is given in *CONTA178 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

CONTA178 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

FKN, GAP, START, FKS, REDFACT, NX,
NY, NZ, TOLN, FTOL, SLTOL, CV1,
CV2
See *Table 1, "CONTA178 Real Constants"* for a description of the real constants

Material Properties
DAMP, MU

Surface Loads
None

Body Loads
Temperatures - T(I), T(J)

Special Features
Nonlinear Gap type

KEYOPT(1)
Gap type:

0 --
Unidirectional gap

1 --
Cylindrical gap

KEYOPT(2)
Contact algorithm:

0 --
Augmented Lagrange method (default)

1 --
Pure Penalty method

3 --
Lagrange multiplier on contact normal and penalty on tangent (uses U/P formulation for normal contact, non-U/P formulation for tangential contact)

4 --
Lagrange multiplier method

KEYOPT(3)
Weak Spring:

0 --
Not used

1 --
Acts across an open contact (only contributes to stiffness)

2 --
Acts across an open contact or free sliding plane (only contributes to stiffness)

3 --
Acts across an open contact (contributes to stiffness and internal force)

4 --
Acts across an open contact or free sliding plane (contributes to stiffness and force)

KEYOPT(4)
Gap size:

0 --
Gap size based on real constant GAP + initial node locations

1 --
Gap size based on real constant GAP (ignore node locations)

KEYOPT(5)
Basis for contact normal:

0 --
Node locations or real constants NX, NY, NZ

1 --
X - component of nodal coordinate system (averaging on two contact nodes)

2 --
Y - component of nodal coordinate system (averaging on two contact nodes)

3 --
Z - component of nodal coordinate system (averaging on two contact nodes)

4 --
X - component of defined element coordinate system (ESYS)

5 --
Y - component of defined element coordinate system (ESYS)

6 --
Z - component of defined element coordinate system (ESYS)

KEYOPT(7)
Element level time incrementation control:

0 --
No control

1 --
Change in contact predictions are made to maintain a reasonable time/load increment. It is activated only if **SOLCONTROL**,ON,ON at the procedure level

2 --
Change in contact predictions are made to achieve the minimum time/load increment whenever a change in contact status occurs. Includes automatic bisection of increment. It is activated only if **SOLCONTROL**,ON,ON at the procedure level

KEYOPT(9)
Initial gap step size application:

0 --
Initial gap size is step applied

1 --
Initial gap size is ramped in the first load step

KEYOPT(10)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial)

KEYOPT(12)
Contact Status:

0 --
Does not print contact status

1 --
Monitor and print contact status, contact stiffness

Table 1 CONTA178 Real Constants

No.	Name	Description
1	FKN	Normal stiffness
2	GAP	Initial gap size
3	START	Initial contact status
4	FKS	Sticking stiffness
5	REDFACT	KN/KS reduction factor
6	NX	Defined gap normal - X component
7	NY	Defined gap normal - Y component
8	NZ	Defined gap normal - Z component
9	TOLN	Penetration tolerance
10	FTOL	Maximum tensile contact force
11	SLTOL	Maximum elastic slip
12	CV1	Damping coefficient
13	CV2	Nonlinear damping coefficient

CONTA178 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in Element Output Definitions.

The value of USEP is determined from the normal displacement (UN), in the element x-direction, between the contact nodes at the end of a substep. This value is used in determining the normal force, FN. The values represented by UT(Y, Z) are the total translational displacements in the element y and z directions. The maximum value printed for the sliding force, FS, is μ|FN|. Sliding may occur in both the element y and z directions. STAT describes the status of the element at the end of a substep.

- If STAT = 3, contact is closed and no sliding occurs
- If STAT = 1, contact is open
- If STAT = 2, node J slides relative to node I

For a frictionless surface (μ = 0.0), the converged element status is either STAT = 2 or 1.

The element coordinate system orientation angles α and β (shown in *Figure 1,"CONTA178 Geometry"*) are computed by the program from the node locations. These values are printed as ALPHA and BETA respectively. α ranges from 0° to 360° and β from -90° to +90°. Elements lying along the Z-axis are assigned values of α = 0°, β = ± 90°, respectively. Elements lying off the Z-axis have their coordinate system oriented as shown for the general α, β position.

Note

For α = 90°, $\beta \rightarrow$ 90°, the element coordinate system flips 90° about the Z-axis. The value of ANGLE represents the principal angle of the friction force in the element y-z plane. A general description of solution output is given in *Section 2.2.2: Element Solution*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 CONTA178 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	*3*
TEMP	T(I), T(J)	Y	Y
USEP	Gap size	Y	Y
FN	Normal force (along I-J line)	Y	Y
STAT	Element status	*1*	*1*

Name	Definition	O	R
OLDST	Old contact status	*1*	*1*
ALPHA, BETA	Element orientation angles	Y	Y
MU	Coefficient of friction	*2*	*2*
UT(Y, Z)	Displacement (node J - node I) in element y and z directions	*2*	*2*
FS(Y, Z)	Tangential (friction) force in element y and z directions	*2*	*2*
ANGLE	Principal angle of friction force in element y-z plane	*2*	*2*

1. If the value of STAT is:

   1 - Open contact

   2 - Sliding contact

   3 - Sticking contact (no sliding)
2. If MU>0.0
3. Available only at centroid as a ***GET** item.

Table 3, "CONTA178 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* for more information. The following notation is used in *Table 3, "CONTA178 Item and Sequence Numbers"* :

Name
output quantity as defined in the Element Output Definition

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 CONTA178 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FN	SMISC	1
FSY	SMISC	2
FSZ	SMISC	3
STAT	NMISC	1
OLDST	NMISC	2
USEP	NMISC	3
ALPHA	NMISC	4
BETA	NMISC	5
UTY	NMISC	6
UTZ	NMISC	7
MU	NMISC	8
ANGLE	NMISC	9
KN	NMISC	10
KS	NMISC	11

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TOLN	NMISC	12
FTOL	NMISC	13
SLTOL	NMISC	14

CONTA178 Assumptions and Restrictions

- The element operates bilinearly only in the static and the nonlinear transient dynamic analyses. If used in other analysis types, the element maintains its initial status throughout the analysis.
- The element is nonlinear and requires an iterative solution.
- Nonconverged substeps are not in equilibrium.
- Unless the contact normal direction is specified by (NX, NY, NZ) or KEYOPT(5), nodes I and J must not be coincident or overlapped since the nodal locations define the interface orientation. In this case the node ordering is not an issue. On the other hand, if the contact normal is not defined by nodal locations, the node ordering is critical. Use **/PSYMB**, **ESYS** to verify the contact normal and use **EINTF**,,,REVE to reverse the normal if wrong ordering is detected. To determine which side of the interface contains the nodes, use **ESEL**,,ENAM,,178 and then **NSLE**,,POS,1.
- The element maintains its original orientation in either a small or a large deflection analysis unless the cylindrical gap option is used.
- For real constants FKN, REDFACT, TOLN, FTOL, SLTOL and FKS, you can specify either a positive or negative value. ANSYS interprets a positive value as a scaling factor and interprets a negative value as the absolute value. These real constants can be changed between load steps or during restart stages.
- The Lagrange multiplier methods introduce zero diagonal terms in the stiffness matrix. The PCG solver may encounter precondition matrix singularity. The Lagrange multiplier methods often overconstrain the model if boundary conditions, coupling, and constraint equations applied on the contact nodes overlay the contact constraints. Chattering is most likely to occur due to change of contact status, typically for contact impact problems. The Lagrange multipliers also introduce more degrees of freedom which may result in spurious modes for modal and linear eigenvalue bucking analysis. Therefore, the augmented Lagrange method option is the best choice for: PCG iterative solver, transient analysis for impact problems, modal, and eigenvalue bucking analysis.
- The element may not be deactivated with the **EKILL** command.

CONTA178 Product Restrictions

There are no product-specific restrictions for this element.

PRETS179

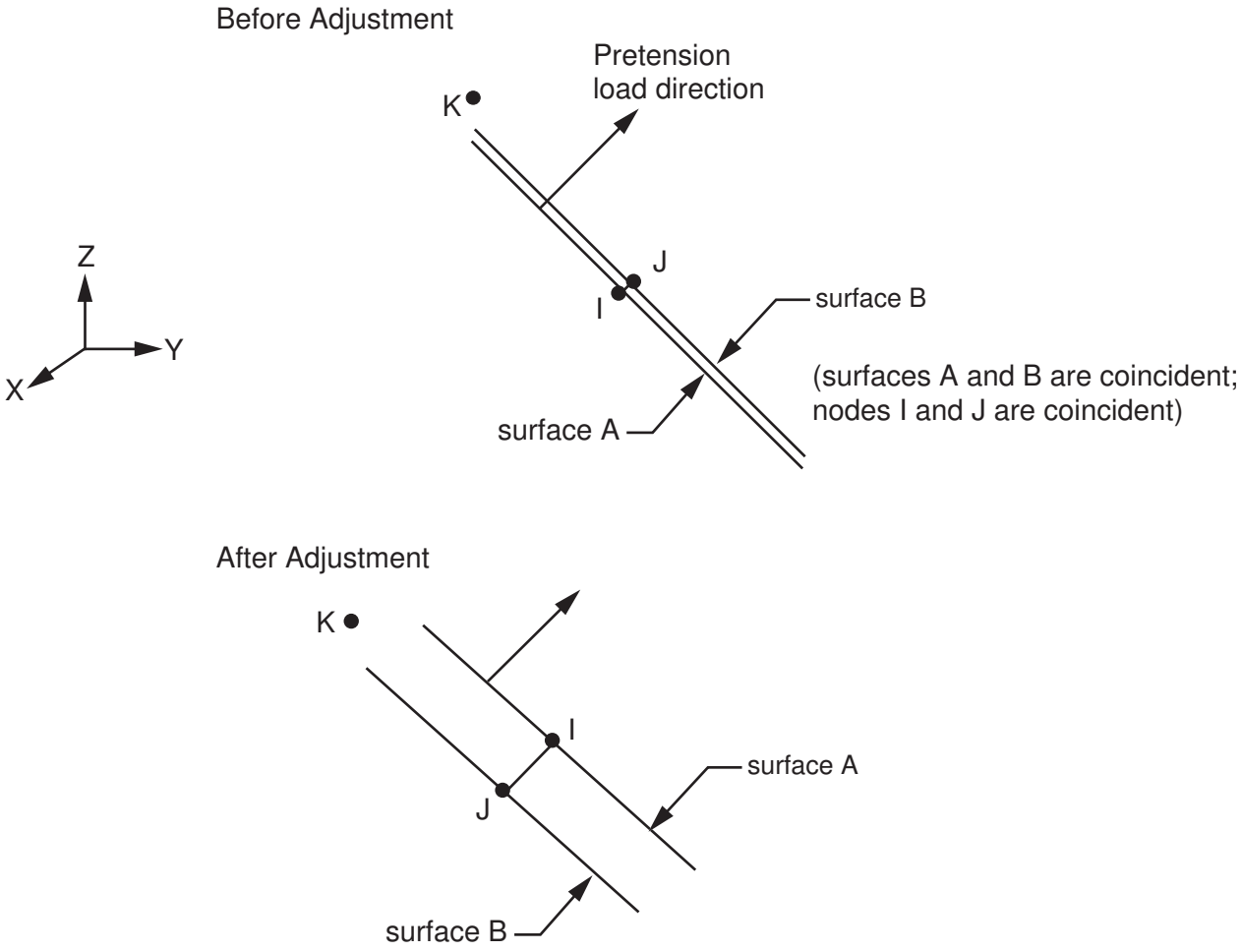
Pretension

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

PRETS179 Element Description

PRETS179 is used to define a 2-D or 3-D pretension section within a meshed structure. The structure can be built from any 2-D or 3-D structural elements (solid, beam, shell, pipe, or link). The PRETS179 element has one translation degree of freedom, UX. (UX represents the defined pretension direction. ANSYS transforms the geometry of the problem so that, internally, the pretension force is applied in the specified pretension load direction, regardless of how the model is defined.) Loads can be applied using the **SLOAD** command. These loads will overwrite any F or D command specifications on the same nodes at solution time. Only tension loads can be applied; bending or torsion loads are ignored. See PRETS179 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about the element. See Defining Pretension in a Joint Fastener in the *Basic Analysis Guide* for a discussion of how to generate PRETS179 elements automatically using the **PSMESH** command. Keep in mind when creating the PRETS179 elements that the pretension load direction is specified relative to surface A. (For backward compatibility, it is also possible to generate such elements using the **EINTF** command.)

Figure 1 PRETS179 Geometry



PRETS179 Input Data

The pretension section is modeled by a set of pretension elements. The geometry, node locations, and the coordinate system for the pretension element are shown in *Figure 1, "PRETS179 Geometry"*. The pretension element is defined by three nodes I, J, K and the section data NX, NY, NZ which define the pretension load direction relative to surface A. The pretension load direction is constant and is not updated for large displacements. Although it is not recommended, the pretension load direction can be changed between load steps by changing the section data. For large-deflection problems, you could track the deflection and change the pretension load direction accordingly.

Nodes I and J are initially coincident and they should be defined in the same nodal coordinate system. No boundary conditions apply on node J. For each pretension section, the node ordering of the pretension elements is critical. The I and J nodes must be ordered so that all nodes I are on surface A and all nodes J are on surface B.

Node K is the pretension node. This pretension node provides a convenient way to assign boundary conditions on an entire pretension section. Node K can be anywhere in space; however, its nodal coordinate system must be global Cartesian. Each pretension section has only one pretension node associated with it. Node K should only connect to pretension elements that use the same section number.

The pretension node K has only one translation degree of freedom UX, which defines the relative displacement between the two sections A and B in the pretension load direction. Sliding motion is prevented automatically. If the pretension node and the bolted structure are not well constrained, rigid body motion can occur. Therefore, in the beginning of each load step, you should verify the boundary conditions for bolt structures carefully.

The following table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

PRETS179 Input Summary

Nodes
I, J, K

Degrees of Freedom
UX (tightening adjustment of the pretension section)

Real Constants
None

Material Properties
DAMP

Surface Loads
None

Body Loads
None

Special Features
Nonlinear

KEYOPTs
None

PRETS179 Output Data

Nodal displacements are included in the overall displacement solution. There is no printed or post element data output for the pretension element. ANSYS automatically determines the deformations of the bolt structure. The underlying elements connected to both sides of cutting surfaces appear overlap under the pretension load. The

displacement of the pretension node gives the adjustment of the pretension. Use **PRNSOL** to list the adjustment. The reaction force on the pretension node provides the total normal force across the pretension section. Use **PRRSOL** or **PRRFOR** command to list the tension force. The stress distribution of underlying elements provides a good estimation of the stress across the pretension section.

PRETS179 Assumptions and Restrictions

- The nodal coordinate system of the pretension node K must be global Cartesian.
- You cannot apply any constraint equations (or coupling) on any pretension element nodes.
- The **NROTAT** command can not be applied on pretension node K. **NROTAT** can be applied to the other nodes I and J in such way that they are rotated into the same nodal coordinate system. If K has been mistakenly rotated into another coordinate system, ANSYS will issue a warning and will automatically rotate it back into the global Cartesian system. Similarly, if I and J are rotated into different coordinate systems, ANSYS will issue a warning and will automatically rotate J to be consistent with I.
- The pretension normal NX, NY, NZ must be specified through section data. You should not change section data either between load steps or during restart stages; otherwise ANSYS assumes the pretension normal varies between the load steps.
- The structure can be composed of superelements. However, all the pretension nodes must remain as the master nodes.
- The element may not be deactivated with the **EKILL** command.
- Use of this element is limited to structural analyses only.
- This element is not supported in cyclic symmetry analyses.

PRETS179 Product Restrictions

There are no product-specific restrictions for this element.

LINK180

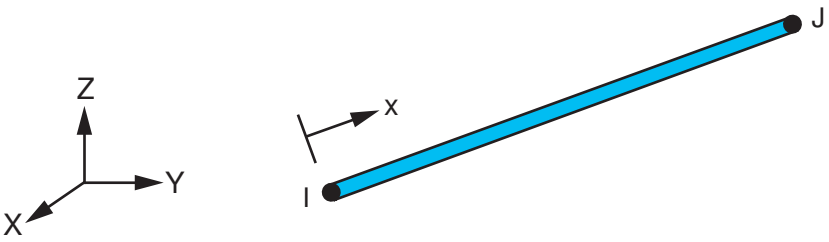
3-D Finite Strain Spar (or Truss)

MP ME ST PR PRN <> <> <> <> <> <> PP VT
Product Restrictions

LINK180 Element Description

LINK180 is a spar that can be used in a variety of engineering applications. This element can be used to model trusses, sagging cables, links, springs, etc. This 3-D spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, rotation, large deflection, and large strain capabilities are included. By default, LINK180 includes stress stiffness terms in any analysis with **NLGEOM**,ON. Elasticity, isotropic hardening plasticity, kinematic hardening plasticity, Hill anisotropic plasticity, Chaboche nonlinear hardening plasticity, and creep are supported. See LINK180 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element. A tension-only compression-only element is defined as LINK10.

Figure 1 LINK180 Geometry



LINK180 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "LINK180 Geometry"*. The element is defined by two nodes, the cross-sectional area (AREA), added mass per unit length (ADDMAS), and the material properties. The element X-axis is oriented along the length of the element from node I toward node J.

Element loads are described in *Section 2.8: Node and Element Loads*. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature T(J) defaults to T(I).

LINK180 allows a change in cross-sectional area as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved, even after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you may choose to keep the cross section constant or rigid.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The *LINK180 Input Summary* table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

LINK180 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Real Constants

AREA - Cross-sectional area
ADDMAS - Added mass (mass/length)

Material Properties
EX, (PRXY or NUXY), ALPX (or CTEX *or* THSX), DENS, GXY, DAMP

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See *Section 2.5: Data Tables - Implicit Analysis* for details of the material models.

KEYOPT(2)
Cross-section scaling (applies only if **NLGEOM**,ON has been invoked):

0 --
Enforce incompressibility; cross section is scaled as a function of axial stretch (default).

1 --
Section is assumed to be rigid.

KEYOPT(10)
User defined initial stress:

0 --
No user subroutine to provide initial stresses (default).

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user-written subroutines

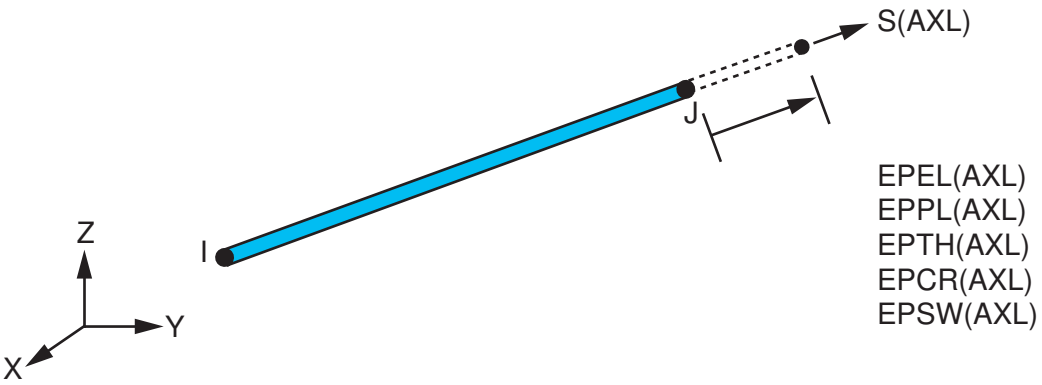
LINK180 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "LINK180 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "LINK180 Stress Output"*. A general description of solution output is given in *Section 2.2: Solution Output*. Element results can be viewed in POST1 with **PRESOL**,ELEM. See the *Basic Analysis Guide* for details.

Figure 2 LINK180 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 LINK180 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Center location	Y	*3*
AREA	Cross-sectional area	Y	Y
FORCE	Member force in the element coordinate system	Y	Y
STRESS	Axial stress	Y	Y
EPEL	Axial elastic strain	Y	Y
TEMP	Temperatures T(I), T(J)	Y	Y
EPTH	Axial thermal strain	Y	Y
EPPL	Axial plastic strain	*1*	*1*
PWRK	Plastic strain energy density	*1*	*1*

Name	Definition	O	R
EPCR	Axial creep strain	*2*	*2*
CWRK	Creep strain energy density	*2*	*2*

1. Nonlinear solution, only if element has a nonlinear material.
2. Nonlinear solution, only if creep is included.
3. Available only at centroid as a ***GET** item.

Table 2,"LINK180 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2,"LINK180 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1,"LINK180 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** and

ESOL

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 2 LINK180 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
STRESS	LS	1	-	-
EPEL	LEPEL	1	-	-
EPTH	LEPTH	1	-	-
EPPL	LEPPL	1	-	-
EPCR	LEPCR	1	-	-
FORCE	SMISC	1	-	-
TEMP	LBFE	-	1	2

LINK180 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends, and of uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The temperature is assumed to vary linearly along the length of the spar.
- The displacement shape function implies a uniform stress in the spar.

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

LINK180 Product Restrictions

There are no product-specific restrictions for this element.

SHELL181

4-Node Finite Strain Shell

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SHELL181 Element Description

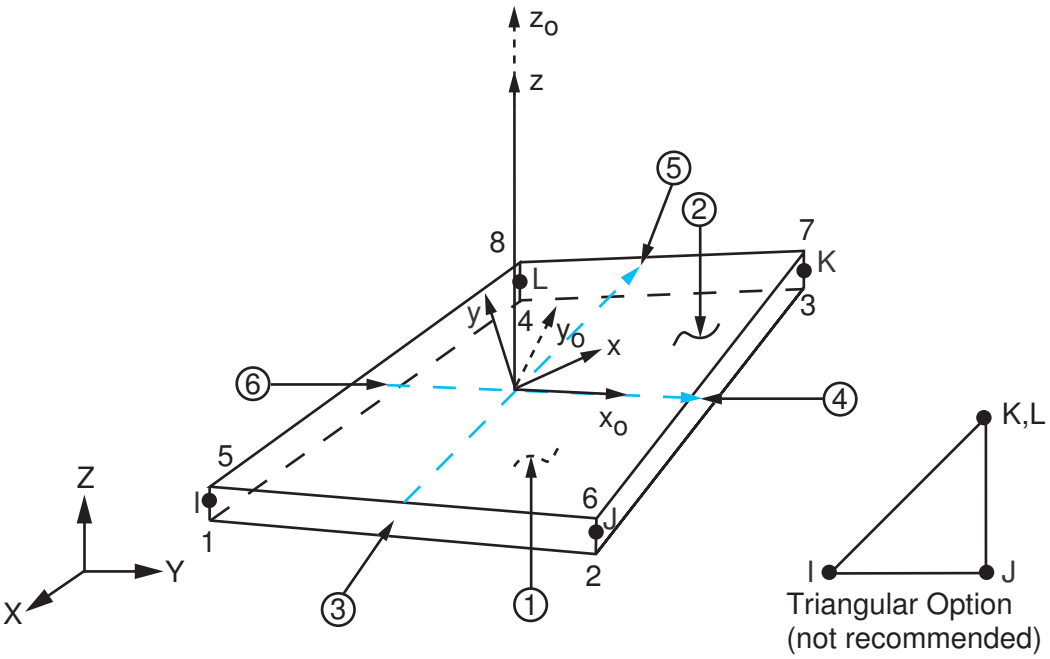
SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.

SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.

SHELL181 may be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory (usually referred to as Mindlin-Reissner shell theory).

SHELL181 can be used instead of SHELL43 for many problems that have convergence difficulty with SHELL43. See SHELL181 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL181 Geometry



x_o = Element x-axis if ESYS is not provided.

x = Element x-axis if ESYS is provided.

SHELL181 Input Data

The geometry, node locations, and the element coordinate system for this element are shown in *Figure 1, "SHELL181 Geometry"*. The element is defined by four nodes: I, J, K, and L. The element formulation is based on

logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small. To define the thickness and other information, you can use either real constants or section definition (and a section can be partially defined using data from a FiberSIM `.xml` file). The option of using real constants is available only for single-layer shells. If a SHELL181 element references both real constant set data and a valid shell section type, real constant data is ignored.

SHELL181 also accepts the preintegrated shell section type (**SECTYPE**,,GENS). When the element is associated with the GENS section type, thickness or material definitions are not required. For more information, see *Section 17.3: Using Preintegrated General Shell Sections*.

Thickness Definition Using Real Constants

The thickness of the shell may be defined at each of its nodes. The thickness is assumed to vary smoothly over the area of the element. If the element has a constant thickness, only TK(I) needs to be input. If the thickness is not constant, all four thicknesses must be input.

Layered Section Definition Using Section Commands

Alternatively the shell thickness and more general properties may be specified using section commands. SHELL181 may be associated with a shell section (see **SECTYPE** command description). Shell section is a more general method to define shell construction than the real constants option. Shell section commands allow for layered composite shell definition, and provide the input options for specifying the thickness, material, orientation and number of integration points through the thickness of the layers. Note that a single layer shell is not precluded using shell section definition, but provides more flexible options such as the use of the ANSYS function builder to define thickness as a function of global coordinates and the number of integration points used.

You may designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer when using section input. When only 1, the point is always located midway between the top and bottom surfaces. If 3 or more points, 2 points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the 2 points. An exception occurs when designating 5 points, where the quarter point locations are moved 5 percent toward their nearest layer surface to agree with the locations selected with real constant input. The default number of integration points for each layer is 3. However, when a single layer is defined and plasticity is present, the number of integration points will be changed to a minimum of 5 during solution. Note that when Real Constants are used, ANSYS uses 5 points of integration and Sections will produce a comparable solution.

Other Input

The default orientation for this element has the S_1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element, which connects the midsides of edges LI and JK. In the most general case, the axis can be defined as:

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where:

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{4}\right)\left[-\{x\}^I + \{x\}^J + \{x\}^K - \{x\}^L\right]$$

$\{x\}^I, \{x\}^J, \{x\}^K, \{x\}^L$ = global nodal coordinates

For undistorted elements, the default orientation is the same as described in *Section 2.3: Coordinate Systems* (the first surface direction is aligned with the IJ side). For spatially warped or otherwise distorted elements, the default orientation represents the stress state better because the element uses a single point of quadrature (by default) in the element domain.

The first surface direction S_1 can be rotated by angle THETA (in degrees) as a real constant for the element or for using the **SECDATA** command. For an element, you can specify a single value of orientation in the plane of the element. Layer-wise orientation is possible when section definition is used.

You can also define element orientation via the **ESYS** command. See *Section 2.3: Coordinate Systems*.

The element supports degeneration into a triangular form; however, use of the triangular form is *not* recommended, except when used as mesh filler elements or with the membrane option (KEYOPT(1) = 1). The triangle form is generally more robust when using the membrane option with large deflections.

To evaluate stresses and strains on exterior surfaces, use KEYOPT(1) = 2. When used as overlaid elements on the faces of 3-D elements, this option is similar to the surface stress option (described in the *Theory Reference for ANSYS and ANSYS Workbench*), but is more general and applicable to nonlinear analysis. The element used with this option does not provide any stiffness, mass, or load contributions. This option should only be used in single-layered shells. Irrespective of other settings, SHELL181 provides stress and strain output at the center of the layer.

SHELL181 uses a penalty method to relate the independent rotational degrees of freedom about the normal (to the shell surface) with the in-plane components of displacements. The ANSYS program chooses an appropriate penalty stiffness by default. However, you can change the default value if necessary by using the tenth real constant (drill stiffness factor; see *Table 1, "SHELL181 Real Constants"*). The value of this real constant is the scaling parameter for the default penalty stiffness. Using a higher value could contribute to a larger nonphysical energy content in the model. For this reason, use caution when changing the default. When using the Section definition with SHELL181, drill stiffness factor may be specified by using the **SECCONTROLS** command.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL181 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-1024 maximum). The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and if exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

Using KEYOPT(3), SHELL181 supports uniform reduced integration and full integration with incompatible modes. By default, this element uses the uniform reduced integration for performance reasons in nonlinear applications.

Using reduced integration with hourglass control creates some usage restrictions, although minimal. For example, to capture the in-plane bending of a cantilever or a stiffener (see *Figure 2, "SHELL181 Typical Bending Applications"*), a number of elements through the thickness direction is necessary. The performance gains achieved by using uniform reduced integration are significant enough to offset the need to use more elements. In relatively well-refined meshes, hourglassing issues are largely irrelevant.

When the reduced integration option is used, you can check the accuracy of the solution by comparing the total energy (SENE label in **ETABLE**) and the artificial energy (AENE label in **ETABLE**) introduced by hourglass control.

If the ratio of artificial energy to total energy is less than 5%, the solution is generally acceptable. The total energy and artificial energy can also be monitored by using **OUTPR**,VENG in the solution phase.

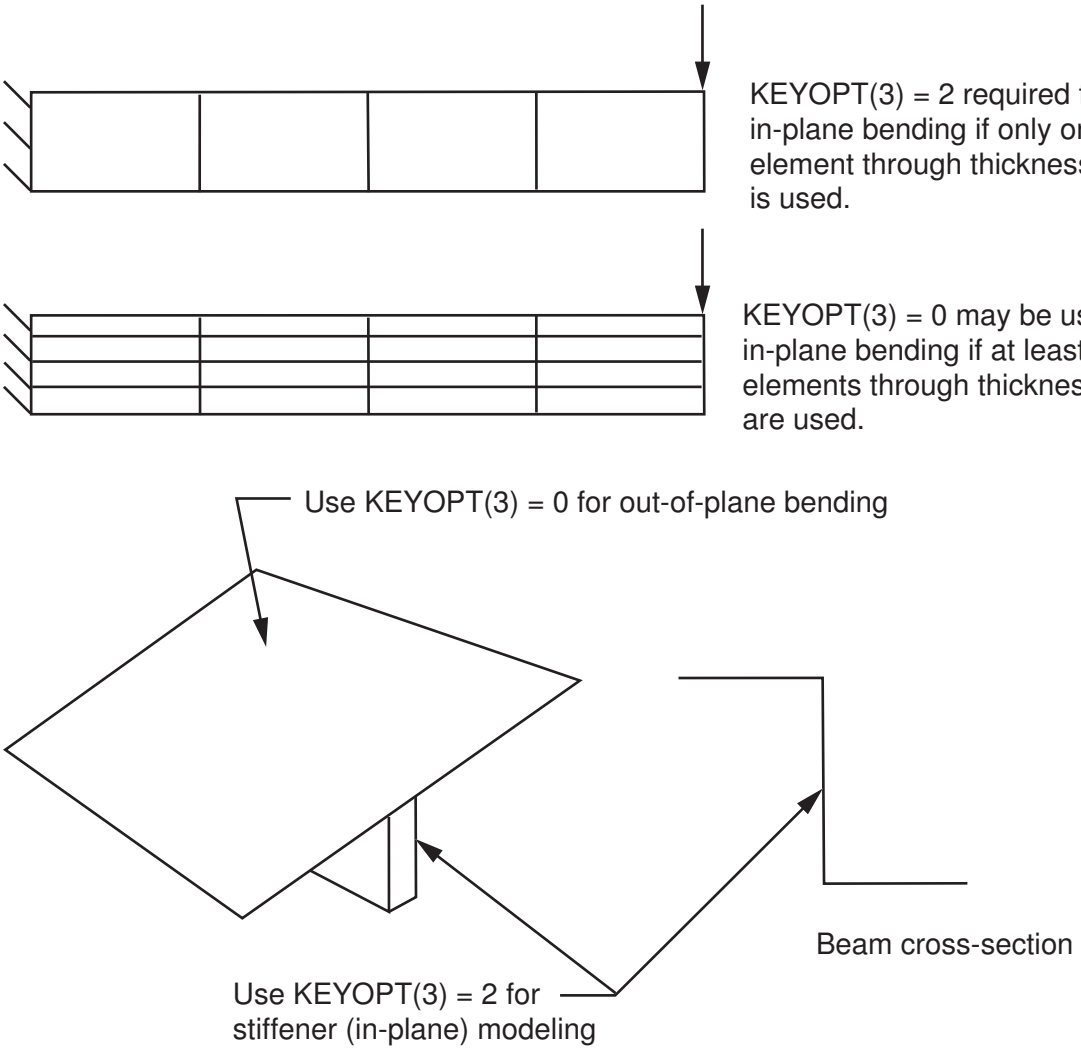
Bilinear elements, when fully integrated, are too stiff in in-plane bending. SHELL181 uses the method of incompatible modes to enhance the accuracy in bending-dominated problems. This approach is also called "extra shapes" or "bubble" modes approach. SHELL181 uses the formulation that ensures satisfaction of the patch test (J. C. Simo and F. Armero, "Geometrically nonlinear enhanced strain mixed methods and the method of incompatible modes," *IJNME*, Vol. 33, pp. 1413-1449, 1992).

When including incompatible modes in the analysis, you must use full integration. KEYOPT(3) = 2 implies the inclusion of incompatible modes and the use of full (2x2) quadrature.

SHELL181, with KEYOPT(3) = 2 specified, does not have any spurious energy mechanisms. This specific form of SHELL181 is highly accurate, even with coarse meshes. We recommend that you use KEYOPT(3) = 2 if you encounter any hourglass-related difficulties with the default options. KEYOPT(3) = 2 is also necessary if the mesh is coarse and in-plane bending of the elements dominate the response. We recommend this option with all layered applications.

KEYOPT(3) = 2 imposes the fewest usage restrictions. You can always choose this option. However, you can improve element performance by choosing the best option for your problem. Consider the problems illustrated in *Figure 2, "SHELL181 Typical Bending Applications"*

Figure 2 SHELL181 Typical Bending Applications



The cantilever beam and the beam cross-section to be modeled with shells are typical examples of in-plane bending-dominated problems. The use of KEYOPT(3) = 2 is the most effective choice in these circumstances. Reduced integration would require refined meshes. For example, reduced integration for the cantilever beam problem requires four elements through the thickness, whereas the full integration with incompatible modes only requires one element through the thickness.

For the stiffened shell, the most effective choice is to use KEYOPT(3) = 0 for the shell and KEYOPT(3) = 2 for the stiffener.

When KEYOPT(3) = 0 is specified, SHELL181 uses an hourglass control method for membrane and bending modes. By default, SHELL181 calculates the hourglass parameters for both metal and hyperelastic applications. You can override the default values by using real constants 11 and 12 (see *Table 1, "SHELL181 Real Constants"*). Instead of changing hourglass stiffness parameters, you should either increase the mesh density or choose a fully integrated option (KEYOPT(3) = 2). When Section definition is used, you may specify the hourglass stiffness scaling factors by using the **SECCONTROLS** command.

SHELL181 includes the linear effects of transverse shear deformation. An assumed shear strain formulation of Bathe-Dvorkin is used to alleviate shear locking. The transverse shear stiffness of the element is a 2x2 matrix as shown below:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ sym & E_{22} \end{bmatrix}$$

In the above matrix, E_{11}, E_{22}, and E_{12} are real constants 7, 8, and 9 (see *Table 1, "SHELL181 Real Constants"*). You can override the default transverse shear stiffness values by assigning different values to those real constants. This option is effective for analyzing sandwich shells. Alternatively the **SECCONTROLS** command provides for the definition of transverse shear stiffness values.

For a single-layer shell with isotropic material, default transverse shear stiffnesses are:

$$E = \begin{bmatrix} kGh & \\ & kGh \end{bmatrix}$$

In the above matrix, k = 5/6, G = shear modulus, and h = thickness of the shell.

SHELL181 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties. Only isotropic, anisotropic, and orthotropic linear elastic properties can be input for elasticity. The von Mises isotropic hardening plasticity models can be invoked with BISO (bilinear isotropic hardening), MISO (multilinear isotropic hardening), and NLISO (nonlinear isotropic hardening) options. The kinematic hardening plasticity models can be invoked with BKIN (bilinear kinematic hardening), MKIN and KINH (multilinear kinematic hardening), and CHABOCHE (nonlinear kinematic hardening). Invoking plasticity assumes that the elastic properties are isotropic (that is, if orthotropic elasticity is used with plasticity, ANSYS assumes the isotropic elastic modulus = EX and Poisson's ratio = NUXY).

Hyperelastic material properties (2, 3, 5, or 9 parameter Mooney-Rivlin material model, Neo-Hookean model, Polynomial form model, Arruda-Boyce model, and user-defined model) can be used with this element. Poisson's ratio is used to specify the compressibility of the material. If less than 0, Poisson's ratio is set to 0; if greater than or equal to 0.5, Poisson's ratio is set to 0.5 (fully incompressible).

Both isotropic and orthotropic thermal expansion coefficients can be input using **MP**,ALPX. When used with hyperelasticity, isotropic expansion is assumed.

Use the **BETAD** command to specify the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to specify the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

With reduced integration and hourglass control (KEYOPT(3) = 0), low frequency spurious modes may appear if the mass matrix employed is not consistent with the quadrature rule. SHELL181 uses a projection scheme that effectively filters out the inertia contributions to the hourglass modes of the element. To be effective, a consistent mass matrix must be used. We recommend setting **LUMPM**,OFF for a modal analysis using this element type. The lumped mass option can, however, be used with the full integration options (KEYOPT(3) = 2).

KEYOPT(8) = 2 is used to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

KEYOPT(9) = 1 is used to read initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SHELL181 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL181 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0

UX, UY, UZ if KEYOPT(1) = 1

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, ADMSUA
E11, E22, E12, DRILL, MEMBRANE, BENDING
See *Table 1, "SHELL181 Real Constants"* for more information.
If a SHELL181 element references a valid shell section type, any real constant data specified will be ignored.

Material Properties

EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ

Specify DAMP only once for the element (use **MAT** command to assign material property set). REFT may be provided once for the element, or may be assigned on a per layer basis. See the discussion in *SHELL181 Input Summary* for more details.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +N direction),
face 2 (I-J-K-L) (top, in -N direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

For KEYOPT(1) = 0 (Bending and membrane stiffness):

T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum). Hence, for one-layer elements, 8 temperatures are used.

For KEYOPT(1) = 1 (Membrane stiffness only):

T1, T2, T3, T4 for layer 1, T5, T6, T7, T8 for layer 2, similarly for all layers (4*NL maximum). Hence, for one-layer elements, 4 temperatures are used.

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ANEL)
Other material (USER, SDAMP)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death
Section definition for layered shells and preintegrated shell sections for input of homogenous section stiffnesses

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default)

1 --
Membrane stiffness only

2 --
Stress/strain evaluation only

KEYOPT(3)
Integration option:

0 --
Reduced integration with hourglass control (default)

2 --
Full integration with incompatible modes

KEYOPT(8)
Specify layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (multi-layer elements) (default)

1 --
Store data for TOP and BOTTOM, for all layers (multi-layer elements)

Note

Volume of data may be excessive.

2 --
Store data for TOP, BOTTOM, and MID for all layers; applies to single- and multi-layer elements

KEYOPT(9)
User thickness option:

0 --
No user subroutine to provide initial thickness (default)

1 --
Read initial thickness data from user subroutine UTHICK

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

Table 1 SHELL181 Real Constants

No.	Name	Description
1	TK(I)	Thickness at node I
2	TK(J)	Thickness at node J
3	TK(K)	Thickness at node K
4	TK(L)	Thickness at node L
5	THETA	Angle of first surface direction, in degrees
6	ADMSUA	Added mass per unit area
7	E_{11}	Transverse shear stiffness[*2*]
8	E_{22}	Transverse shear stiffness[*2*]
9	E_{12}	Transverse shear stiffness[*2*]
10	Drill Stiffness Factor	In-plane rotation stiffness[*1*,*2*]
11	Membrane HG Factor	Membrane hourglass control factor[*1*,*2*]
12	Bending HG Factor	Bending hourglass control factor[*1*,*2*]

1. Valid values for these real constants are any positive number. However, we recommend using values between 1 and 10. If you specify 0.0, the value defaults to 1.0.

2. ANSYS provides default values.

*See **SECCONTROLS** command if section definition is used.

SHELL181 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL181 Element Output Definitions"*

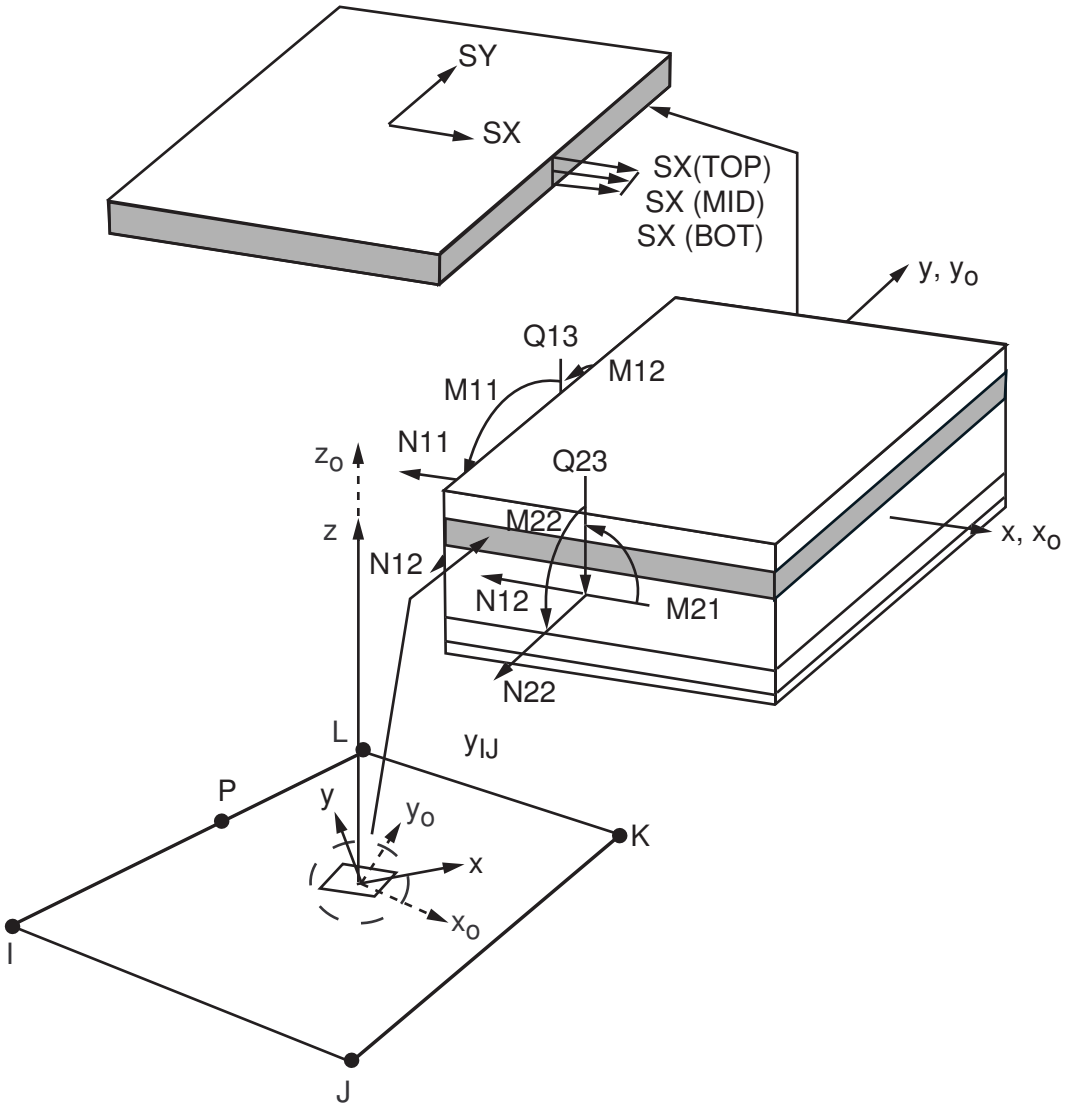
Several items are illustrated in *Figure 3, "SHELL181 Stress Output"*.

KEYOPT(8) controls the amount of data output to the results file for processing with the **LAYER** command. Interlaminar shear stress is available as SYZ and SXZ evaluated at the layer interfaces. KEYOPT(8) must be set to either 1 or 2 to output these stresses in POST1. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear forces Q13, Q23 are available only in resultant form: that is, use SMISC,7 (or 8). Likewise, the transverse shear strains, γ_{13} and γ_{23}, are constant through the thickness and are only available as SMISC items (SMISC,15 and SMISC,16, respectively).

SHELL181 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, we suggest using **OUTRES** to ensure that the required results are stored in the database.

Figure 3 SHELL181 Stress Output



x_o = Element x-axis if ESYS is not provided.
x = Element x-axis if ESYS is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL181 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y

Name	Definition	O	R
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	*4*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J,I; P4 at K,J; P5 at L,K; P6 at I,L	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*3*	*1*
S:INT	Stress intensity	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	*3*	*1*
EPEL:EQV	Equivalent elastic strains [*7*]	*3*	*1*
EPTH:X, Y, Z, XY	Thermal strains	*3*	*1*
EPTH:EQV	Equivalent thermal strains [*7*]	*3*	*1*
EPPL:X, Y, Z, XY	Average plastic strains	*3*	*2*
EPPL:EQV	Equivalent plastic strains [*7*]	*3*	*2*
EPCR:X, Y, Z, XY	Average creep strains	*3*	*2*
EPCR:EQV	Equivalent creep strains [*7*]	*3*	*2*
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	*2*
NL:CREQ	Accumulated equivalent creep strain	-	*2*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	*2*
NL:PLWK	Plastic work	-	*2*
NL:HPRES	Hydrostatic pressure	-	*2*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*2*
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	*8*
Q13, Q23	Transverse shear forces (per unit length)	-	*8*
$\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{12}$	Membrane strains	-	Y
k_{11}, k_{22}, k_{12}	Curvatures	-	*8*
γ_{13}, γ_{23}	Transverse shear strains	-	*8*
LOCI:X, Y, Z	Integration point locations	-	*5*
SVAR:1, 2, ... , N	State variables	-	*6*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material.

3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all five section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the USERMAT subroutine and **TB**,STATE are used.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
8. Not available if the membrane element option is used (KEYOPT(1) = 1).

Table 3, "SHELL181 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SHELL181 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SHELL181 Element Output Definitions"*

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 3 SHELL181 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	J	K	L	
N11	SMISC	1	-	-	-	-	
N22	SMISC	2	-	-	-	-	
N12	SMISC	3	-	-	-	-	
M11	SMISC	4	-	-	-	-	
M22	SMISC	5	-	-	-	-	
M12	SMISC	6	-	-	-	-	
Q13	SMISC	7	-	-	-	-	
Q23	SMISC	8	-	-	-	-	
ε_{11}	SMISC	9	-	-	-	-	
ε_{22}	SMISC	10	-	-	-	-	
ε_{12}	SMISC	11	-	-	-	-	
k_{11}	SMISC	12	-	-	-	-	
k_{22}	SMISC	13	-	-	-	-	
k_{12}	SMISC	14	-	-	-	-	
γ_{13}	SMISC	15	-	-	-	-	
γ_{23}	SMISC	16	-	-	-	-	
THICK	SMISC	17	-	-	-	-	

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	E	I	J	K	L	
P1	SMISC	-	18	19	20	21	
P2	SMISC	-	22	23	24	25	
P3	SMISC	-	27	26	-	-	
P4	SMISC	-	-	29	28	-	
P5	SMISC	-	-	-	31	30	
P6	SMISC	-	32	-	-	33	

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SHELL181 Assumptions and Restrictions

- Zero area elements are not allowed (this occurs most often whenever the elements are not numbered properly).
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed (but zero thickness layers are allowed).
- In a nonlinear analysis, the solution is terminated if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).
- We do not recommend using this element in triangular form.
- This element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,ON). For nonlinear problems dominated by large rotations and loading, we recommend that you *not* use **PRED**,ON.
- If reduced integration is used (KEYOPT(3) = 0) SHELL181 will ignore rotary inertia effects when a unbalanced laminate construction is used.
- If reduced integration is used (KEYOPT(3) = 0) all inertial effects are assumed to be in the nodal plane, i.e., an unbalanced laminate construction and offsets have no effect on the mass properties of the element.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers may not change between load steps.
- The section definition will permit use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- Transverse shear stiffness of the shell section is estimated by an energy equivalence procedure (of the generalized section forces & strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.

- A maximum of 250 layers is supported.
- We recommend the use of KEYOPT(3) = 2 for most composite analysis (necessary to capture the stress gradients).
- The layer orientation angle has no effect if the material of the layer is hyperelastic.
- If a shell section has only one layer and the number of section integration points is equal to one, or if KEYOPT(1) = 1, then the shell does not have any bending stiffness. This may result in solver difficulties, and may adversely affect convergence.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.
- The through-thickness stress, SZ, is always zero.
- When the element is associated with preintegrated shell sections (**SECTYPE**,,GENS), additional restrictions apply. For more information, see *Section 17.3.2: Considerations for Employing Preintegrated Shell Sections*.

SHELL181 Product Restrictions

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

PLANE182

2-D 4-Node Structural Solid

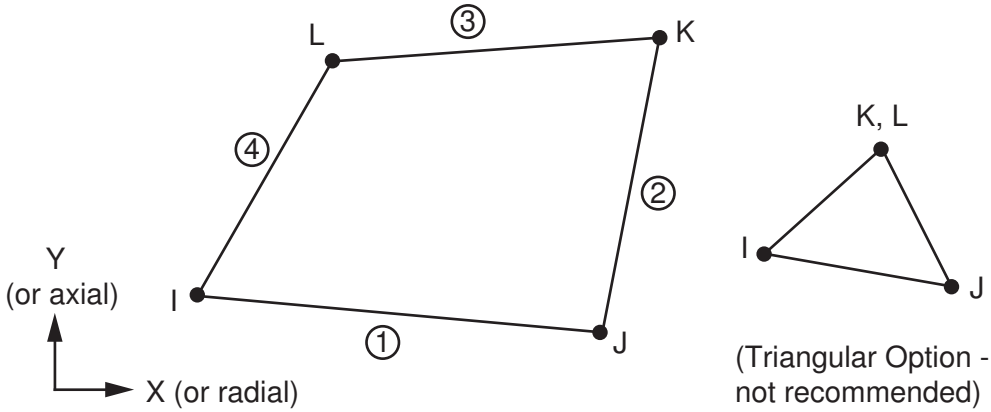
MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

PLANE182 Element Description

PLANE182 is used for 2-D modeling of solid structures. The element can be used as either a plane element (plane stress, plane strain or generalized plane strain) or an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

See PLANE182 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE182 Geometry



PLANE182 Input Data

The geometry and node locations for this element are shown in *Figure 1, "PLANE182 Geometry"*. The element input data includes four nodes, a thickness (for the plane stress option only), and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "PLANE182 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3 or KEYOPT(3) = 5) and on a full 360° basis for an axisymmetric analysis.

KEYOPT(3) = 5 is used to enable generalized plane strain. For more information about the generalized plane strain option, see *Section 2.11: Generalized Plane Strain Option of 18x Solid Elements* in the *Elements Reference*.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

As described in *Section 2.3: Coordinate Systems*, you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

PLANE182 Input Summary contains a summary of the element input. For a general description of element input, see *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE182 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY

Real Constants

THK - Thickness (used ony if KEYOPT(3) = 3)
HGSTF - Hourglass stiffness scaling factor (used only if KEYOPT(1) = 1); default is 1.0 (if you input 0.0, the default value is used)

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features --

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import

Nonlinear stabilization
Manual rezoning
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. CAST, EDP, SMA, and UNIAXIAL are not applicable for plane stress. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Element technology:

0 --
Full integration with B-bar method

1 --
Uniform reduced integration with hourglass control

2 --
Enhanced strain formulation

3 --
Simplified enhanced strain formulation

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness input

5 --
Generalized plane strain

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed u-P formulation (not valid with plane stress)

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses (default).

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

PLANE182 Element Technology

PLANE182 uses the $\bar{B}$ method (also known as the selective reduced integration method), the uniform reduced integration method, or the enhanced strain formulation method, as follows:

- **$\bar{B}$ method (selective reduced integration)**

  Helps to prevent volumetric mesh locking in nearly incompressible cases. This option replaces volumetric strain at the Gauss integration point with the average volumetric strain of the elements. This method cannot, however, prevent any shear locking in bending dominated problems. In such situations, use the enhanced strain formulation of this element. If it is not clear if the deformation is bending dominated, enhanced strain formulation is recommended. For more information, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Uniform reduced integration**

  Also helps to prevent volumetric mesh locking in nearly incompressible cases. Because it has only one integration point, this option is more efficient than the $\bar{B}$ method (selective reduced integration) option. However, the artificial energy introduced to control the hourglass effect may affect solution accuracy adversely.

  When using this option, check the solution accuracy by comparing the total energy (SENE label in **ETABLE**) and the artificial energy (AENE label in **ETABLE**) introduced by hourglass control. If the ratio of artificial energy to total energy is less than 5%, the solution is generally acceptable. If the ratio exceeds five percent, refine the mesh. You can also monitor the total energy and artificial energy by issuing the **OUTPR**,VENG command in the solution phase.

  For more information about uniform reduced integration, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Enhanced strain formulation**

  Prevents shear locking in bending-dominated problems and volumetric locking in nearly incompressible cases. The formulation introduces 4 *internal* DOFs (inaccessible to ANSYS users) to overcome shear locking in plane strain, axisymmetric problems, and generalized plane strain problems (all with mixed u-P formulations), and plane stress. For plane strain, axisymmetric problems, and generalized plane strain (all with pure displacement formulations), an additional internal DOF is introduced for volumetric locking (for a total of 5 internal DOFs). All internal DOFs are introduced automatically at the element level and condensed out.

  Because of the extra internal DOFs and static condensation, this option is less efficient than either the $\bar{B}$ method (selective reduced integration) option or the uniform reduced integration option.

For more information about enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Simplified enhanced strain formulation**

  Prevents shear locking in bending-dominated problems. This is a special case of the enhanced strain formulation and always introduces four *internal* DOFs (inaccessible to ANSYS users). For the plane stress state, this formulation is the same as the enhanced strain formulation, so only KEYOPT(1) = 2 is allowed. Because there are no internal DOFs to handle volumetric locking, this formulation should not be used when the material is nearly incompressible, except when the Mixed u-P formulation is also used. When used with the Mixed u-P formulation, the simplified enhanced strain formulation gives the same results as the enhanced strain formulation. All internal DOFs are introduced automatically at the element level and condensed out.

  Because of the extra internal DOFs and static condensation, this option is less efficient than either the $\bar{B}$ method (selective reduced integration) option or the uniform reduced integration option, but is more efficient than the enhanced strain formulation due to using fewer internal DOFs.

  For more information about the simplified enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

PLANE182 Output Data

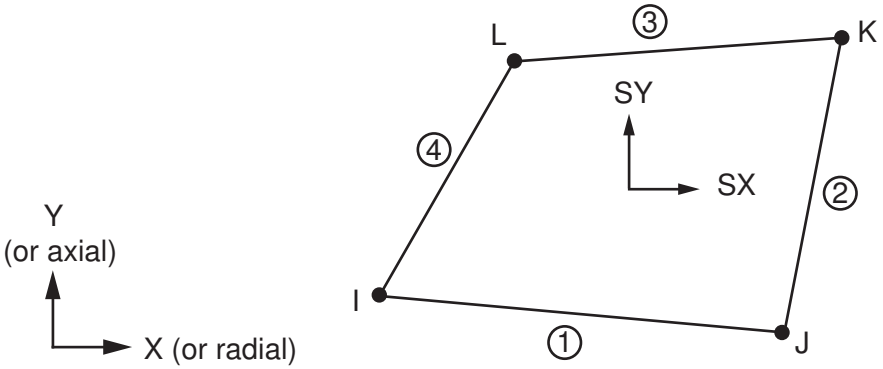
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE182 Element Output Definitions"*

Several items are illustrated in *Figure 2, "PLANE182 Stress Output"*.

The element stress directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE182 Stress Output



Stress directions are shown for Global.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE182 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain [*6*]	Y	Y
EPTH:X, Y, Z, XY	Thermal strains	*2*	*2*
EPTH:EQV	Equivalent thermal strain [*6*]	*2*	*2*
EPPL:X, Y, Z, XY	Plastic strains[*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strain [*6*]	*1*	*1*
EPCR:X, Y, Z, XY	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent plastic strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:PLWK	Plastic work	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*

1. Nonlinear solution, output only if the element has a nonlinear material.
2. Output only if element has a thermal load.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the USERMAT subroutine and **TB**,STATE are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Note

For axisymmetric solutions in a global coordinate system, the X, Y, Z, and XY stress and strain outputs correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively.

Table 2, "PLANE182 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE182 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "PLANE182 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE**

E
: sequence number for single-valued or constant element data

I,J,K,L
: sequence number for data at nodes I, J, K, L

Table 2 PLANE182 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P1	SMISC	-	2	1	-	-
P2	SMISC	-	-	4	3	-
P3	SMISC	-	-	-	6	5
P4	SMISC	-	7	-	-	8
THICK	NMISC	1	-	-	-	-

PLANE182 Assumptions and Restrictions

- The area of the element must be nonzero.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE182 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- You can form a triangular element by defining duplicate K and L node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). For triangular elements where the $\bar{B}$ or enhanced strain formulations are specified, degenerated shape functions and a conventional integration scheme are used.
- If you use the mixed formulation (KEYOPT(6) = 1), you must use either the sparse solver (default) or the frontal solver.
- For modal cyclic symmetry analyses, ANSYS recommends using enhanced strain formulation.

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

PLANE182 Product Restrictions

There are no product-specific restrictions for this element.

PLANE183

2-D 8-Node or 6-Node Structural Solid

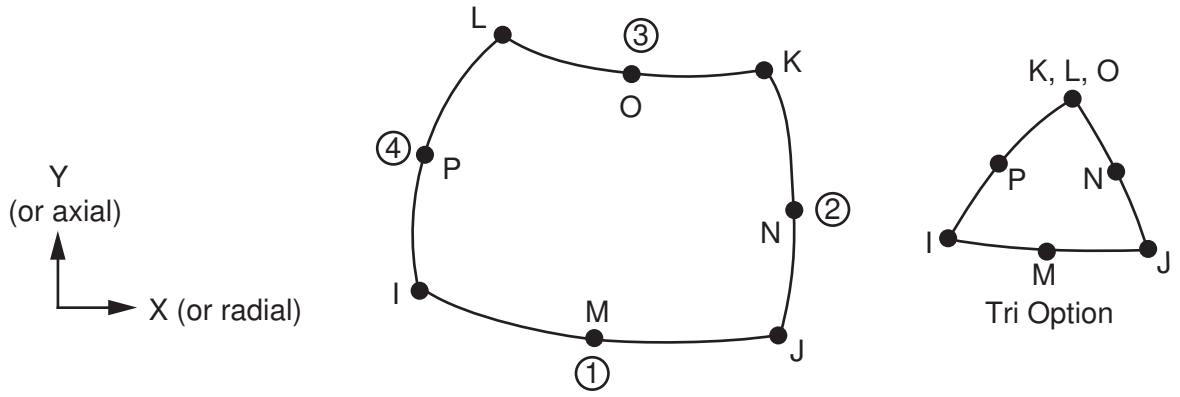
MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

PLANE183 Element Description

PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

This element is defined by 8 nodes or 6-nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element. This element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. Initial stress import is supported. Various printout options are also available. See PLANE183 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE183 Geometry



PLANE183 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE183 Geometry"*.

For KEYOPT(1) = 0, a degenerated triangular-shaped element may be formed by defining the same node number for nodes K, L and O. In addition to the nodes, the element input data includes a thickness (TK) (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "PLANE183 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3 or KEYOPT(3) = 5) and on a full 360° basis for an axisymmetric analysis.

As described in *Section 2.3: Coordinate Systems*, you can use **ESYS** to orient the material properties and strain/stress output. Use **ESYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(3) = 5 is used to enable generalized plane strain. For more information about the generalized plane strain option, see *Section 2.11: Generalized Plane Strain Option of 18x Solid Elements* in the *Elements Reference*.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE183 Input Summary

Nodes

I, J, K, L, M, N, O, P when KEYOPT(1) = 0

I, J, K, L, M, N when KEYOPT(1) = 1)

Degrees of Freedom

UX, UY

Real Constants

None, if KEYOPT (3) = 0, 1, or 2
THK - Thickness if KEYOPT (3) = 3

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I), face 2 (K-J), face 3 (I-K), face 4 (I-L) when KEYOPT(1) = 0

face 1 (J-I), face 2 (K-J), face 3 (I-K) when KEYOPT(1) = 1

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P) when KEYOPT(1) = 0

T(I), T(J), T(K), T(L), T(M), T(N) when KEYOPT(1) = 1

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Manual rezoning
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. CAST, EDP, SMA, and UNIAXIAL are not applicable for plane stress. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Element shape:

0 --
8-node quadrilateral

1 --
6-node triangle

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (TK) real constant input

5 --
Generalized plane strain

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed u-P formulation (not valid with plane stress)

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

PLANE183 Output Data

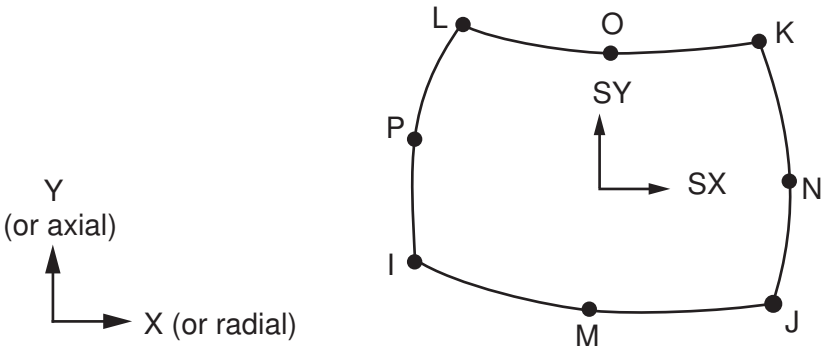
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE183 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "PLANE183 Stress Output"*.

The element stress directions are parallel to the element coordinate system. Surface stresses are defined parallel and perpendicular to the IJ face (and the KL face) and along the Z-axis for a plane analysis or in the hoop direction for an axisymmetric analysis. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 PLANE183 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE183 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L (for KEYOPT(1) = 0 and I, J, K (for KEY-OPT(1) = 1)	-	Y
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*4*
PRES	Pressures P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L (P4 only for KEYOPT(1) = 0	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L) only for KEYOPT(1) = 0	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S: INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [*7*]	-	Y
EPTH:X, Y, Z, XY	Thermal strains	*3*	*3*
EPTH:EQV	Equivalent thermal strain [*7*]	-	*3*
EPPL:X, Y, Z, XY	Plastic strains[*8*]	*1*	*1*
EPPL:EQV	Equivalent plastic strain [*7*]	-	*1*
EPCR:X, Y, Z, XY	Creep strains	*2*	*2*
EPCR:EQV	Equivalent creep strains [*7*]	*2*	*2*
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:PLWK	Plastic work	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*5*
SVAR:1, 2, ... , N	State variables	-	*6*

1. Nonlinear solution, output only if the element has a nonlinear material.
2. Output only if element has a creep load.
3. Output only if element has a thermal load.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the USERMAT subroutine and **TB**,STATE are used.

7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
8. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Note

For axisymmetric solutions, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains.

Table 2, "PLANE183 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE183 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "PLANE183 Element Output Definitions"*

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 2 PLANE183 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	-	-	-	-	-	-
P2	SMISC	-	-	4	3	-	-	-	-	-
P3	SMISC	-	-	-	6	5	-	-	-	-
P4[*1*]	SMISC	-	7	-	-	8	-	-	-	-
THICK	NMISC	1	-	-	-	-	-	-	-	-

1. P4 is only for KEYOPT(1) = 0

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for **ETABLE**.

PLANE183 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE183 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- Use at least two elements to avoid hourglass mode for KEYOPT(1) = 0.

- A triangular element may be formed by defining duplicate K-L-O node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*). For these degenerated elements, the triangular shape function is used and the solution is the same as for the regular triangular 6-node elements, but might be slightly less efficient for KEYOPT(1) = 0. Since these degenerated elements are less efficient, the triangle shape option (KEYOPT(1) = 1) is suggested for this case.
- When mixed formulation is used (KEYOPT(6) = 1), no midside nodes can be missed. If you use the mixed formulation (KEYOPT(6) = 1), you must use either the sparse solver (default) or the frontal solver.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

PLANE183 Product Restrictions

There are no product-specific restrictions for this element.

MPC184

Multipoint Constraint Element

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Element Description

MPC184 comprises a general class of multipoint constraint elements that apply kinematic constraints between nodes. The elements are loosely classified here as "constraint elements" (rigid link, rigid beam, and slider) and "joint elements" (x-axis revolute, z-axis revolute, universal, slot, point-in-plane, translational, x-axis cylindrical, z-axis cylindrical, x-axis planar, z-axis planar, weld, orient, spherical, and general). The constraint may be as simple as that of identical displacements between nodes. Constraints can also be more complicated, such as those modeling rigid parts, or those transmitting motion between flexible bodies in a particular way. For example, a structure may consist of rigid parts and moving parts connected together by rotational or sliding connections. The rigid part of the structure may be modeled with the MPC184 link/beam elements, while the moving parts may be connected with any of the MPC184 joint elements.

The kinematic constraints are imposed using one of the following two methods:

- The **direct elimination method**, wherein the kinematic constraints are imposed by internally generated constraint equations. The degrees of freedom of a dependent node in the equations are eliminated in favor of an independent node.
  - The dependent degrees of freedom are eliminated. Therefore, the constraint forces and moments are not available from the element output table (**ETABLE**) for output purposes. However, the global constraint reaction forces are available at independent nodes in the results file, Jobname.rst (**PRRSOL** command, etc.).
  - The direct elimination method should be used whenever it is available since the degrees of freedom at the dependent nodes are eliminated, thereby reducing the problem size and solution time.
- The **Lagrange multiplier method**, wherein the kinematic constraints are imposed using Lagrange multipliers. In this case, all the participating degrees of freedom are retained.
  - The Lagrange multiplier method should be used when the direct elimination method is not available or not suitable for the analysis purposes.
  - In this method, the constraint forces and moments are available from the element output table (**ETABLE**).
  - The disadvantage of the Lagrange multiplier method is that the Lagrange multipliers are additional solution variables and, hence, the problem size and solution time become larger when compared with the direct elimination method.

Currently, the MPC184 rigid link/beam elements can use the direct elimination method or the Lagrange multiplier method. All other MPC184 element options use the Lagrange multiplier method only.

Constraint Elements

The following types of constraint elements are available:

Rigid link/beam
Slider

Joint Elements

Numerical simulations often involve modeling of joints between two parts. These joints or connections may need simple kinematic constraints such as identical displacements between the two parts at the junction or more complicated kinematic constraints that allow for transmission of motion between two flexible bodies. These complex joints may also include some sort of control mechanism like limits or stops, and locks on the components of relative motion between the two bodies. In many instances, these joints may also have stiffness, damping, or friction forces based on the unconstrained components of relative motion between the two bodies. For detailed information on how to use joint elements, see *Section 2.3: Connecting Multibody Components with Joint Elements* in the *Multibody Analysis Guide*.

The following types of joint elements are available:

x-axis Revolute joint
z-axis Revolute joint
Universal joint
Slot joint
Point-in-plane joint
Translational joint
x-axis Cylindrical joint
z-axis Cylindrical joint
x-axis Planar joint
z-axis Planar joint
Weld joint
Orient joint
Spherical joint
General joint

These elements are well suited for linear, large rotation, and/or large strain nonlinear applications. If finite rotations and/or large strain effects are to be considered, the **NLGEOM**,ON command must be used; otherwise, linear behavior is assumed. For example, if a revolute joint element is used in an analysis and **NLGEOM**,ON is not set, the calculations are carried out in the original configuration and the end result may not reflect the expected deformed configuration. However, if the **NLGEOM**,ON command is used, the calculations will take into account the rotation of the revolute joint element.

Two nodes define these joint elements. Depending on the joint to be defined, the kinematic constraints are imposed on some of the quantities that define the relative motion between the two nodes. These kinematic constraints are applied using Lagrange multipliers. In some instances, one of the nodes is required to be "grounded" or attached to "ground" or some other reference location that is not moving. In such cases, only one of the two nodes may be specified. The specified node and the "grounded" node are assumed to be coincident in the element calculations.

The joint element has six degrees of freedom at each node, defining six components of relative motion: three relative displacements and three relative rotations. These six components of relative motion are of primary interest in simulations that involve joint elements. Some of these components may be constrained by the kinematic constraints relevant to a particular joint element, while the other components are "free" or "unconstrained". For example, in the case of universal and revolute joint elements the two nodes are assumed to be connected, and thus the relative displacements are zero. For the revolute joint only one rotational component of the relative motion (rotation about the revolute axis) is unconstrained, while for the universal joint two such components are available.

The capabilities of these elements include certain control features such as stops, locks, and actuating loads/boundary conditions that can be imposed on the components of relative motion between the two nodes

of the element. For example, in a revolute joint, stops can be specified for the rotation about the revolute axis. This limits the rotation around the revolute axis to be within a certain range. Displacement, force, velocity, and acceleration boundary conditions may be imposed on the components of relative motion between the two nodes allowing for "actuation" of the joints. The driving force or displacements arise from the actuating mechanisms like an electric or hydraulic system that drives these joints.

You can impose linear and nonlinear elastic stiffness and damping behavior or hysteretic friction behavior on the available components of relative motion of a joint element. The properties can be made temperature dependent if necessary.

In addition to the existing output options available in ANSYS, outputs related to the components of relative motion are available for joint elements.

Joint Input Data

Certain input requirements are common to most MPC184 joint elements. Any specific requirements for individual joint elements are highlighted in the description for that element.

The following types of input data should be considered:

- Element Connectivity Definition - A joint element is typically defined by specifying two nodes, I and J. One of these nodes may be a "grounded" node.
- Section Definition - Each joint element must have an associated section definition (**SECTYPE** command).
- Local Coordinate System Specification - Local coordinate systems at the nodes are often required to define the kinematic constraints of a joint element (**SECJOINT** command).
- Stops or Limits - You can impose stops or limits on the available components of relative motion between the two nodes of a joint element (**SECSTOP** command).
- Locks - Locking limits may also be imposed on the available components of relative motion between the two nodes of a joint element to "freeze" the joint in a desired configuration (**SECLOCK** command).
- Material Behavior - The JOIN material option on the **TB** command allows you to impose linear and nonlinear elastic stiffness and damping behavior or hysteretic friction behavior on the available components of relative motion of a joint element.
- Reference Lengths and Angles - These correspond to the free relative degrees of freedom in a joint element for which constitutive calculations are performed and are used when stiffness, damping, or hysteretic friction are specified for the joint elements (**SECDATA** command).
- Boundary Conditions - You can impose boundary conditions (**DJ** command) or apply concentrated forces (**FJ** command) on the available components of relative motion of the joint element.

MPC184 Input Data

Use KEYOPT(1) to specify the type of MPC184 constraint or joint element you want to use. The remaining input data will vary depending on the type of constraint or joint element specified. The individual MPC184 element descriptions each contain an input summary that applies only to that particular element. It is recommended that you review these element-specific input summaries after you determine which constraint or joint element you will be using.

KEYOPT(1)

Element behavior:

0 --

Rigid link (default)

1 --
Rigid beam

3 --
Slider element

6 --
x-axis or z-axis revolute joint element

7 --
Universal joint element

8 --
Slot joint element

9 --
Point-in-plane joint element

10 --
Translational joint element

11 --
x-axis or z-axis cylindrical joint element

12 --
x-axis or z-axis planar joint element

13 --
Weld joint element

14 --
Orient joint element

15 --
Spherical joint element

16 --
General joint element

MPC184 Output Data

The solution output associated with the constraint and joint elements is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in the individual constraint and joint element descriptions. This output is available via the **ETABLE** command using the Sequence Number method.

Refer to the individual element descriptions for complete listings of the output for each element.

MPC184 Assumptions and Restrictions

The following restrictions apply to all forms of the MPC184 element:

- For MPC184, the element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

There are additional assumptions and restrictions for each type of constraint and joint element. For details, see the Assumptions and Restrictions section in the individual constraint and joint element descriptions.

MPC184 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The DAMP material property is not allowed.
- No special features are allowed.

MPC184-Link/Beam

Multipoint Constraint Element: Rigid Link or Rigid Beam

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

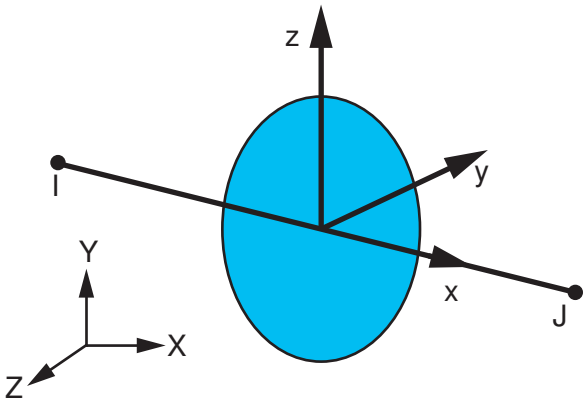
MPC184 Rigid Link/Beam Element Description

The MPC184 rigid link/beam element can be used to model a rigid constraint between two deformable bodies or as a rigid component used to transmit forces and moments in engineering applications. This element is well suited for linear, large rotation, and/or large strain nonlinear applications.

The kinematic constraints are imposed using one of the following two methods:

- The **direct elimination method**, wherein the kinematic constraints are imposed by internally generated MPC (multipoint constraint) equations. The degrees of freedom of a dependent node in the MPC equations are eliminated in favor of an independent node.
- The **Lagrange multiplier method**, wherein the kinematic constraints are imposed using Lagrange multipliers. In this case, all the participating degrees of freedom are retained.

Figure 1 MPC184 Rigid Link/Beam Geometry



MPC184 Rigid Link/Beam Input Data

Figure 1, "MPC184 Rigid Link/Beam Geometry" shows the geometry, node locations, and the coordinate system for this element. Two nodes define the element. The element x-axis is oriented from node I toward node J. The cross-sectional area of the element is assumed to be one unit. ANSYS selects the cross-section coordinate system automatically; see BEAM4 (without orientation node) for a description of the method employed. The cross-section coordinate system is relevant only for the output of bending moments when the element is used as a rigid beam.

If KEYOPT(1) = 0 (default), the element is a rigid link with two nodes and three degrees of freedom at each node (UX, UY, UZ). If KEYOPT(1) = 1, the element is a rigid beam with two nodes and six degrees of freedom at each node (UX, UY, UZ, ROTX, ROTY, ROTZ).

If KEYOPT(2) = 0 (default), then the constraints are implemented using the direct elimination method. If KEYOPT(2) = 1, then the Lagrange multiplier method is used to impose the constraints.

The MPC184 rigid link/beam element with KEYOPT(2) = 1 can also be used in applications that call for thermal expansion on an otherwise rigid structure. The direct elimination method cannot be used for thermal expansion problems.

Because the element models a rigid constraint or a rigid component, material stiffness properties are not required. When thermal expansion effects are desired, the coefficient of thermal expansion must be specified. Density must be specified if the mass of the rigid element is to be accounted for in the analysis. If density is specified, ANSYS calculates a lumped mass matrix for the element.

The element supports the birth and death options using **EALIVE** and **EKILL**.

Section 2.8: Node and Element Loads describes element loads. You can input temperatures as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I).

MPC184 Rigid Link/Beam Input Summary

This input summary applies to the rigid link and rigid beam options of MPC184: KEYOPT(1) = 0 and 1.

Nodes
I, J

Degrees of Freedom
UX, UY, UZ if KEYOPT(1) = 0

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 1

Real Constants
None

Material Properties
ALPX (or CTEX *or* THSX), DENS

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
None

Special Features

Large deflection
Birth and death

KEYOPT(1)
Element behavior:

0 --
Rigid link (default)

1 --
Rigid beam

KEYOPT(2)
Reduction method:

0 --
Direct elimination method (default)

1 --
Lagrange multiplier method

MPC184 Rigid Link/Beam Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Rigid Link/Beam Element Output Definitions"*.

Table 1, "MPC184 Rigid Link/Beam Element Output Definitions" uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Rigid Link/Beam Element Output Definitions

Name	Definition	O	R
Link/Beam Elements (KEYOPT(1) = 0 or 1, and KEYOPT(2) = 0 or 1)			
EL	Element number	-	Y
NODES	Element node numbers (I and J)	-	Y
Link/Beam Elements (KEYOPT(1) = 0 or 1, and KEYOPT(2) = 1)			
MAT	Material number for the element	-	Y
TEMP	Temperature at nodes I and J	-	Y
FX	Axial force	-	Y
MY, MZ	Bending moments	-	Y
SF:Y, Z	Section shear forces	-	Y
MX	Torsional moment	-	Y

Table 2, "MPC184 Rigid Link/Beam Item and Sequence Numbers" lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
: output quantity as defined in the Element Output Definitions table.

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 MPC184 Rigid Link/Beam Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Link/Beam Constraint (with KEYOPT(2) = 1)		
FX	SMISC	1
MY	SMISC	2
MZ	SMISC	3

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	4
SFZ	SMISC	5
SFY	SMISC	6

MPC184 Rigid Link/Beam Assumptions and Restrictions

The following restrictions apply to both the direct elimination method and the Lagrange multiplier method (KEYOPT(2) = 0 and 1):

- A finite element model cannot be made up of only rigid elements in a static analysis. At a minimum, a deformable element (or elements) must be connected to one of the end nodes of a rigid element.
- The cross-sectional area of the element is assumed to be unity.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

Direct Elimination Method (KEYOPT(2) = 0)

These additional restrictions apply to the direct elimination method:

- The MPC184 rigid link/beam using the direct elimination method can be used in static, transient, modal, and buckling analyses.
- This element can be used with the SPARSE, PCG, JCG, ICCG, AMG, and ITER solvers (**EQSLV** command). It cannot be used with the FRONT and DSPARSE solvers.
- Displacement boundary conditions on the nodes of rigid link/beams must be applied prudently. In a rigid linkage (structure) made of a number of rigid link/beam elements, if displacement boundary conditions are applied at more than one location, ANSYS will use the first encountered displacement boundary condition to constrain the entire rigid linkage according to rigid kinematic conditions. In some cases where the applied displacements may be redundant or self-contradictory, ANSYS will issue warning or error messages.
- The direct elimination method cannot be used in problems involving thermal expansion. Use the Lagrange Multiplier method instead.
- Reaction forces at the constrained nodes of a rigid link/beam may not always be available since the dependent and independent nodes are determined by ANSYS internally. We recommend that you check the interface nodes which connect rigid and deformable elements since reaction forces are available on these nodes.
- The nodes of a rigid link/beam using the direct elimination method should not be linked with a node of an element implemented via the Lagrange multiplier method. For example, a rigid beam implemented using the direct elimination method (KEYOPT(2) = 0) should not be linked to a rigid beam implemented via the Lagrange multiplier method (KEYOPT(2) = 1). Or, a rigid beam implemented via the direct elimination method should not be linked to a node of a contact element that is implemented via the Lagrange multiplier method (KEYOPT(2) = 2 on the contact element).
- Coupling constraints (**CP** command) cannot be applied to nodes of rigid links/beams using the direct elimination method.
- Nodes of rigid links/beams cannot be part of the retained nodes (nodes specified by the **M** command) in a substructure. However, the rigid links/beams can be entirely within the substructure.
- Rigid links/beams should be not used in cyclic symmetry analyses.

- Rigid links/beams cannot be used with Distributed ANSYS.

Lagrange Multiplier Method (KEYOPT(2) = 1)

These additional restrictions apply to the Lagrange Multiplier method:

- To employ this feature successfully, use as few of these elements as possible. For example, it may be sufficient to overlay rigid line elements on a perimeter of a rigid region modeled with shell elements, as opposed to overlaying rigid line elements along each element boundary of the interior.
- Modeling that avoids overconstraining the problem is necessary. Overconstrained models may result in trivial solutions, zero pivot messages (in a properly restrained system), or nonlinear convergence difficulties.
- The temperature is assumed to vary linearly along the spar of the rigid link or rigid beam element.
- If constraint equations are specified for the DOFs of a rigid element, it may be an overconstrained system. Similarly, prescribed displacements on both ends of the element is an indication of overconstraint.
- When used as a link element, exercise the same precautions that you would when using a truss element (for example, LINK180 or LINK8).
- In most cases, the equation solver (**EQSLV**) must be the sparse solver. If you use this element in a harmonic analysis, you must employ the frontal solver.
- The element is valid for static and transient analyses (linear and nonlinear), and rigid beam is valid for harmonic response analyses. The element is not supported for buckling analyses or reduced transient analyses.

MPC184 Rigid Link/Beam Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Slider

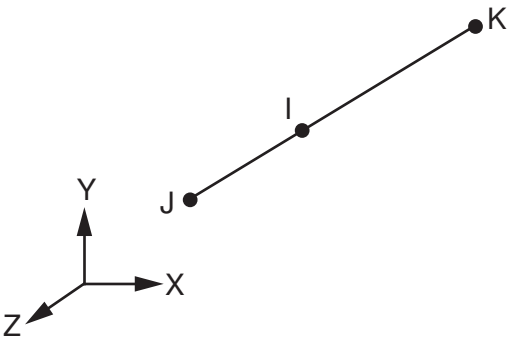
Multipoint Constraint Element: Slider

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Slider Element Description

The MPC184 slider element can be used to model a slider constraint. The element has three nodes with three degrees of freedom (translations in X, Y, and Z) at each node. The slider element imposes a kinematic constraint such that a "dependent" node (I) must always lie on a line joining two other "independent" nodes (J and K). The I node is allowed to slide on the line joining J and K nodes. The kinematic constraints are imposed using the Lagrange multiplier method.

Figure 1 MPC184 Slider Geometry



MPC184 Slider Input Data

Set KEYOPT(1) = 3 to define a three-node slider element.

Figure 1, "MPC184 Slider Geometry" shows the geometry and node locations for this element. Three nodes (I, J, and K) define the element. The node I is expected to lie initially on the line joining the nodes J and K.

Material stiffness properties are not required for this element. The element currently does not support birth or death options.

MPC184 Slider Input Summary

This input summary applies to the slider element option of MPC184: KEYOPT(1) = 3.

Nodes
I, J, K

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

3 --
Slider element

MPC184 Slider Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Slider Element Output Definitions"*.

Table 1, "MPC184 Slider Element Output Definitions" uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Slider Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J, K)	-	Y
FY	Constraint Force 1	-	Y
FZ	Constraint Force 2	-	Y

Table 2, "MPC184 Slider Item and Sequence Numbers" lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 MPC184 Slider Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	1
FZ	SMISC	2

MPC184 Slider Assumptions and Restrictions

- The distance between nodes I and J must be greater than zero.
- Node I must initially lie between the J and K nodes.
- Displacement boundary conditions cannot be applied on the nodes forming the slider element.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Slider Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Spherical

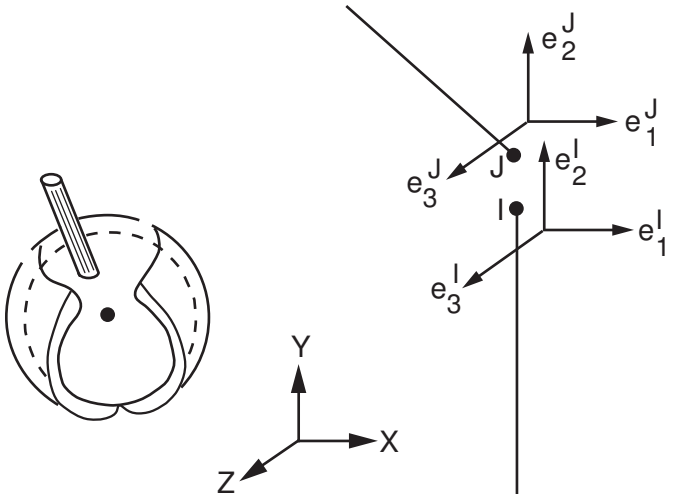
Multipoint Constraint Element: Spherical Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Spherical Joint Element Description

The MPC184 spherical joint element is a two-node element with the relative displacement degrees of freedom constrained. The relative rotational degrees of freedom are left unconstrained. These rotations cannot be controlled. The kinematic constraints are imposed using the Lagrange multiplier method.

Figure 1 MPC184 Spherical Joint Geometry



MPC184 Spherical Joint Input Data

Set KEYOPT(1) = 15 to define a two-node spherical joint element.

Figure 1, "MPC184 Spherical Joint Geometry" shows the geometry and node locations for this element. Two nodes define the element. The two nodes (I and J) are expected to have identical spatial locations initially. If the two nodes are not coincident, the relative positions of the two nodes are maintained.

A local Cartesian coordinate system should be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, the local coordinate system at node J is assumed to be the same as that at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a spherical joint element are described below. Referring to *Figure 1, "MPC184 Spherical Joint Geometry"*, the constraints imposed at any given time are as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

For output purposes, the relative rotations between nodes I and J are characterized by the Cardan (or Bryant) angles as follows:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\varphi = -\sin^{-1}(e_1^I \cdot e_3^J)$$

$$\chi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

Since the output of relative rotations is characterized by the Cardan (or Bryant) angles, the rotation around the local e_2 axis is limited to between -PI/2 to +PI/2 (see the expression for Φ above). When this rotation value reaches |PI/2|, the other two angles become indeterminate. Therefore, if the accumulated angles around an axis of rotation is greater than |PI/2|, the axis of rotation should typically be specified as the local e_1 or e_3 axis.

Since the relative rotational degrees of freedom cannot be controlled, the spherical joint element does not allow stops and locks or material behavior specifications. Other input data that are common to all joint elements are described in *Joint Input Data* in the MPC184 element description.

MPC184 Spherical Joint Input Summary

This input summary applies to the spherical joint element option of MPC184: KEYOPT(1) = 15.

Nodes
I, J,

Note

For a grounded spherical joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

15 --
Spherical joint element

MPC184 Spherical Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Spherical Joint Element Output Definitions"* and *Table 2, "MPC184 Spherical Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Spherical Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JRP5	Joint relative position of DOF 5	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRU5	Joint relative rotation of DOF 5	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRV5	Joint relative rotational velocity of DOF 5	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JRA5	Joint relative rotational acceleration of DOF 5	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the spherical joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Spherical Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Spherical Joint Item and Sequence Numbers -SMISC Items" and *Table 4, "MPC184 Spherical Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
: output quantity as defined in the Element Output Definitions table.

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 MPC184 Spherical Joint Item and Sequence Numbers -SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
JRP4-6	SMISC	34-36
JRU4-6	SMISC	64-66
JRV4-6	SMISC	70-72
JRA4-6	SMISC	76-78

Table 4 MPC184 Spherical Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Spherical Joint Assumptions and Restrictions

- The nodes I and J should be coincident. If the nodes are not coincident, the relative positions between the two nodes are maintained.
- Boundary conditions cannot be applied on the nodes forming the spherical element.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the spherical joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. For the values to be accumulated correctly, it is essential that the substep size be restricted such that the rotation in a given substep is less than π.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Spherical Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Revolute

Multipoint Constraint Element: Revolute Joint

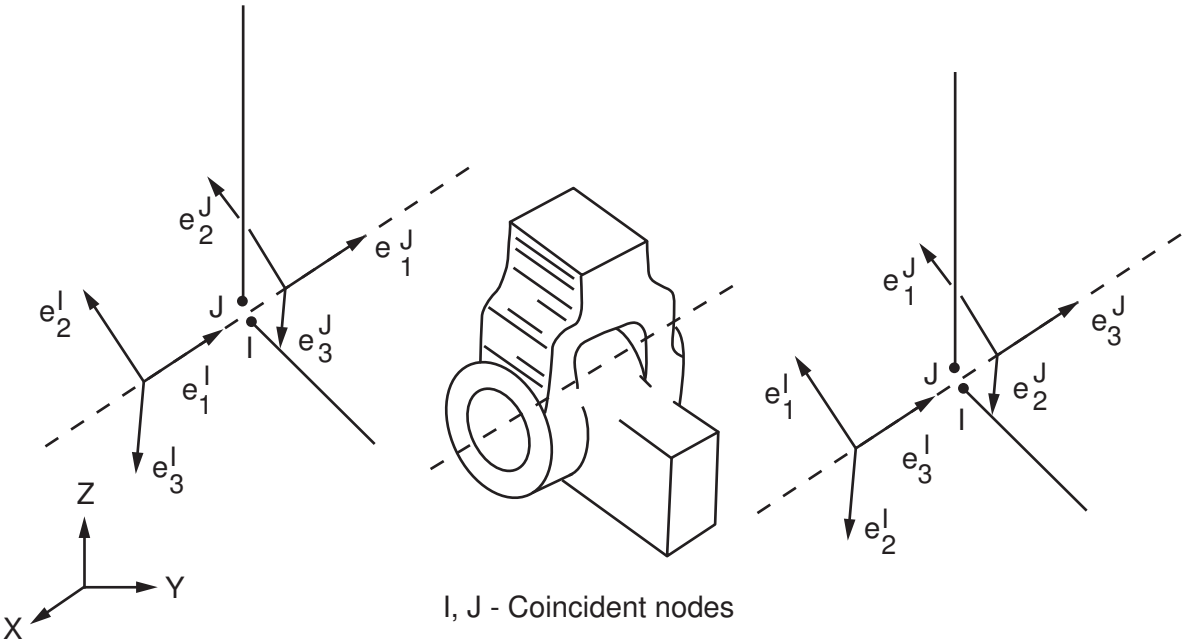
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Revolute Joint Element Description

The MPC184 revolute joint is a two-node element that has only one primary degree of freedom, the relative rotation about the revolute (or hinge) axis. This element imposes kinematic constraints such that the nodes forming the element have the same displacements. Additionally, only a relative rotation is allowed about the revolute axis, while the rotations about the other two directions are fixed.

Another revolute joint element in the ANSYS element library is the COMBIN7 element. The COMBIN7 element allows you to control the joint flexibility, friction, damping, and certain control features. A local coordinate system is fixed to and moves with the joint thereby allowing the element to be used in large deflection analysis (see *Section 14.7: COMBIN7 - Revolute Joint* for additional details).

Figure 1 MPC184 Revolute Joint Geometry



Local x-axis as axis of revolute

Local z-axis as axis of revolute

MPC184 Revolute Joint Input Data

Set KEYOPT(1) = 6 to define a two-node revolute joint element.

Figure 1, "MPC184 Revolute Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element. The two nodes are expected to have identical spatial coordinates initially.

If KEYOPT(4) = 0, then element is an x-axis revolute joint with the local e_1 axis as the revolute axis.

If KEYOPT(4) = 1, then element is a z-axis revolute joint with the local e_3 axis as the revolute axis.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, then the local coordinate system at node J is assumed to be the same as that at node I.

Either the local e_1 or local e_3 direction may be specified as the axis of rotation at the nodes. The specification of the other two local directions is not critical, but it will be used to determine the relative rotation between the two nodes during the course of deformation. The orientation of local directions must follow the convention specified in *Figure 1, "MPC184 Revolute Joint Geometry"*. These local coordinate systems evolve with the rotations at the respective nodes (if any). Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a revolute joint element with the local e_1 axis as the revolute axis are described below. Similar constraint conditions are set up when the local e_3 axis is the revolute axis.

Consider the two local coordinate systems (Cartesian) attached to node I and node J (see *Figure 1, "MPC184 Revolute Joint Geometry"*). At any given instant of time, the constraints imposed in a revolute joint are as described below.

Displacement constraints:

$$u^I = u^J$$

Where, u^I is the displacement vector at node I and u^J is the displacement vector at node J.

Rotation constraints:

$$e_1^I \cdot e_2^J = 0$$

$$e_1^I \cdot e_3^J = 0$$

If the revolute axes e_1^I and e_1^J are not aligned at the start of the analysis, then the angle between the two is held fixed at the starting value.

The relative position of the local coordinate system at node I with respect to node J is characterized by the first Cardan (or Bryant) angle given by:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

The change in the relative angular position between the two local coordinate system is given by:

$$u_r = \phi - \phi_0 + m\pi$$

Where, ϕ_0 is the initial angular offset (the first Cardan (or Bryant) angle measured in the reference configuration) between the two coordinate systems and m is an integer accounting for multiple rotations about the revolute axis.

The constitutive calculations use the following definition of the joint rotation:

$$u_r^c = \phi + m\pi - \phi_1^{ref}$$

Where ϕ_1^{ref} is the reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then ϕ_0 is used in place of ϕ_1^{ref}.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Revolute Joint Input Summary

This input summary applies to the revolute joint element option of MPC184: KEYOPT(1) = 6.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads

Rotations --

ROTX (KEYOPT(4) = 0)
ROTZ (KEYOPT(4) = 1)

Angular velocities --

OMGX (KEYOPT(4) = 0)
OMGZ (KEYOPT(4) = 1)

Angular accelerations --

DMGX (KEYOPT(4) = 0)
DMGZ (KEYOPT(4) = 1)

Moments --

MX (KEYOPT(4) = 0)
MZ (KEYOPT(4) = 1)

Special Features

Large deflection

KEYOPT(1)
Element behavior:

6 --
Revolute joint element

KEYOPT(4)
Element configuration:

0 --
x-axis revolute joint with local 1 direction as the revolute axis.

1 --
z-axis revolute joint with local 3 direction as the revolute axis.

MPC184 Revolute Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Revolute Joint Element Output Definitions"* and *Table 2, "MPC184 Revolute Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Revolute Joint Element Output Definitions

Name	Definition	O	R
x-axis Revolute Joint (KEYOPT(4) = 0)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y
MY	Constraint Moment in Y direction	-	Y
MZ	Constraint Moment in Z direction	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST4	Constraint stop status[*1*]	-	Y

Name	Definition	O	R
CLST4	Constraint lock status[*2*]	-	Y
JRP4	Joint relative position	-	Y
JCD4	Joint constitutive rotation	-	Y
JEF4	Joint elastic moment	-	Y
JDF4	Joint damping moment	-	Y
JFF4	Joint friction moment	-	Y
JRU4	Joint relative rotation	-	Y
JRV4	Joint relative velocity	-	Y
JRA4	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y
z-axis Revolute Joint (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST6	Constraint stop status[*1*]	-	Y
CLST6	Constraint lock status[*2*]	-	Y
JRP6	Joint relative position	-	Y
JCD6	Joint constitutive rotation	-	Y
JEF6	Joint elastic moment	-	Y
JDF6	Joint damping moment	-	Y
JFF6	Joint friction moment	-	Y
JRU6	Joint relative rotation	-	Y
JRV6	Joint relative velocity	-	Y
JRA6	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the revolute joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Revolute Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all revolute joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Revolute Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Revolute Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
: output quantity as defined in the Element Output Definitions table.

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 MPC184 Revolute Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Revolute Joint (KEYOPT(4) = 0)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP4	SMISC	10
CLOCK4	SMISC	16
CSST4	SMISC	22

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CLST4	SMISC	28
JRP4	SMISC	34
JCD4	SMISC	40
JEF4	SMISC	46
JDF4	SMISC	52
JFF4	SMISC	58
JRU4	SMISC	64
JRV4	SMISC	70
JRA4	SMISC	76
JTEMP	SMISC	79
z-axis Revolute Joint (KEYOPT(4) = 1)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
CSTOP6	SMISC	12
CLOCK6	SMISC	18
CSST6	SMISC	24
CLST6	SMISC	30
JRP6	SMISC	36
JCD6	SMISC	42
JEF6	SMISC	48
JDF6	SMISC	54
JFF6	SMISC	60
JRU6	SMISC	66
JRV6	SMISC	72
JRA6	SMISC	78
JTEMP	SMISC	79

Table 4 MPC184 Revolute Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all revolute joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Revolute Joint Assumptions and Restrictions

- The nodes I and J must be coincident.
- The local coordinate systems at the nodes must be specified such that the revolute axis is well defined. Otherwise, it is possible that the rotational motion might not be what is expected.
- Boundary conditions cannot be applied on the nodes forming the revolute joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the Revolute Joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the component of relative motion (rotation around the revolute axis) is accumulated over all the substeps. It is essential that the substep size be restricted such that this rotation in a given substep is less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Revolute Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Universal

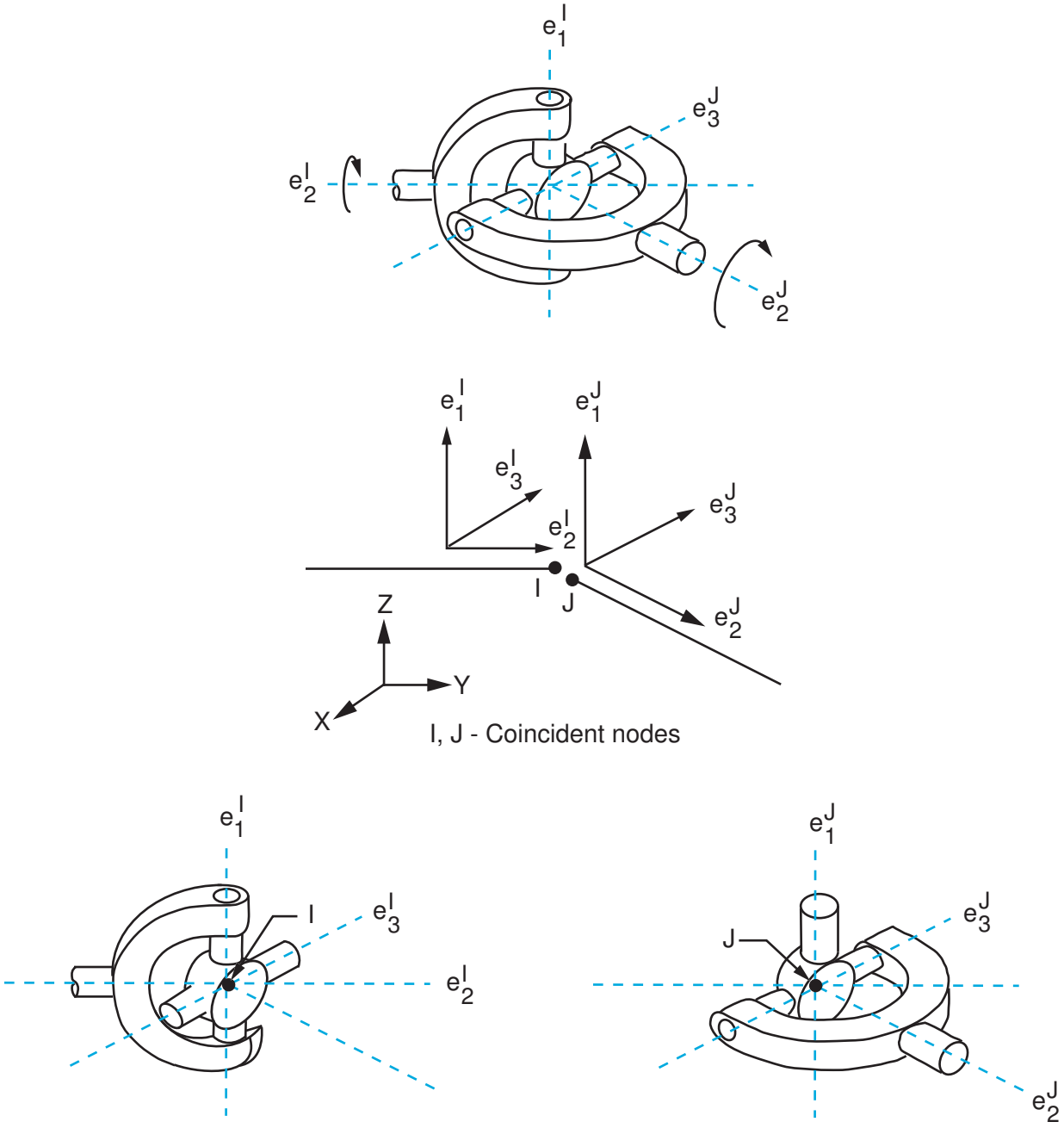
Multipoint Constraint Element: Universal Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Universal Joint Element Description

The MPC184 universal joint element is a two-node element that has two free relative rotational degrees of freedom. The two nodes forming the element must have identical spatial coordinates.

Figure 1 MPC184 Universal Joint Geometry



MPC184 Universal Joint Input Data

Set KEYOPT(1) = 7 to define a two-node universal joint element.

Figure 1, "MPC184 Universal Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element. The two nodes are expected to have identical spatial coordinates.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, then the local coordinate system at node J is assumed to be the same as that at node I. The local 2 direction is usually aligned along the shaft axes of the universal joint. The orientation of local directions must follow the convention specified in *Figure 1, "MPC184 Universal Joint Geometry"*. These local coordinate systems evolve with the rotations at the respective nodes (if any). Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a universal joint element are easily described by considering the two local coordinate systems (Cartesian) attached to node I and node J (*Figure 1, "MPC184 Universal Joint Geometry"*). At any given instant of time, the constraints imposed in a universal joint are as described below.

Displacement constraints:

$$u^I = u^J$$

Where, u^I is the displacement vector at node I, and u^J is the displacement vector at node J.

Rotation constraints:

$$e_1^I \cdot e_3^J = 0$$

If the axes e_2^I and e_2^J are not aligned at the start of the analysis, then the angle between the two is held fixed at the initial value.

The relative position of the local coordinate system at node I with respect to node J is characterized by the first and the third Cardan (or Bryant) angles as:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\psi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

The change in the relative angular position between the two local coordinate system is given by

$$u_{r4} = \phi - \phi_0$$

$$u_{r6} = \psi - \psi_0$$

Where, ϕ_0 and ψ_0 are the initial angular offsets between the two coordinate systems (that is, the first and third Cardan (or Bryant) angles measured in the reference configuration).

The constitutive calculations use the following definition of the joint rotation:

$$u^c_{r4} = \phi - \phi^{ref}_1$$

$$u^c_{r6} = \psi - \phi^{ref}_3$$

Where, ϕ^{ref}_1, ϕ^{ref}_3 are the reference angles, angle1 and angle3, specified on the **SECDATA** command. If these values are not specified, then ϕ_0 and ψ_0 are used in place of ϕ^{ref}_1 and ϕ^{ref}_3, respectively.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Universal Joint Input Summary

This input summary applies to the universal joint element option of MPC184: KEYOPT(1) = 7.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads

Rotations --
ROTX, ROTZ

Moments --
MX, MZ

Special Features

Large deflection

KEYOPT(1)
Element behavior:

7 --
Universal joint element

MPC184 Universal Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Universal Joint Element Output Definitions"* and *Table 2, "MPC184 Universal Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Universal Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MY	Constraint moment in Y direction	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST4	Constraint stop status on DOF 4[*1*]	-	Y
CLST4	Constraint lock status on DOF 4[*2*]	-	Y
CSST6	Constraint stop status on DOF 6[*1*]	-	Y
CLST6	Constraint lock status on DOF 6[*2*]	-	Y
JRP4	Joint relative position of DOF4	-	Y
JRP6	Joint relative position of DOF6	-	Y
JCD4	Joint constitutive rotation on DOF4	-	Y
JCD6	Joint constitutive rotation on DOF6	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JFF4	Joint friction moment in direction -4	-	Y
JFF6	Joint friction moment in direction -6	-	Y
JRU4	Joint relative rotation of DOF4	-	Y

Name	Definition	O	R
JRU6	Joint relative rotation of DOF6	-	Y
JRV4	Joint relative rotational velocity of DOF4	-	Y
JRV6	Joint relative rotational velocity of DOF6	-	Y
JRA4	Joint relative rotational acceleration of DOF4	-	Y
JRA6	Joint relative rotational acceleration of DOF6	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for the universal joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Universal Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Universal Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Universal Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
   output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Universal Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP4	SMISC	10
CSTOP6	SMISC	12
CLOCK4	SMISC	16
CLOCK6	SMISC	18
CSST4	SMISC	22
CLST4	SMISC	28
CSST6	SMISC	24
CLST6	SMISC	30
JRP4	SMISC	34
JRP6	SMISC	36
JCD4	SMISC	40
JCD6	SMISC	42
JEF4	SMISC	46
JEF6	SMISC	48
JDF4	SMISC	52
JDF6	SMISC	54
JFF4	SMISC	58
JFF6	SMISC	60
JRU4	SMISC	64
JRU6	SMISC	66
JRV4	SMISC	70
JRV6	SMISC	72
JRA4	SMISC	76
JRA6	SMISC	78
JTEMP	SMISC	79

Table 4 MPC184 Universal Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Universal Joint Assumptions and Restrictions

- The nodes I and J must be coincident.
- The local coordinate systems at the nodes must be specified such that the axes of rotation are well defined. Otherwise, it is possible that the rotational motion might not be what is expected.
- Boundary conditions cannot be applied on the nodes forming the universal joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the universal joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Universal Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Slot

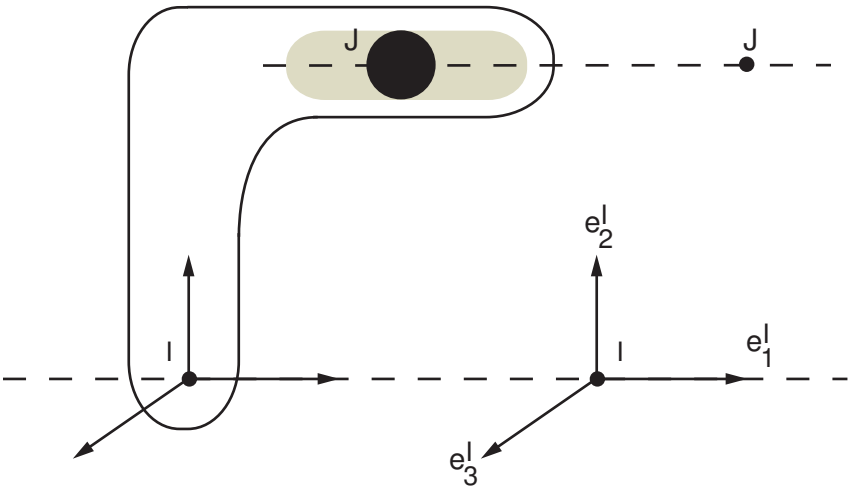
Multipoint Constraint Element: Slot Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Slot Joint Element Description

The MPC184 slot joint element is a two-node element that has one relative displacement degree of freedom. The rotational degrees of freedom at nodes I and J are left free.

Figure 1 MPC184 Slot Joint Geometry



MPC184 Slot Joint Input Data

Set KEYOPT(1) = 8 to define a two-node slot joint element.

Figure 1, "MPC184 Slot Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The second node, J, is constrained to move on the local $\mathbf{e}_1$ axis specified at node I. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a slot joint element are easily described by referring to *Figure 1, "MPC184 Slot Joint Geometry"*. At any given instant of time, the constraints imposed in a 3-D slot joint are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$
$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Where, $\mathbf{x}^I$ and $\mathbf{x}^J$ are the position vectors of nodes I and J in the current configuration, and $\mathbf{X}^I$ and $\mathbf{X}^J$ are the position vectors of nodes I and J in the reference configuration. Essentially these constraints force the node J to move along the $\mathbf{e}_1$ axis of the local coordinate system specified at node I. $\mathbf{e}^I$ are in the current configuration, while $\mathbf{E}^I$ are specified in the initial configuration.

The change in the relative position of the nodes I and J is given by:

$$u_1 = \ell - \ell_0$$

Where u is the initial offset computed based on the initial configuration and the local coordinate system associated with node I, and

$$\ell = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) \text{ and } \ell_0 = \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \ell - \ell_1^{ref}$$

where:

ℓ_1^{ref} = reference length, length1, specified on **SECDATA** command.

If the reference length is not specified, the initial offset is used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Slot Joint Input Summary

This input summary applies to the slot joint element option of MPC184: KEYOPT(1) = 8.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads

Displacements --
UX

Forces --
FX

Special Features

Large deflection

KEYOPT(1)
Element behavior:

8 --
Slot joint element

MPC184 Slot Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Slot Joint Element Output Definitions"* and *Table 2, "MPC184 Slot Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Slot Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CSST1	Constraint stop status[*1*]	-	Y
CLST1	Constraint lock status[*2*]	-	Y
JRP1	Joint relative position	-	Y
JCD1	Joint constitutive displacement	-	Y
JEF1	Joint elastic force	-	Y
JDF1	Joint damping force	-	Y
JFF1	Joint friction force	-	Y
JRU1	Joint relative displacement	-	Y
JRA1	Joint relative acceleration	-	Y
JRV1	Joint relative velocity	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

0 = stop not active, or deactivated
1 = stopped at minimum value
2 = stopped at maximum value

2. Constraint lock status:

0 = lock not active
1 = locked at minimum value
2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for the slot joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Slot Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Slot Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Slot Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Slot Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	2
FZ	SMISC	3
CSTOP1	SMISC	7
CLOCK1	SMISC	13
CSST1	SMISC	19
CLST1	SMISC	25
JRP1	SMISC	31
JCD1	SMISC	37
JEF1	SMISC	43
JDF1	SMISC	49
JFF1	SMISC	55
JRU1	SMISC	61
JRV1	SMISC	67
JRA1	SMISC	73
JTEMP	SMISC	79

Table 4 MPC184 Slot Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Slot Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the slot joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the slot joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) can only be applied on the relative x-direction. These are not applicable to the rotational degrees of freedom.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Slot Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Point

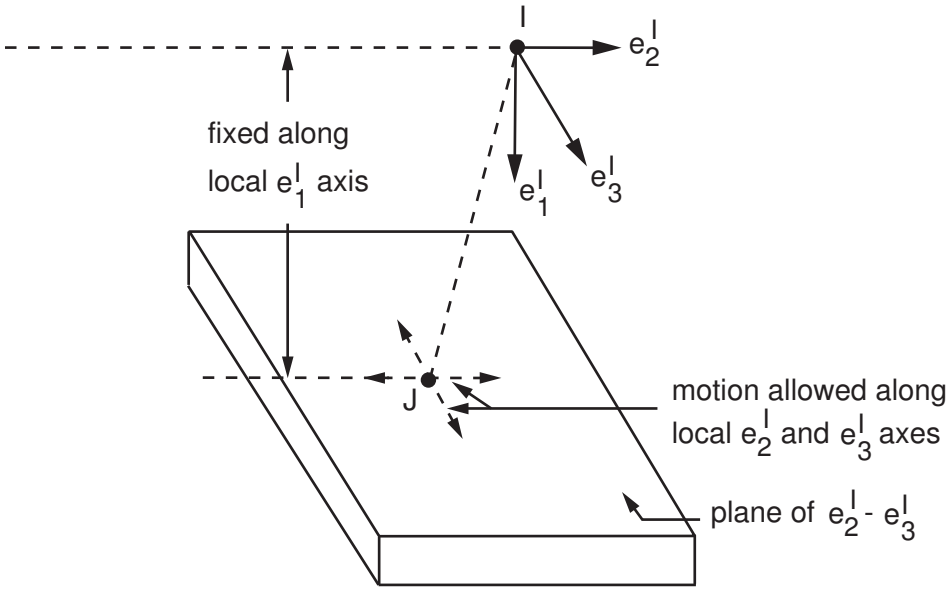
Multipoint Constraint Element: Point-in-plane Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Point-in-plane Joint Element Description

The MPC184 point-in-plane joint element is a two-node element that has two relative displacement degrees of freedom. The relative rotational degrees of freedom are not considered and cannot be controlled.

Figure 1 MPC184 Point-in-plane Joint Geometry



MPC184 Point-in-plane Joint Input Data

Set KEYOPT(1) = 9 to define a two-node point-in-plane joint element.

Figure 1, "MPC184 Point-in-plane Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The second node, J, is constrained such that it remains on a plane defined by the local e_1^I and e_2^I axes. The normal distance from this plane containing node J to node I is held fixed. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a point-in-plane joint element are easily described by referring to *Figure 1, "MPC184 Point-in-plane Joint Geometry"*. At any given instant of time, the constraint imposed is as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Where, x^I and x^J are the positional vectors of nodes I and J in the current configuration, and X^I and X^J are the position vectors of nodes I and J in the reference configuration. e^I are in the current configuration, while E^I are specified in the initial configuration.

The changes in the relative position of the nodes I and J are given by:

$$u_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_2^c = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

$$u_3^c = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_3^{ref}$$

where:

ℓ_2^{ref} = reference length specified on the **SECDATA** command

ℓ_3^{ref} = reference length specified on the **SECDATA** command

If the reference lengths are not specified, the initial offsets are used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Point-in-plane Joint Input Summary

This input summary applies to the point-in-plane joint element option of MPC184: KEYOPT(1) = 9.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

9 --
Point-in-plane joint element

MPC184 Point-in-plane Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Point-in-plane Joint Element Output Definitions"* and *Table 2, "MPC184 Point-in-plane Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Point-in-plane Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CSST2	Constraint stop status on DOF 2[*1*]	-	Y
CLST2	Constraint lock status on DOF 2[*2*]	-	Y
CSST3	Constraint stop status on DOF 3[*1*]	-	Y
CLST3	Constraint lock status on DOF 3[*2*]	-	Y
JRP2	Joint relative position of DOF2	-	Y
JRP3	Joint relative position of DOF3	-	Y
JCD2	Joint constitutive displacement on DOF2	-	Y
JCD3	Joint constitutive displacement on DOF3	-	Y
JEF2	Joint elastic force in direction -2	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JDF2	Joint damping force in direction -2	-	Y

Name	Definition	O	R
JDF3	Joint damping force in direction -3	-	Y
JFF2	Joint friction force in direction -2	-	Y
JFF3	Joint friction force in direction -3	-	Y
JRU2	Joint relative displacement in direction -2	-	Y
JRU3	Joint relative displacement in direction -3	-	Y
JRV2	Joint relative velocity in direction -2	-	Y
JRV3	Joint relative velocity in direction -3	-	Y
JRA2	Joint relative acceleration in direction -2	-	Y
JRA3	Joint relative acceleration in direction -3	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for the point-in-plane joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Point-in-plane Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Point-in-plane Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Point-in-plane Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the

Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Point-in-plane Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
CSTOP2	SMISC	8
CSTOP3	SMISC	9
CLOCK2	SMISC	14
CLOCK3	SMISC	15
CSST2	SMISC	20
CLST2	SMISC	26
CSST3	SMISC	21
CLST3	SMISC	27
JRP2	SMISC	32
JRP3	SMISC	33
JCD2	SMISC	38
JCD3	SMISC	39
JEF2	SMISC	44
JEF3	SMISC	45
JDF2	SMISC	50
JDF3	SMISC	51
JFF2	SMISC	56
JFF3	SMISC	57
JRU2	SMISC	62
JRU3	SMISC	63
JRV2	SMISC	68
JRV3	SMISC	69
JRA2	SMISC	74
JRA3	SMISC	75
JTEMP	SMISC	79

Table 4 MPC184 Point-in-plane Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Point-in-plane Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the point-in-plane joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the point-in-plane joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Point-in-plane Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Trans

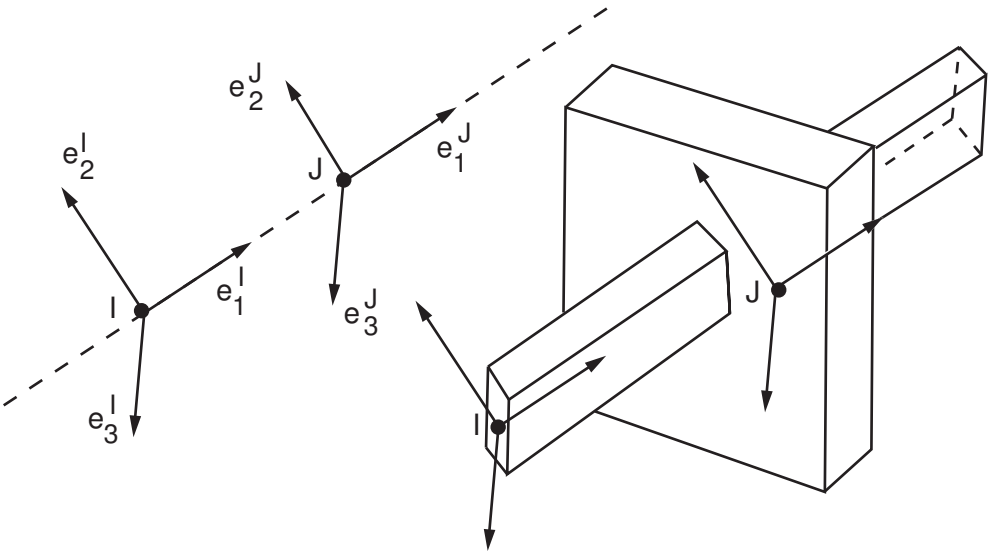
Multipoint Constraint Element: Translational Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Translational Joint Element Description

The MPC184 translational joint element is a two-node element that has one relative displacement degree of freedom. All other relative degrees of freedom are fixed.

Figure 1 MPC184 Translational Joint Geometry



MPC184 Translational Joint Input Data

Set KEYOPT(1) = 10 to define a two-node translational joint element.

Figure 1, "MPC184 Translational Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The second node J is constrained to move along the e_1 axis specified at node I. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a translational joint element are easily described by referring to *Figure 1, "MPC184 Translational Joint Geometry"*. At any given instant of time, the constraints imposed are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathrm{E}_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathrm{E}_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - \mathbf{E}_1^I \cdot \mathbf{E}_2^J = 0$$

$$\mathbf{e}_2^I \cdot \mathbf{e}_3^J - \mathbf{E}_2^I \cdot \mathbf{E}_3^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathbf{E}_1^I \cdot \mathbf{E}_3^J = 0$$

Where, x^I and x^J are the positional vectors of nodes I and J in the current configuration, and X^I and X^J are the position vectors of nodes I and J in the reference configuration. Essentially these constraints force the node J to move along the e_1 axis of the local coordinate system specified at node I. e^I are in the current configuration, while E^I are specified in the initial configuration.

The change in the relative position of the nodes I and J is given by:

$$u_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

where:

ℓ_1^{ref} = reference length, length1, specified on **SECDATA** command

If the reference length is not specified, the initial offset is used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Translational Joint Input Summary

This input summary applies to the translational joint element option of MPC184: KEYOPT(1) = 10.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

10 --
Translational joint element

MPC184 Translational Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Translational Joint Element Output Definitions"* and *Table 2, "MPC184 Translational Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Translational Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CSST1	Constraint stop status[*1*]	-	Y
CLST1	Constraint lock status[*2*]	-	Y
JRP1	Joint relative position	-	Y
JCD1	Joint constitutive displacement	-	Y
JEF1	Joint elastic force	-	Y
JDF1	Joint damping force	-	Y

Name	Definition	O	R
JFF1	Joint friction force	-	Y
JRU1	Joint relative displacement	-	Y
JRV1	Joint relative velocity	-	Y
JRA1	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for the translational joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Translational Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Translational Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Translational Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Translational Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6
CSTOP1	SMISC	7
CLOCK1	SMISC	13
CSST1	SMISC	19
CLST1	SMISC	25
JRP1	SMISC	31
JCD1	SMISC	37
JEF1	SMISC	43
JDF1	SMISC	49
JFF1	SMISC	55
JRU1	SMISC	61
JRV1	SMISC	67
JRA1	SMISC	73
JTEMP	SMISC	79

Table 4 MPC184 Translational Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Translational Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the translational joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the translational joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Translational Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Cylin

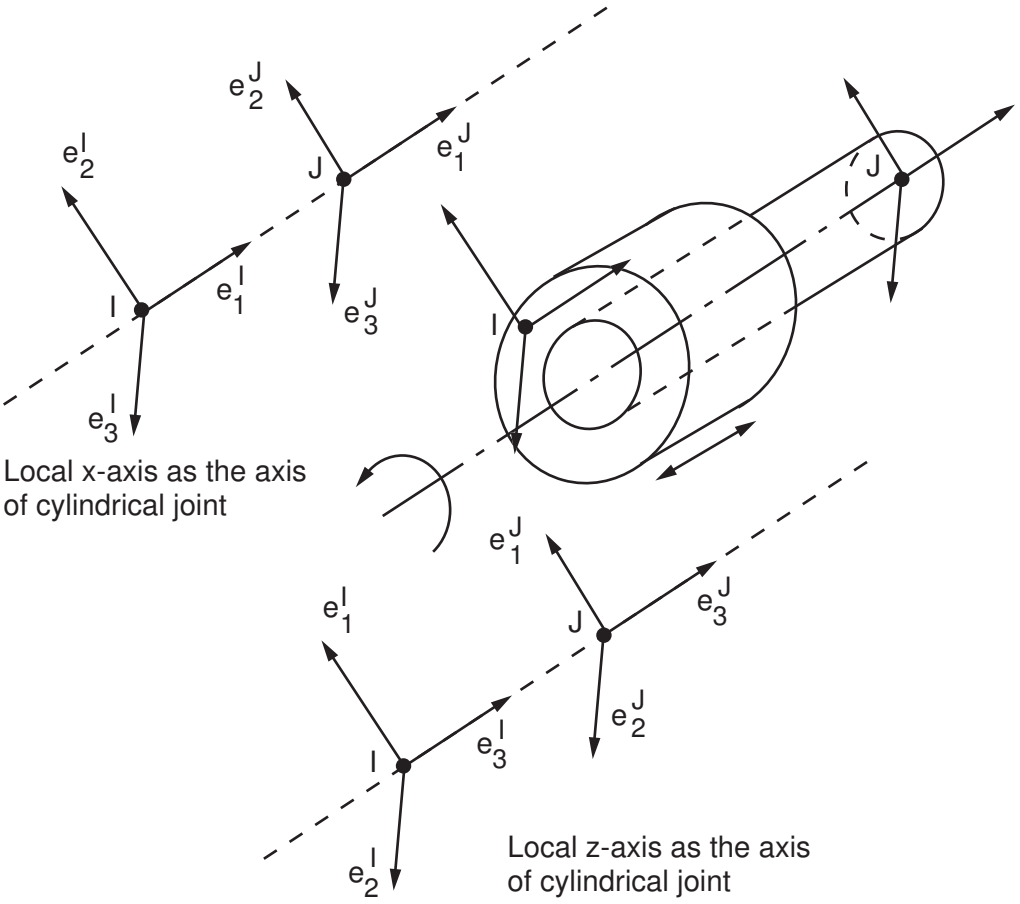
Multipoint Constraint Element: Cylindrical Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Cylindrical Joint Element Description

The MPC184 cylindrical joint element is a two-node element that has one free relative displacement degree of freedom and one free relative rotational degree of freedom (around the cylindrical or revolute axis). All other relative degrees of freedom are fixed.

Figure 1 MPC184 Cylindrical Joint Geometry



MPC184 Cylindrical Joint Input Data

Set KEYOPT(1) = 11 to define a two-node cylindrical joint element.

Figure 1,"MPC184 Cylindrical Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

If KEYOPT(4) = 0, then the element is an x-axis cylindrical joint element with the local e_1 axis as the cylindrical or revolute axis. Translational motion along this axis is also allowed.

If KEYOPT(4) = 1, then the element is a z-axis cylindrical joint element with the local e_3 axis as the cylindrical or revolute axis. Translational motion along this axis is also allowed.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a cylindrical joint element with local e_1 axis as the cylindrical or revolute axis are described below. Similar constraint conditions are set up when the local e_3 axis is the cylindrical or revolute axis. Referring to *Figure 1, "MPC184 Cylindrical Joint Geometry"*, with local coordinate systems specified at nodes I and J, the constraints imposed at any given time are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - \mathbf{E}_1^I \cdot \mathbf{E}_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathbf{E}_1^I \cdot \mathbf{E}_3^J = 0$$

The change in the relative position of the nodes I and J is given by:

$$u_1 = \ell - \ell_0$$

where:

$$\ell = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) \text{ and } \ell_0 = \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The relative rotation between nodes I and J is given by:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

The change in the relative angular position between the two local coordinate systems is given by

$$u_r = \phi - \phi_0 + m\pi$$

where ϕ_0 is the initial angular offset between the two coordinate systems and m is an integer accounting for multiple rotations about the cylindrical axis.

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

where:

ℓ_1^{ref} = reference length specified on **SECDATA** command.

The constitutive calculations use the following definition of the joint rotation:

$$u_{r4}^{c} = \phi + m\pi - \phi_1^{ref}$$

where:

ϕ_1^{ref} = reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then Φ_0 is used in place of ϕ_1^{ref}

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Cylindrical Joint Input Summary

This input summary applies to the cylindrical joint element option of MPC184: KEYOPT(1) = 11.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads when KEYOPT(4) = 0:

Displacements/Rotations --
UX, ROTX

Velocities --
VELX, OMGX

Accelerations --
ACCX, DMGX

Force/Moments --
FX, MX

Element Loads when KEYOPT(4) = 1:

Displacements/Rotations --
UZ, ROTZ

Velocities --
VELZ, OMGZ

Accelerations --
ACCZ, DMGZ

Force/Moments --
FZ, MZ

Special Features

Large deflection

KEYOPT(1)
Element behavior:

11 --
Cylindrical joint element

KEYOPT(4)
Element configuration:

0 --
x-axis Cylindrical joint with local 1 direction as the cylindrical axis.

1 --
z-axis Cylindrical joint with local 3 direction as the cylindrical axis.

MPC184 Cylindrical Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Cylindrical Joint Element Output Definitions"* and *Table 2, "MPC184 Cylindrical Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Cylindrical Joint Element Output Definitions

Name	Definition	O	R
x-axis Cylindrical Joint Element (KEYOPT(4) = 0)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y

Name	Definition	O	R
MY	Constraint Moment in Y direction	-	Y
MZ	Constraint Moment in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST1	Constraint stop status on DOF 1[*1*]	-	Y
CLST1	Constraint lock status on DOF 1[*2*]	-	Y
CSST4	Constraint stop status on DOF 4[*1*]	-	Y
CLST4	Constraint lock status on DOF 4[*2*]	-	Y
JRP1	Joint relative position of DOF 1	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JCD1	Joint constitutive displacement on DOF 1	-	Y
JCD4	Joint constitutive rotation on DOF 4	-	Y
JEF1	Joint elastic force in direction -1	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JDF1	Joint damping force in direction -1	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JFF1	Joint friction force in direction -1	-	Y
JFF4	Joint friction moment in direction -4	-	Y
JRU1	Joint relative displacement of DOF 1	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRV1	Joint relative velocity of DOF 1	-	Y
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRA1	Joint relative acceleration of DOF 1	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y
z-axis Cylindrical Joint Element (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST3	Constraint stop status on DOF 3[*1*]	-	Y
CLST3	Constraint lock status on DOF 3[*2*]	-	Y
CSST6	Constraint stop status on DOF 6[*1*]	-	Y
CLST6	Constraint lock status on DOF 6[*2*]	-	Y
JRP3	Joint relative position of DOF 3	-	Y
JRP6	Joint relative position of DOF 6	-	Y

Name	Definition	O	R
JCD3	Joint constitutive displacement on DOF 3	-	Y
JCD6	Joint constitutive rotation on DOF 6	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF3	Joint damping force in direction -3	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JFF3	Joint friction force in direction -3	-	Y
JFF6	Joint friction moment in direction -6	-	Y
JRU3	Joint relative displacement of DOF 3	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV3	Joint relative velocity of DOF 3	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA3	Joint relative acceleration of DOF 3	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the cylindrical joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Cylindrical Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all cylindrical joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y

Name	Definition	O	R
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Cylindrical Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Cylindrical Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Cylindrical Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Cylindrical Joint Element (KEYOPT(4) = 0)		
FY	SMISC	2
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP1	SMISC	7
CSTOP4	SMISC	10
CLOCK1	SMISC	13
CLOCK4	SMISC	16
CSST1	SMISC	19
CSST4	SMISC	22
CLST1	SMISC	25
CLST4	SMISC	28
JRP1	SMISC	31
JRP4	SMISC	34
JCD1	SMISC	37
JCD4	SMISC	40
JEF1	SMISC	43
JEF4	SMISC	46
JDF1	SMISC	49
JDF4	SMISC	52
JFF1	SMISC	55
JFF4	SMISC	58
JRU1	SMISC	61

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JRU4	SMISC	64
JRV1	SMISC	67
JRV4	SMISC	70
JRA1	SMISC	73
JRA4	SMISC	76
JTEMP	SMISC	79
z-axis Cylindrical Joint Element (KEYOPT(4) = 1)		
FX	SMISC	1
FY	SMISC	2
MX	SMISC	4
MY	SMISC	5
CSTOP3	SMISC	9
CSTOP6	SMISC	12
CLOCK3	SMISC	15
CLOCK6	SMISC	18
CSST3	SMISC	21
CSST6	SMISC	24
CLST3	SMISC	27
CLST6	SMISC	30
JRP3	SMISC	33
JRP6	SMISC	36
JCD3	SMISC	39
JCD6	SMISC	42
JEF3	SMISC	45
JEF6	SMISC	48
JDF3	SMISC	51
JDF6	SMISC	54
JFF3	SMISC	57
JFF6	SMISC	60
JRU3	SMISC	63
JRU6	SMISC	66
JRV3	SMISC	69
JRV6	SMISC	72
JRA3	SMISC	75
JRA6	SMISC	78
JTEMP	SMISC	79

Table 4 MPC184 Cylindrical Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all cylindrical joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Cylindrical Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the cylindrical joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the cylindrical joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Cylindrical Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Planar

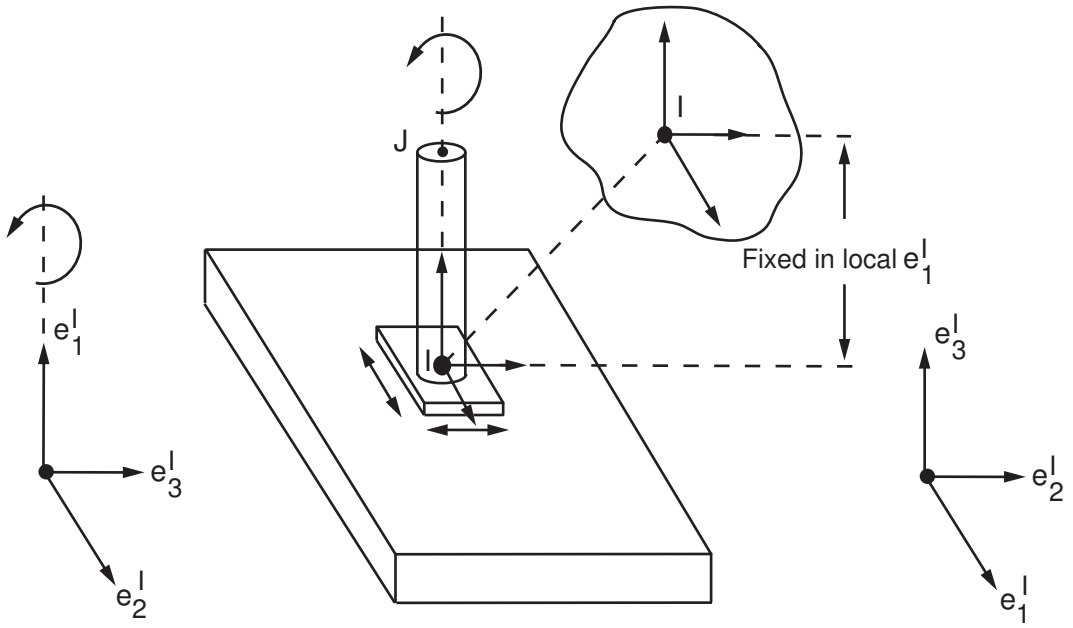
Multipoint Constraint Element: Planar Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Planar Joint Element Description

The MPC184 planar joint element is a two-node element that has two relative displacement degrees of freedom and one relative rotational degree of freedom. All other relative degrees of freedom are fixed.

Figure 1 MPC184 Planar Joint Geometry



Local x-axis as the axis of rotation

Local z-axis as the axis of rotation

MPC184 Planar Joint Input Data

Set KEYOPT(1) = 12 to define a two-node planar joint element.

Figure 1, "MPC184 Planar Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

If KEYOPT(4) = 0, the element is an x-axis planar joint element with the local e_1 axis as the rotation axis and also the axis along which the distance is fixed.

If KEYOPT(4) = 1, the element is a z-axis planar joint element with the local e_3 axis as the rotation axis and also the axis along which the distance is fixed.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a planar joint element with local e_1 axis as the axis of rotation are described below. Similar constraint conditions are set up when the local e_3 axis is the axis of rotation. Referring to *Figure 1, "MPC184 Planar Joint Geometry"*, the constraints imposed at any given time are as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - E_1^I \cdot E_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - E_1^I \cdot E_3^J = 0$$

The changes in the relative position of the nodes I and J are given by:

$$u_2 = \mathbf{e}_2 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_2 \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_3 \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The change in the relative angular position between the two local coordinate systems is given by:

$$u_r = \phi - \phi_0 + m\pi$$

The constitutive calculations use the following definition of the joint displacement:

$$u_2 = \mathbf{e}_2 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

$$u_3 = \mathbf{e}_3 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_3^{ref}$$

where:

ℓ_2^{ref} and ℓ_3^{ref} = reference lengths, length2 and length3, specified on the **SECDATA** command.

The constitutive calculations use the following definition of the joint rotation:

$$u_{r4}^c = \phi + m\pi - \phi_1^{ref}$$

where:

ϕ_1^{ref} = reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then ϕ_0 is used in place of ϕ_1^{ref}.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Planar Joint Input Summary

This input summary applies to the planar joint element option of MPC184: KEYOPT(1) = 12.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads when KEYOPT(4) = 0

Displacements/Rotations --
UY, UZ, ROTX

Velocities --
VELY, VELZ, OMGX

Accelerations --
ACCY, ACCZ, DMGX

Element Loads when KEYOPT(4) = 1

Displacements/Rotations --
UX, UY, ROTZ

Velocities --
VELX, VELY, OMGZ

Accelerations --
ACCX, ACCY, DMGZ

Special Features

Large deflection

KEYOPT(1)
Element behavior:

12 --
Planar joint element

KEYOPT(4)
Element configuration:

0 --
x-axis Planar joint with local 1 direction as the rotation axis.

1 --
z-axis Planar joint with local 3 direction as the rotation axis.

MPC184 Planar Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Planar Joint Element Output Definitions"* and *Table 2, "MPC184 Planar Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Planar Joint Element Output Definitions

Name	Definition	O	R
x-axis Planar Joint Element (KEYOPT(4) = 0)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST2	Constraint stop status on DOF 2[*1*]	-	Y
CLST2	Constraint lock status on DOF 2[*2*]	-	Y
CSST3	Constraint stop status on DOF 3[*1*]	-	Y
CLST3	Constraint lock status on DOF 3[*2*]	-	Y
CSST4	Constraint stop status on DOF 4[*1*]	-	Y
CLST4	Constraint lock status on DOF 4[*2*]	-	Y
JRP2	Joint relative position of DOF 2	-	Y
JRP3	Joint relative position of DOF 3	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JCD2	Joint constitutive displacement on DOF 2	-	Y
JCD3	Joint constitutive displacement on DOF 3	-	Y
JCD4	Joint constitutive rotation on DOF 4	-	Y
JEF2	Joint elastic force in direction -2	-	Y

Name	Definition	O	R
JEF3	Joint elastic force in direction -3	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JDF2	Joint damping force in direction -2	-	Y
JDF3	Joint damping force in direction -3	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JFF2	Joint friction force in direction -2	-	Y
JFF3	Joint friction force in direction -3	-	Y
JFF4	Joint friction moment in direction -4	-	Y
JRU2	Joint relative displacement of DOF 2	-	Y
JRU3	Joint relative displacement of DOF 3	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRV2	Joint relative velocity of DOF 2	-	Y
JRV3	Joint relative velocity of DOF 3	-	Y
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRA2	Joint relative acceleration of DOF 2	-	Y
JRA3	Joint relative acceleration of DOF 3	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y
z-axis Planar Joint Element (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FZ	Constraint Force in Z direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST1	Constraint stop status on DOF 1[*1*]	-	Y
CLST1	Constraint lock status on DOF 1[*2*]	-	Y
CSST2	Constraint stop status on DOF 2[*1*]	-	Y
CLST2	Constraint lock status on DOF 2[*2*]	-	Y
CSST6	Constraint stop status on DOF 6[*1*]	-	Y
CLST6	Constraint lock status on DOF 6[*2*]	-	Y
JRP1	Joint relative position of DOF 1	-	Y
JRP2	Joint relative position of DOF 2	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JCD1	Joint constitutive displacement on DOF 1	-	Y
JCD2	Joint constitutive displacement on DOF 2	-	Y
JCD6	Joint constitutive rotation on DOF 6	-	Y
JEF1	Joint elastic force in direction -1	-	Y
JEF2	Joint elastic force in direction -2	-	Y

Name	Definition	O	R
JEF6	Joint elastic moment in direction -6	-	Y
JDF1	Joint damping force in direction -1	-	Y
JDF2	Joint damping force in direction -2	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JFF1	Joint friction force in direction -1	-	Y
JFF2	Joint friction force in direction -2	-	Y
JFF6	Joint friction moment in direction -6	-	Y
JRU1	Joint relative displacement of DOF 1	-	Y
JRU2	Joint relative displacement of DOF 2	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV1	Joint relative velocity of DOF 1	-	Y
JRV2	Joint relative velocity of DOF 2	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA1	Joint relative acceleration of DOF 1	-	Y
JRA2	Joint relative acceleration of DOF 2	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the planar joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Planar Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all planar joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y

Name	Definition	O	R
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Planar Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Planar Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 Planar Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Planar Joint Element (KEYOPT(4) = 0)		
FX	SMISC	1
MY	SMISC	5
MZ	SMISC	6
CSTOP2	SMISC	8
CSTOP3	SMISC	9
CSTOP4	SMISC	10
CLOCK2	SMISC	14
CLOCK3	SMISC	15
CLOCK4	SMISC	16
CSST2	SMISC	20
CSST3	SMISC	21
CSST4	SMISC	22
CLST2	SMISC	26
CLST3	SMISC	27
CLST4	SMISC	28
JRP2	SMISC	32
JRP3	SMISC	33
JRP4	SMISC	34
JCD2	SMISC	38
JCD3	SMISC	39
JCD4	SMISC	40

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JEF2	SMISC	44
JEF3	SMISC	45
JEF4	SMISC	46
JDF2	SMISC	50
JDF3	SMISC	51
JDF4	SMISC	52
JFF2	SMISC	56
JFF3	SMISC	57
JFF4	SMISC	58
JRU2	SMISC	62
JRU3	SMISC	63
JRU4	SMISC	64
JRV2	SMISC	68
JRV3	SMISC	69
JRV4	SMISC	70
JRA2	SMISC	74
JRA3	SMISC	75
JRA4	SMISC	76
JTEMP	SMISC	79
z-axis Planar Joint Element (KEYOPT(4) = 1)		
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
CSTOP1	SMISC	7
CSTOP2	SMISC	8
CSTOP6	SMISC	12
CLOCK1	SMISC	13
CLOCK2	SMISC	14
CLOCK6	SMISC	18
CSST1	SMISC	19
CSST2	SMISC	20
CSST6	SMISC	24
CLST1	SMISC	25
CLST2	SMISC	26
CLST6	SMISC	30
JRP1	SMISC	31
JRP2	SMISC	32
JRP6	SMISC	36
JCD1	SMISC	37
JCD2	SMISC	38
JCD6	SMISC	42
JEF1	SMISC	43
JEF2	SMISC	44

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JEF6	SMISC	48
JDF1	SMISC	49
JDF2	SMISC	50
JDF6	SMISC	54
JFF1	SMISC	55
JFF2	SMISC	56
JFF6	SMISC	60
JRU1	SMISC	61
JRU2	SMISC	62
JRU6	SMISC	66
JRV1	SMISC	67
JRV2	SMISC	68
JRV6	SMISC	72
JRA1	SMISC	73
JRA2	SMISC	74
JRA6	SMISC	78
JTEMP	SMISC	79

Table 4 MPC184 Planar Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all planar joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Planar Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the planar joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the planar joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Planar Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Weld

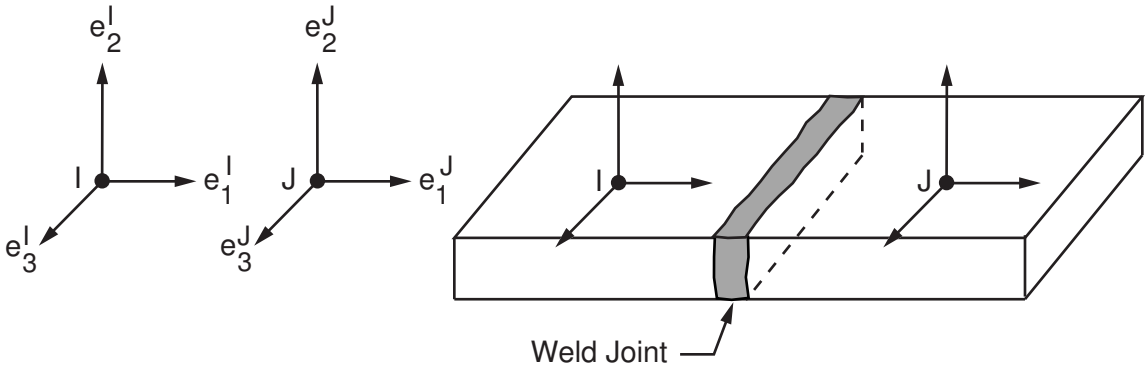
Multipoint Constraint Element: Weld Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Weld Joint Element Description

The MPC184 weld joint element is a two-node element that has all relative degrees of freedom fixed.

Figure 1 MPC184 Weld Joint Geometry



MPC184 Weld Joint Input Data

Set KEYOPT(1) = 13 to define a two-node weld joint element.

Figure 1, "MPC184 Weld Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

Other input data that are common to all joint elements (material behavior, etc.) are described in *Joint Input Data* in the MPC184 element description.

Note

The weld joint may also be simulated by using the **CE** command. See the **CE** command description for additional details.

MPC184 Weld Joint Input Summary

This input summary applies to the weld joint element option of MPC184: KEYOPT(1) = 13.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

13 --
Weld joint element

MPC184 Weld Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1,"MPC184 Weld Joint Element Output Definitions"* and *Table 2, "MPC184 Weld Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Weld Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y

Name	Definition	O	R
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the weld joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Weld Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Weld Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Weld Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
: output quantity as defined in the Element Output Definitions table.

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 MPC184 Weld Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MZ	SMISC	6

Table 4 MPC184 Weld Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Weld Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the weld joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the weld joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.

- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Weld Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-Orient

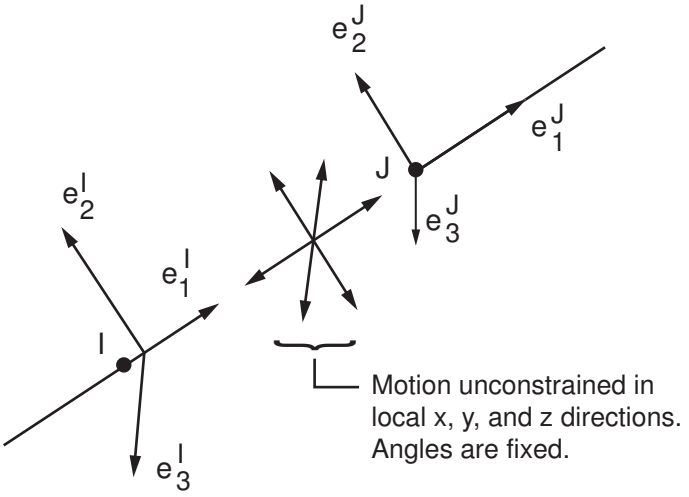
Multipoint Constraint Element: Orient Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 Orient Joint Element Description

The MPC184 orient joint is a two-node element. In this joint, the relative rotational degrees of freedom are fixed while the displacement degrees of freedom are left free.

Figure 1 MPC184 Orient Joint Geometry



MPC184 Orient Joint Input Data

Set KEYOPT(1) = 14 to define a two-node orient joint element.

Figure 1, "MPC184 Orient Joint Geometry" shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on an orient joint element are easily described by referring to *Figure 1, "MPC184 Orient Joint Geometry"*. At any given instant of time, the constraints imposed are as follows:

$$\mathbf{e}_2^I \cdot \mathbf{e}_3^J - \mathbf{E}_2^I \cdot \mathbf{E}_3^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - \mathrm{E}_1^I \cdot \mathrm{E}_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathrm{E}_1^I \cdot \mathrm{E}_3^J = 0$$

Other input data that are common to all joint elements (material behavior, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 Orient Joint Input Summary

This input summary applies to the orient joint element option of MPC184: KEYOPT(1) = 14.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection

KEYOPT(1)
Element behavior:

14 --
Orient joint element

MPC184 Orient Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 Orient Joint Element Output Definitions"* and *Table 2, "MPC184 Orient Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 Orient Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the orient joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 Orient Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 Orient Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 Orient Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The table uses the following notation:

Name
: output quantity as defined in the Element Output Definitions table.

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 3 MPC184 Orient Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6

Table 4 MPC184 Orient Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Orient Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the orient joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the orient joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.

- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.
- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 Orient Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

MPC184-General

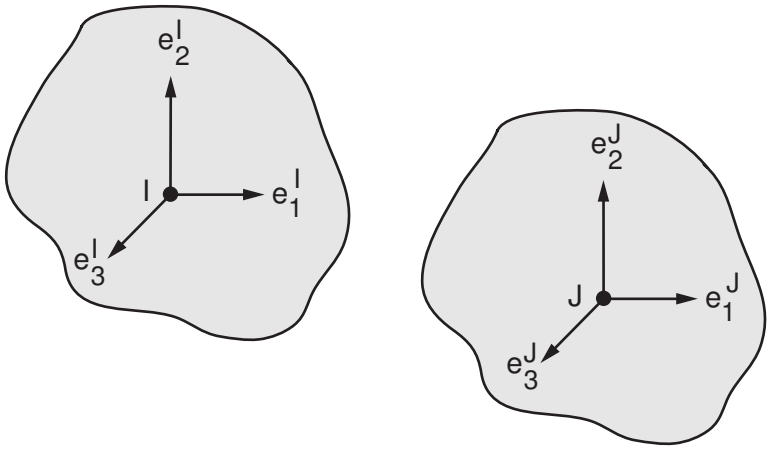
Multipoint Constraint Element: General Joint

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

MPC184 General Joint Element Description

The MPC184 general joint is a two-node element. By default, no relative degrees of freedom are fixed. However, you can specify which relative degrees of freedom need to be constrained. By specifying as many relative degrees of freedom to be constrained as needed, you can simulate different joint elements.

Figure 1 MPC184 General Joint Geometry



By default, a general joint has both displacement and rotational degrees of freedom activated at the nodes of the element. In some cases only displacement degrees of freedom are needed (as in a model consisting of only continuum elements). In such cases, you can specify a general joint with only displacement degrees of freedom activated by setting KEYOPT(4) = 1.

MPC184 General Joint Input Data

Set KEYOPT(1) = 16 to define a two-node general joint element.

Use KEYOPT(4) to specify the active degree-of-freedom set:

KEYOPT(4) = 0 (default) - both displacement and rotational degrees of freedom are activated.
KEYOPT(4) = 1 - only displacement degrees of freedom are activated.

For this element, you can specify which relative degrees of freedom need to be constrained. First, define the section type (**SECTYPE** command) for this joint. Then define the **SECJOINT** command as follows:

```
SECJ,LSYS,local cs1,local cs2  ! Defines the local coordinate systems for the joints.
SECJ,RDOF,dof1,dof2,…,dof6   ! Defines the relative DOFs to be constrained.
```

Note that the **SECJOINT** command is issued twice when the general joint is used with some degrees of freedom constrained. The first **SECJOINT** command defines the local coordinate systems for the joint. In the second **SECJOINT** command, specify as many relative degrees of freedom as needed (a maximum of 6 may be specified). The general joint element allows you to simulate different joint elements by specifying different relative degrees of freedom to be constrained. The following examples highlight the different joint elements that can be simulated:

```
SECJ,RDOF,dof1,dof2,dof3,dof5,dof6  ! Simulates a revolute joint with local e_1 axis as the axis of revolute.
```

```
SECJ,RDOF,dof2,dof3,dof5,dof6   ! Simulates a cylindrical joint with local e_1 axis as the axis of rotation.

SECJ,RDOF,dof1,dof2,dof3,dof4,dof5,dof6   ! Simulates a weld joint.
```

When KEYOPT(4) = 1, the local coordinate systems specified at nodes I and J remain fixed in their initial orientation. The rotation at the nodes, if any, is ignored.

When KEYOPT(4) = 0, the local coordinate systems specified at nodes I and J are assumed to evolve with the rotations at the nodes.

For an unconstrained general joint (KEYOPT(4) = 0 or 1), the relative displacements between nodes I and J are as follows:

$$u_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The relative rotations between nodes I and J are characterized by the Cardan (or Bryant) angles as follows (only if KEYOPT(4) = 0):

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\varphi = -\sin^{-1}(e_1^I \cdot e_3^J)$$

$$\chi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

The relative displacements and rotations are suitably constrained when some or all of the relative degrees of freedom are fixed.

Note that the relative angular positions for the general joint are characterized by the Cardan (or Bryant) angles. This requires that the rotations about the local e_2 axis be restricted between –PI/2 to +PI/2. Thus, the local e_2 axis should not be used to simulate the axis of rotation.

For an unconstrained general joint, the constitutive calculations use the following definitions for relative displacement:

$$\mathbf{u}_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

$$\mathbf{u}_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

$$\mathbf{u}_3 = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_3^{ref}$$

where:

ℓ_1^{ref}, ℓ_2^{ref}, and ℓ_3^{ref} = reference lengths, length1, length2, and length3, specified on the **SECDATA** command.

The following definitions are for relative rotations:

$$u_{r4} = \phi - \phi_1^{ref}$$

$$u_{r5} = \psi - \phi_2^{ref}$$

$$u_{r6} = \chi - \phi_3^{ref}$$

where:

ϕ_1^{ref}, ϕ_3^{ref}, and ϕ_2^{ref} = reference angle specifications, angle1, angle2, and angle3 on the **SECDATA** command.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in *Joint Input Data* in the MPC184 element description.

MPC184 General Joint Input Summary

This input summary applies to the general joint element option of MPC184: KEYOPT(1) = 16.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ (KEYOPT(4) = 0)

UX, UY, UZ (KEYOPT(4) = 1)

Real Constants
None

Material Properties
Use the JOIN label on the **TB** command to define stiffness, damping, and hysteretic friction behavior. (See *Section 2.5.16: MPC184 Joint Materials* for detailed information on defining joint materials.)

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
For an unconstrained general joint:

Displacement (KEYOPT(4) = 0 or 1) --
UX, UY, UZ

Rotation (KEYOPT(4) = 0) --
ROTX, ROTY, ROTZ

For a constrained general joint, loads are based on the free relative degrees of freedom in the joint.

Special Features

Large deflection

KEYOPT(1)
Element behavior:

16 --
General joint element

KEYOPT(4)
Element configuration:

0 --
General joint with both displacement and rotational degrees of freedom activated.

1 --
General joint with only displacement degrees of freedom activated.

MPC184 General Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "MPC184 General Joint Element Output Definitions"* and *Table 2, "MPC184 General Joint Element - NMISC Output "*.

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 MPC184 General Joint Element Output Definitions

Name	Definition	O	R
General joint with displacement and rotation DOF (KEYOPT(4) = 0)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
The constraint force and moment output depends on which of the relative DOFs are constrained.			
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y

Name	Definition	O	R
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
The following output depends on which of the relative DOFs are unconstrained.			
CSTOP1-6	Constraint force/moment if stop is specified on DOFs 1-6	-	Y
CLOCK1-6	Constraint force/moment if lock is specified on DOFs 1-6	-	Y
CSST1-6	Constraint stop status on relative DOFs 1-6[*1*]	-	Y
CLST1-6	Constraint lock status on relative DOFs 1-6[*2*]	-	Y
JRP1-6	Joint relative position of DOFs 1-6	-	Y
JCD1-6	Joint constitutive displacement/rotation of DOFs 1-6	-	Y
JEF1-6	Joint elastic force/moment 1-6	-	Y
JDF1-6	Joint damping force/moment 1-6	-	Y
JFF1-6	Joint friction force/moment 1-6	-	Y
JRU1-6	Joint relative displacement/rotation 1-6	-	Y
JRV1-6	Joint relative velocity (or rotational velocity) 1-6	-	Y
JRA1-6	Joint relative acceleration (or rotational acceleration) 1-6	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y
General joint with displacement DOF (KEYOPT(4) =1)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
The constraint force and moment output depends on which of the relative DOFs are constrained.			
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
The following output depends on which of the relative DOFs are unconstrained.			
CSTOP1-3	Constraint force if stop is specified on DOFs 1-3	-	Y
CLOCK1-3	Constraint force if lock is specified on DOFs 1-3	-	Y
CSST1-3	Constraint stop status on relative DOFs 1-3[*1*]	-	Y
CLST1-3	Constraint lock status on relative DOFs 1-3[*2*]	-	Y
JRP1-3	Joint relative position of DOFs 1-3	-	Y
JCD1-3	Joint constitutive displacement of DOFs 1-3	-	Y
JEF1-3	Joint elastic force 1-3	-	Y
JDF1-3	Joint damping force 1-3	-	Y
JFF1-3	Joint friction force 1-3	-	Y
JRU1-3	Joint relative displacement 1-3	-	Y
JRV1-3	Joint relative velocity 1-3	-	Y
JRA1-3	Joint relative acceleration 1-3	-	Y
JTEMP	Average temperature in the element[*3*]	-	Y

1. Constraint stop status:

0 = stop not active, or deactivated
1 = stopped at minimum value
2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the general joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 2 MPC184 General Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all general joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved coordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 3, "MPC184 General Joint Item and Sequence Numbers - SMISC Items" and *Table 4, "MPC184 General Joint Item and Sequence Numbers - NMISC Items"* list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* for further information. The tables use the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 3 MPC184 General Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
General joint with displacement and rotation DOF (KEYOPT(4) = 0) (Some of these values may be zero depending on which relative degrees of freedom are constrained.)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6
CSTOP1-6	SMISC	7-12
CLOCK1-6	SMISC	13-18
CSST1-6	SMISC	19-24
CLST1-6	SMISC	25-30
JRP1-6	SMISC	31-36
JCD1-6	SMISC	37-42
JEF1-6	SMISC	43-48
JDF1-6	SMISC	49-54
JFF1-6	SMISC	55-60
JRU1-6	SMISC	61-66
JRV1-6	SMISC	67-72
JRA1-6	SMISC	73-78
JTEMP	SMISC	79
General joint with displacement DOF (KEYOPT(4) = 1) (Some of these values may be zero depending on which relative degrees of freedom are constrained.)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
CSTOP1-3	SMISC	7-9
CLOCK1-3	SMISC	13-15
CSST1-3	SMISC	19–21
CLST1-3	SMISC	25-27
JRP1-3	SMISC	31-33
JCD1-3	SMISC	37-39
JEF1-3	SMISC	43-45
JDF1-3	SMISC	49-51
JFF1-3	SMISC	55-57
JRU1-3	SMISC	61-63
JRV1-3	SMISC	67-69
JRA1-3	SMISC	73-78
JTEMP	SMISC	79

Table 4 MPC184 General Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all general joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 General Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the general joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the general joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

- This element cannot be used with the arc-length method (**ARCLEN**).

MPC184 General Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

SOLID185

3-D 8-Node Structural Solid or Layered Solid

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SOLID185 Element Description

SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID185 is available in two forms:

- Structural Solid (KEYOPT(3) = 0, the default) -- See *SOLID185 Structural Solid Element Description*.
- Layered Solid (KEYOPT(3) = 1) -- See *SOLID185 Layered Solid Element Description*.

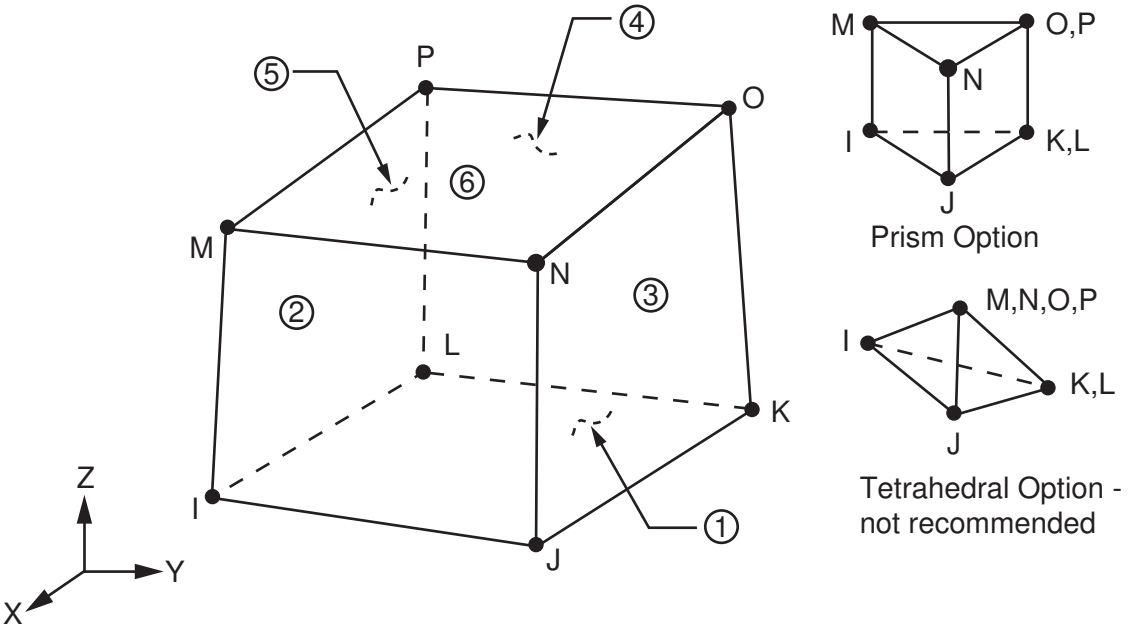
See SOLID185 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

A higher-order version of the SOLID185 element is SOLID186.

SOLID185 Structural Solid Element Description

SOLID185 Structural Solid is suitable for modeling general 3-D solid structures. It allows for prism and tetrahedral degenerations when used in irregular regions. Various element technologies such as B-bar, uniformly reduced integration, and enhanced strains are supported.

Figure 1 SOLID185 Structural Solid Geometry



SOLID185 Structural Solid Input Data

The geometry and node locations for this element are shown in *Figure 1, "SOLID185 Structural Solid Geometry"*. The element is defined by eight nodes and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "SOLID185 Structural Solid Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

As described in *Section 2.3: Coordinate Systems*, you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetrical matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

SOLID185 Structural Solid Input Summary contains a summary of element input. For a general description of element input, see *Section 2.1: Element Input*.

SOLID185 Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

None, if KEYOPT(2) = 0,
HGSTF - Hourglass Stiffness Scaling factor if KEYOPT(2) = 1 (Default is 1.0; any positive number is valid. If set to 0.0, value is automatically reset to 1.0.)

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(2)
Element technology:

0 --

Full integration with $\bar{B}$ method

1 --

Uniform reduced integration with hourglass control

2 --

Enhanced strain formulation

3 --

Simplified enhanced strain formulation

KEYOPT(3)
Layer construction:

0 --

Structural Solid (default) -- nonlayered

1 --
Layered Solid (not applicable to SOLID185 Structural Solid)

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses (default).

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

SOLID185 Structural Solid Element Technology

SOLID185 uses the $\bar{B}$ method (also known as the selective reduced integration method), the uniform reduced integration method, or the enhanced strain formulation method, as follows:

- **$\bar{B}$ method (selective reduced integration)**

  Helps to prevent volumetric mesh locking in nearly incompressible cases. This option replaces volumetric strain at the Gauss integration point with the average volumetric strain of the elements. This method cannot, however, prevent any shear locking in bending dominated problems. In such situations, use the enhanced strain formulation of this element. If it is not clear if the deformation is bending dominated, enhanced strain formulation is recommended. For more information, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Uniform reduced integration**

  Also helps to prevent volumetric mesh locking in nearly incompressible cases. Because it has only one integration point, this option is more efficient than the $\bar{B}$ method (selective reduced integration) option. However, the artificial energy introduced to control the hourglass effect may affect solution accuracy adversely.

  When using this option, check the solution accuracy by comparing the total energy (SENE label in **ETABLE**) and the artificial energy (AENE label in **ETABLE**) introduced by hourglass control. If the ratio of artificial energy to total energy is less than 5%, the solution is generally acceptable. If the ratio exceeds five percent, refine the mesh. You can also monitor the total energy and artificial energy by issuing the **OUTPR**,VENG command in the solution phase.

  For more information about uniform reduced integration, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Enhanced strain formulation**

Prevents shear locking in bending-dominated problems and volumetric locking in nearly incompressible cases. The formulation introduces 13 *internal* DOFs (inaccessible to ANSYS users). If mixed u-P formulation is employed with enhanced strain formulation, only 9 DOFs for overcoming shear-locking are used. All internal DOFs are introduced automatically at the element level and condensed out.

Because of the extra internal DOFs and static condensation, this option is less efficient than either the $\bar{B}$ method (selective reduced integration) option or the uniform reduced integration option.

For more information about enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Simplified enhanced strain formulation**

  Prevents shear locking in bending-dominated problems. This is a special case of the enhanced strain formulation and always introduces 9 *internal* DOFs (inaccessible to ANSYS users). Because there are no internal DOFs to handle volumetric locking, this formulation should not be used when the material is nearly incompressible, except when the Mixed u-P formulation is also used. When used with the Mixed u-P formulation, the simplified enhanced strain formulation gives the same results as the enhanced strain formulation. All internal DOFs are introduced automatically at the element level and condensed out.

  Because of the extra internal DOFs and static condensation, this option is less efficient than either the $\bar{B}$ method (selective reduced integration) option or the uniform reduced integration option, but is more efficient than the enhanced strain formulation due to using fewer internal DOFs.

  For more information about the simplified enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

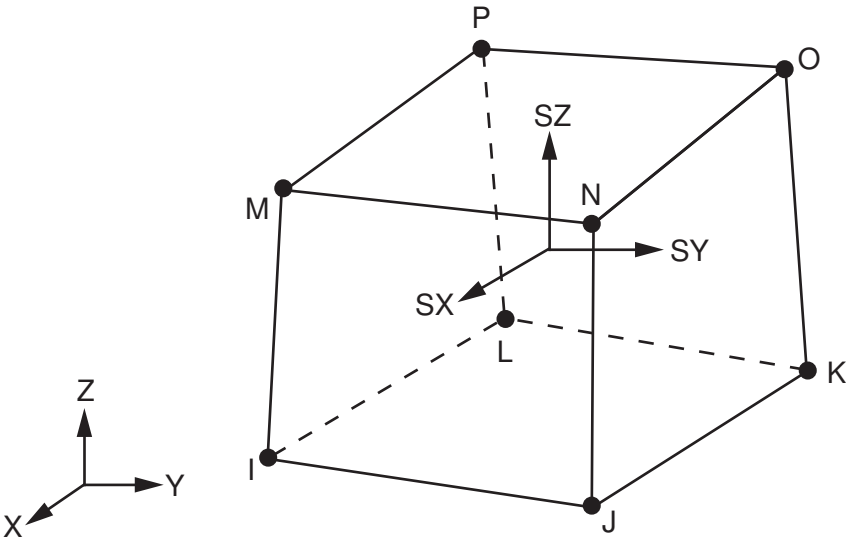
SOLID185 Structural Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID185 Structural Solid Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID185 Structural Solid Stress Output"*. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information.

Figure 2 SOLID185 Structural Solid Stress Output



Stress directions shown are for global directions.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID185 Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [*6*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*2*	*2*
EPTH:EQV	Equivalent thermal strains [*6*]	*2*	*2*

Name	Definition	O	R
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*

1. Nonlinear solution, output only if the element has a nonlinear material
2. Output only if element has a thermal load
3. Available only at centroid as a ***GET** item
4. Available only if **OUTRES**,LOCI is used
5. Available only if the USERMAT subroutine and **TB**,STATE are used
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 2,"SOLID185 Structural Solid Item and Sequence Numbers" lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information. The following notation is used in *Table 2, "SOLID185 Structural Solid Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SOLID185 Structural Solid Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

I,J,...,P
: sequence number for data at nodes I, J, ..., P

Table 2 SOLID185 Structural Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24

SOLID185 Structural Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1, "SOLID185 Structural Solid Geometry"* or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See *Section 2.9: Triangle, Prism and Tetrahedral Elements*.) A tetrahedron shape is also available.
- For the degenerated shape elements where the $\bar{B}$ or enhanced strain formulations are specified, degenerated shape functions and a conventional integration scheme are used.
- If you use the mixed formulation (KEYOPT(6) = 1), you must use either the sparse solver (default) or the frontal solver.
- For modal cyclic symmetry analyses, ANSYS recommends using enhanced strain formulation.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

This element has a layered option (KEYOPT(3) = 1). See *SOLID185 Layered Solid Assumptions and Restrictions* for additional information.

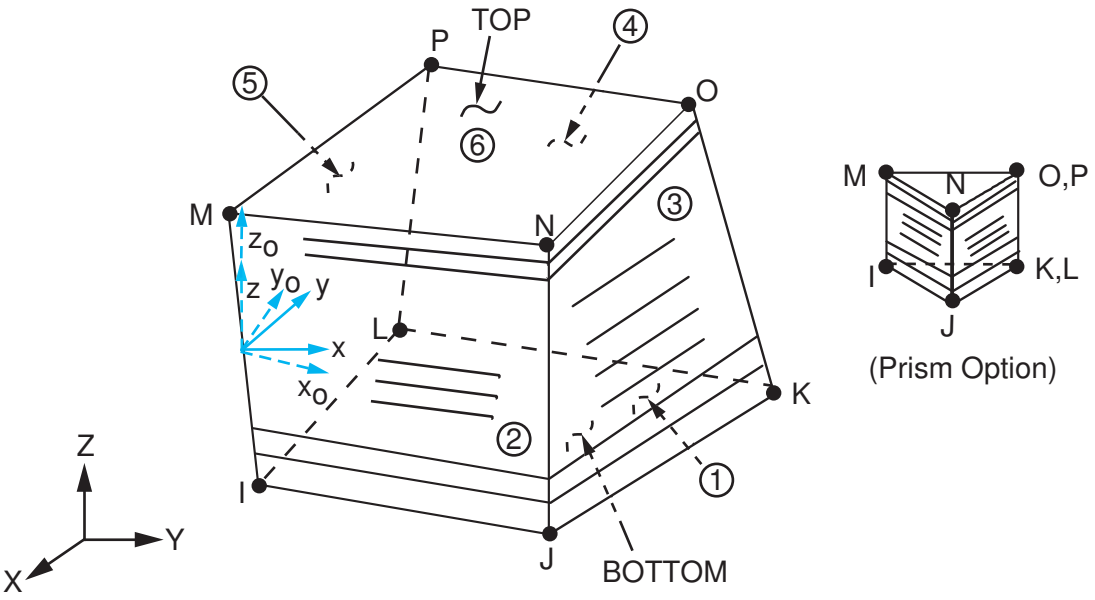
SOLID185 Structural Solid Product Restrictions

None.

SOLID185 Layered Solid Element Description

Use SOLID185 Layered Solid to model layered thick shells or solids. The element allows up to 250 different material layers. The element may be stacked for modeling composites with more than 250 layers or for improving solution accuracy. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Figure 3 SOLID185 Layered Solid Geometry



x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID185 Layered Solid Input Data

The geometry and node locations for this element are shown in *Figure 3, "SOLID185 Layered Solid Geometry"*. The element is defined by eight nodes. A prism-shaped element may be formed by defining the same node numbers for nodes K and L, and O and P.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command in the same way that you would for shell elements (as described in *Section 2.3: Coordinate Systems*). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID185 Layered Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness.

When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID185 Layered Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element.

The default first surface direction S1 can be reoriented in the element reference plane (as shown in *Figure 3, "SOLID185 Layered Solid Geometry"*) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See *Section 2.3: Coordinate Systems* for details.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 3, "SOLID185 Layered Solid Geometry"*. Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, if you define temperatures with commands other than **BFE**--SOLID185 Layered Solid adopts an element-wise temperature pattern and requires only eight temperatures for the eight element corner nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). `For any other input temperature pattern, unspecified nodal temperatures default to TUNIF. ANSYS computes all layer interface temperatures by interpolating nodal temperatures.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-1024 maximum). In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NLayer. If you input exactly NLayer+1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **ISTRESS** or **ISFILE** command. For more information, see *Section 2.5.13: Initial State Loading* in the *Basic Analysis Guide*. Alternatively, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details about user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetrical matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. *Section 2.1: Element Input* provides a general description of element input.

SOLID185 Layered Solid Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants

None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NLayer (4 * (NLayer + 1) maximum)

Special Features

Plasticity
Hyperelasticity
Viscoelasticity
Viscoplasticity
Creep
Stress stiffening
Large deflection
Large strain
Initial stress import
Automatic selection of element technology
Birth and death

Supports the following types of data tables associated with the **TB** command: AHYPER, ANEL, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, RATE, CREEP, HYPER, PRONY, SHIFT, CAST, SMA, ELASTIC, SDAMP, PLASTIC, and USER.

Note

See the *Theory Reference for ANSYS and ANSYS Workbench* for details about material models.

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information about selecting element technologies.

KEYOPT(2)
Element technology:

2 --
Enhanced strain formulation

3 --
Simplified enhanced strain formulation (default)

KEYOPT(3)
Layer construction:

0 --
Structural Solid (not applicable to SOLID185 Layered Solid)

1 --
Layered Solid

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

KEYOPT(8)
Layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (default)

1 --
Store top and bottom data for all layers

Note

The amount of data involved can be very large when KEYOPT(8) = 1.

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

SOLID185 Layered Solid Element Technology

SOLID185 Layered Solid uses the enhanced strain formulation method, as follows:

- **Enhanced strain formulation**

  Prevents shear locking in bending-dominated problems and volumetric locking in nearly incompressible cases. The formulation introduces 13 internal DOFs (inaccessible to ANSYS users). If mixed u-P formulation is employed with enhanced strain formulation, only nine DOFs for overcoming shear-locking are used. All internal DOFs are introduced automatically at the element level and condensed out.

  For more information about enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

- **Simplified enhanced strain formulation**

Prevents shear locking in bending-dominated problems. This is a special case of the enhanced strain formulation and always introduces 9 *internal* DOFs (inaccessible to ANSYS users). Because there are no internal DOFs to handle volumetric locking, this formulation should not be used when the material is nearly incompressible, except when the Mixed u-P formulation is also used. When used with the Mixed u-P formulation, the simplified enhanced strain formulation gives the same results as the enhanced strain formulation. All internal DOFs are introduced automatically at the element level and condensed out.

This option is more efficient than the enhanced strain formulation because it uses fewer internal DOFs.

For more information about the simplified enhanced strain formulation, see the *Theory Reference for ANSYS and ANSYS Workbench*.

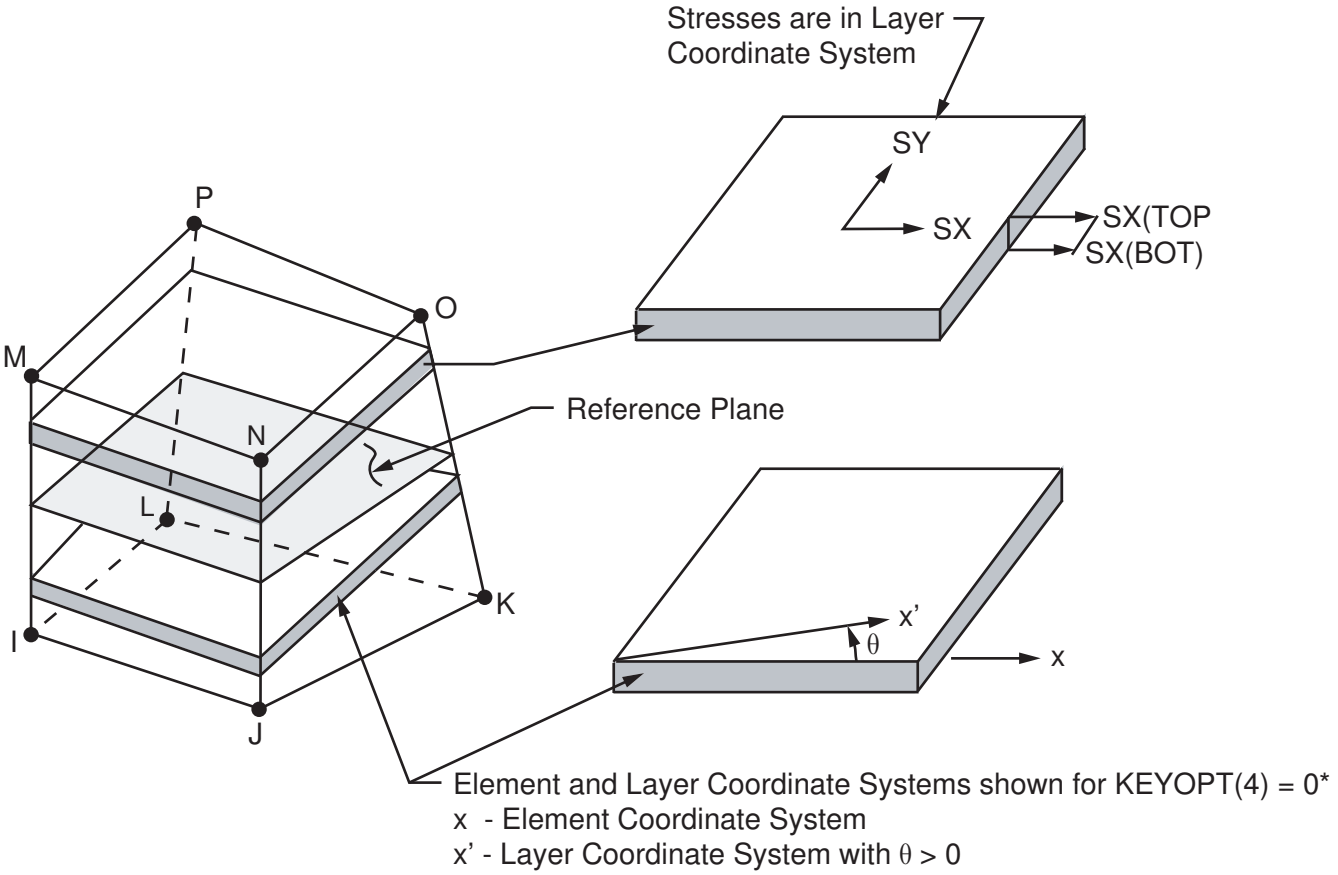
SOLID185 Layered Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 3, "SOLID185 Layered Solid Element Output Definitions"*

Several items are illustrated in *Figure 4, "SOLID185 Layered Solid Stress Output"*. See *Section 5.1.3.2: Filling the Element Table for Variables Identified By Sequence Number* in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information.

Figure 4 SOLID185 Layered Solid Stress Output



*Note: Layer Coordinate System x-y plane is parallel to the reference plane (KREF)

Stress directions shown are for global directions.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 SOLID185 Layered Solid Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y

Name	Definition	O	R
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [*6*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*2*	*2*
EPTH:EQV	Equivalent thermal strains [*6*]	*2*	*2*
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. Nonlinear solution, output only if the element has a nonlinear material
2. Output only if element has a thermal load
3. Available only at centroid as a ***GET** item
4. Available only if **OUTRES**,LOCI is used
5. Available only if the USERMAT subroutine and **TB**,STATE are used
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 4, "SOLID185 Layered Solid Item and Sequence Numbers" lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and

Section 2.2.2.2: The Item and Sequence Number Table in this document for more information. The following notation is used in *Table 4, "SOLID185 Layered Solid Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 3, "SOLID185 Layered Solid Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 4 SOLID185 Layered Solid Item and Sequence Numbers

Output Quantity Name		ETABLE and ESOL Command Input								
		Item	I	J	K	L	M	N	O	P
P1		SMISC	2	1	4	3	-	-	-	-
P2		SMISC	5	6	-	-	8	7	-	-
P3		SMISC	-	9	10	-	-	12	11	-
P4		SMISC	-	-	13	14	-	-	16	15
P5		SMISC	18	-	-	17	19	-	-	20
P6		SMISC	-	-	-	-	21	22	23	24

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 41	8 * (NL - 1) + 42
ILSYZ	SMISC	8 * (i - 1) + 43	8 * (NL - 1) + 44
ILSUM	SMISC	8 * (i - 1) + 45	8 * (NL - 1) + 46
ILANG	SMISC	8 * (i - 1) + 47	8 * (NL - 1) + 48

SOLID185 Layered Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 3, "SOLID185 Layered Solid Geometry"* or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See *Section 2.9: Triangle, Prism and Tetrahedral Elements*.) A tetrahedron shape is not allowed.
- If you use the mixed formulation (KEYOPT(6) = 1), you must use either the sparse solver (default) or the frontal solver.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated via the **PSTRES** command.
- The maximum number of layers is 250.
- If the material of a layer is hyperelastic, the layer orientation angle has no effect.

SOLID185 Layered Solid Product Restrictions

There are no product-specific restrictions for this element.

SOLID186

3-D 20-Node Structural Solid or Layered Solid

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SOLID186 Element Description

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID186 is available in two forms:

- Structural Solid (KEYOPT(3) = 0, the default) -- See *SOLID186 Structural Solid Element Description* .
- Layered Solid (KEYOPT(3) = 1) -- See *SOLID186 Layered Solid Element Description*.

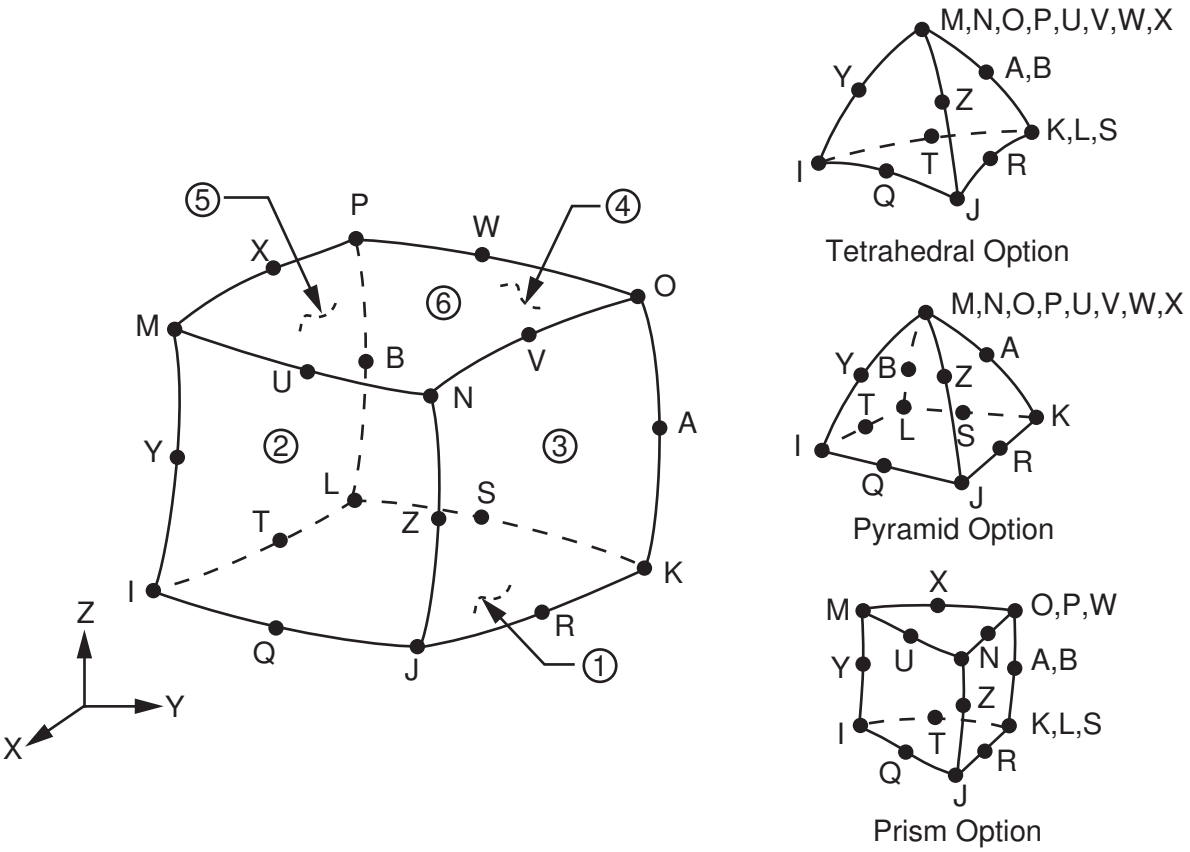
A lower-order version of the SOLID186 element is SOLID185.

SOLID186 Structural Solid Element Description

SOLID186 Structural Solid is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems). The element may have any spatial orientation.

Various printout options are available. See SOLID186 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details.

Figure 1 SOLID186 Structural Solid Geometry



SOLID186 Structural Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in *Figure 1, "SOLID186 Structural Solid Geometry"*. A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in *Figure 1, "SOLID186 Structural Solid Geometry"*. SOLID187 is a similar, but 10-node tetrahedron element.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID186 Structural Solid Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in *Section 2.3: Coordinate Systems*, you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. *Section 2.1: Element Input* provides a general description of element input.

SOLID186 Structural Solid Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX,THSY, THSZ),
PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X), T(Y), T(Z), T(A), T(B)

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(2)
Element technology:

0 --
Uniform reduced integration (default)

1 --
Full integration

KEYOPT(3)
Layer construction:

0 --
Structural Solid (default) -- nonlayered

1 --
Layered Solid (not applicable to SOLID186 Structural Solid)

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

SOLID186 Structural Solid Element Technology

SOLID186 uses the uniform reduced integration method or the full integration method, as follows:

- **Uniform reduced integration method**

  Helps to prevent volumetric mesh locking in nearly incompressible cases. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

- **Full integration**

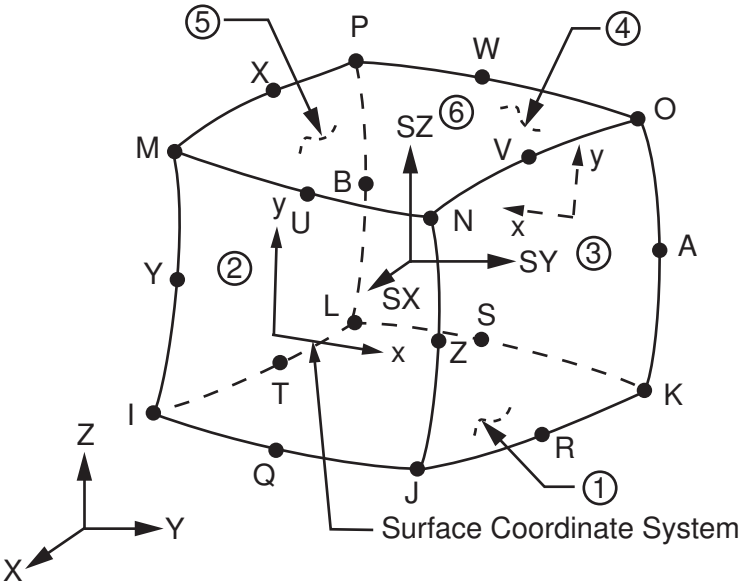
  The full integration method does not cause hourglass mode, but can cause volumetric locking in nearly incompressible cases. This method is used primarily for purely linear analyses, or when the model has only one layer of elements in each direction.

SOLID186 Structural Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID186 Structural Solid Element Output Definitions"*

Figure 2 SOLID186 Structural Solid Stress Output



The element stress directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID186 Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y

Name	Definition	O	R
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [*6*]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*2*	*2*
EPTH:EQV	Equivalent thermal strains [*6*]	*2*	*2*
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy density	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*

1. Nonlinear solution (output only if the element has a nonlinear material)
2. Output only if element has a thermal load
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the USERMAT subroutine and **TB**,STATE are used.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 2, "SOLID186 Structural Solid Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information. The following notation is used in *Table 2, "SOLID186 Structural Solid Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "SOLID186 Structural Solid Element Output Definitions"*

Item
: predetermined Item label for **ETABLE**

I,J,...,B
: sequence number for data at nodes I, J, ..., B

Table 2 SOLID186 Structural Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	J	K	L	M	N	O	P	Q,...,B
P1	SMISC	2	1	4	3	-	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-	-
P3	SMISC	-	9	10	-	-	12	11	-	-
P4	SMISC	-	-	13	14	-	-	16	15	-
P5	SMISC	18	-	-	17	19	-	-	20	-
P6	SMISC	-	-	-	-	21	22	23	24	-

See *Section 2.2.2.5: Surface Solution* in this document for the item and sequence numbers for surface output for **ETABLE**.

SOLID186 Structural Solid Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is not numbered properly). Elements may be numbered either as shown in *Figure 1, "SOLID186 Structural Solid Geometry"* or may have the planes IJKL and MNOP interchanged.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Use at least two elements in each direction to avoid hourglass mode if uniform reduced integration is used (KEYOPT(2) = 0).
- When degenerated into a tetrahedron, wedge, or pyramid element shape (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*), the corresponding degenerated shape functions are used. Degeneration to a pyramidal form should be used with caution. The element sizes, when degenerated, should be small to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- For mixed formulation (KEYOPT(6) = 1), no midside nodes can be missed, and no degenerated shapes are recommended. If you use the mixed formulation, you must use either the sparse solver (default) or the frontal solver.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated via the **PSTRES** command.

This element has a layered option (KEYOPT(3) = 1). See *SOLID186 Layered Solid Assumptions and Restrictions* for additional information.

SOLID186 Structural Solid Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

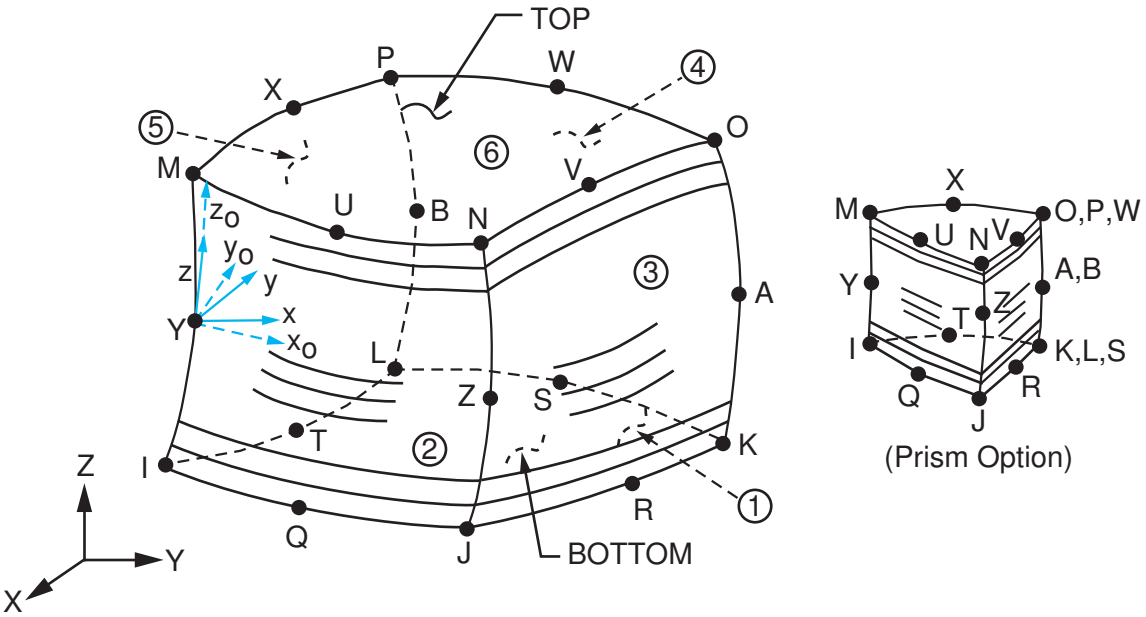
ANSYS Professional

- The only special feature allowed is stress stiffening.

SOLID186 Layered Solid Element Description

Use SOLID186 Layered Solid to model layered thick shells or solids. The element allows up to 250 different material layers. The element may be stacked for modeling composites with more than 250 layers or for improving solution accuracy. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Figure 3 SOLID186 Layered Solid Geometry



x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID186 Layered Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in *Figure 3, "SOLID186 Layered Solid Geometry"*. A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command in the same way that you would for shell elements (as described in *Section 2.3: Coordinate Systems*). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID186 Layered Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a

single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID186 Layered Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element.

The default first surface direction S1 can be reoriented in the element reference plane (as shown in *Figure 3, "SOLID186 Layered Solid Geometry"*) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See *Section 2.3: Coordinate Systems* for details.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 3, "SOLID186 Layered Solid Geometry"*. Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, if you define temperatures with commands other than **BFE**--SOLID186 Layered Solid adopts an element-wise temperature pattern and requires only eight temperatures for the eight element corner nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified nodal temperatures default to TUNIF. ANSYS computes all layer interface temperatures by interpolating nodal temperatures.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-1024 maximum). In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NLayer. If you input exactly NLayer+1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

As described in *Section 2.3: Coordinate Systems*, you can use the **ESYS** command to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than the material/element coordinate system.

KEYOPT(6) = 1 sets the element for using u-P mixed formulation. For details about the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **ISTRESS** or **ISFILE** command. For more information, see Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses

from the user subroutine USTRESS. For details about user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. *Section 2.1: Element Input* provides a general description of element input.

SOLID186 Layered Solid Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX,THSY, THSZ),
PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NLayer (4 * (NLayer + 1) maximum)

Special Features

Plasticity
Hyperelasticity
Viscoelasticity
Viscoplasticity
Creep
Stress stiffening
Large deflection
Large strain
Initial stress import
Automatic selection of element technology
Birth and death
Supports the following types of data tables associated with the **TB** command: ANEL, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, RATE, CREEP, HYPER, PRONY, SHIFT, CAST, SMA, ELASTIC, SDAMP, PLASTIC, and USER.

Note

See the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material models.

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(2)

Element technology:

0 --

Uniform reduced integration (default)

KEYOPT(3)

Layer construction:

0 --

Structural Solid (not applicable to SOLID186 Layered Solid)

1 --

Layered Solid

KEYOPT(6)

Element formulation:

0 --

Use pure displacement formulation (default)

1 --

Use mixed formulation

KEYOPT(8)

Layer data storage:

0 --

Store data for bottom of bottom layer and top of top layer

1 --

Store top and bottom data for all layers

Note

Be aware that the amount of data involved can be very large when KEYOPT(8) = 1.

KEYOPT(10)

User-defined initial stress:

0 --

No user subroutine to provide initial stresses (default)

1 --

Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user-written subroutines.

SOLID186 Layered Solid Element Technology

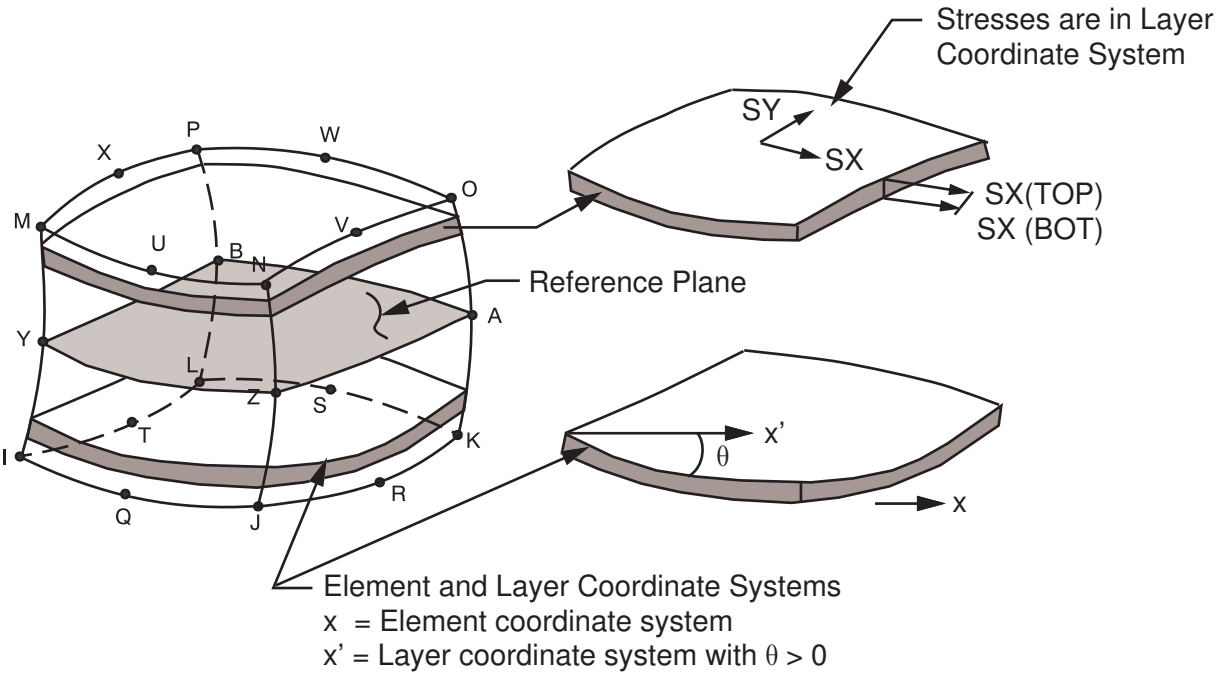
SOLID186 Layered Solid supports only the uniform reduced integration method (KEYOPT(2) = 0), which helps to prevent volumetric mesh locking in nearly incompressible cases. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

SOLID186 Layered Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 3, "SOLID186 Layered Solid Element Output Definitions"*

Figure 4 SOLID186 Layered Solid Stress Output



The element stress directions are parallel to the layer coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 3 SOLID186 Layered Solid Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y

Name	Definition	O	R
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [*6*]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*2*	*2*
EPTH:EQV	Equivalent thermal strains [*6*]	*2*	*2*
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy density	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. Nonlinear solution (output only if the element has a nonlinear material)
2. Output only if element has a thermal load
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the USERMAT subroutine and **TB**,STATE are used.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 4, "SOLID186 Layered Solid Item and Sequence Numbers" lists output available via **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item*

and Sequence Number Table in this document for more information. The following notation is used in *Table 4, "SOLID186 Layered Solid Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 3, "SOLID186 Layered Solid Element Output Definitions"*

Item
predetermined Item label for **ETABLE**

I,J,...,B
sequence number for data at nodes I, J, ..., B

Table 4 SOLID186 Layered Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	J	K	L	M	N	O	P	Q,...,B
P1	SMISC	2	1	4	3	-	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-	-
P3	SMISC	-	9	10	-	-	12	11	-	-
P4	SMISC	-	-	13	14	-	-	16	15	-
P5	SMISC	18	-	-	17	19	-	-	20	-
P6	SMISC	-	-	-	-	21	22	23	24	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 41	8 * (NL - 1) + 42
ILSYZ	SMISC	8 * (i - 1) + 43	8 * (NL - 1) + 44
ILSUM	SMISC	8 * (i - 1) + 45	8 * (NL - 1) + 46
ILANG	SMISC	8 * (i - 1) + 47	8 * (NL - 1) + 48

See *Section 2.2.2.5: Surface Solution* in this document for the item and sequence numbers for surface output for **ETABLE**.

SOLID186 Layered Solid Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is not numbered properly). Elements may be numbered either as shown in *Figure 3, "SOLID186 Layered Solid Geometry"* or may have the planes IJKL and MNOP interchanged.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Use at least two elements in each direction to avoid hourglass mode.
- When degenerated into a wedge element shape (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*), the corresponding degenerated shape functions are used. The element sizes, when degenerated, should be small to minimize the stress gradients.
- For mixed formulation (KEYOPT(6) = 1), no midside nodes can be missed, and no degenerated shapes are recommended. If you use the mixed formulation, you must use either the sparse solver (default) or the frontal solver.

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated via the **PSTRES** command.
- The maximum number of layers is 250.
- If the material of a layer is hyperelastic, the layer orientation angle has no effect.

SOLID186 Layered Solid Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is stress stiffening.

SOLID187

3-D 10-Node Tetrahedral Structural Solid

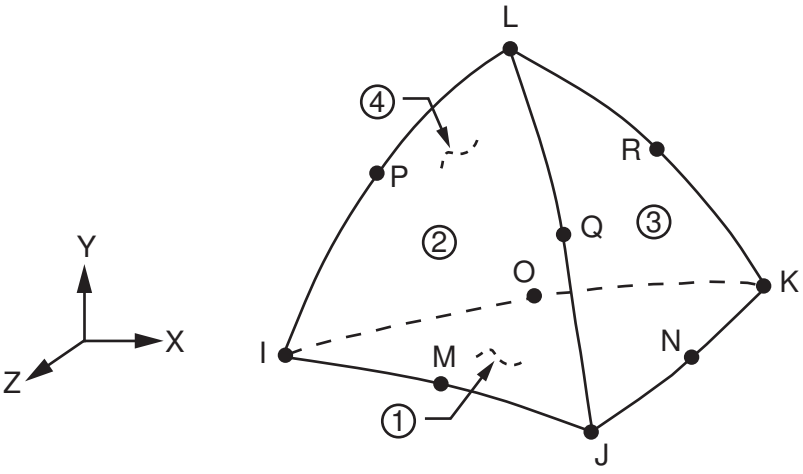
MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

SOLID187 Element Description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. See SOLID187 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID187 Geometry



SOLID187 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID187 Geometry"*.

In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.4: Linear Material Properties*.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLID187 Geometry"*. Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in *Section 2.3: Coordinate Systems*, you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(6) = 1 or 2 sets the element for using mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

SOLID187 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(6)

Element formulation:

0 --

Use pure displacement formulation (default)

1 --

Use mixed formulation, hydrostatic pressure is constant in an element (recommended for hyperelastic materials)

2 --

Use mixed formulation, hydrostatic pressure is interpolated linearly in an element (recommended for nearly incompressible elastoplastic materials)

KEYOPT(10)

User-defined initial stress:

0 --

No user subroutine to provide initial stresses (default).

1 --

Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)

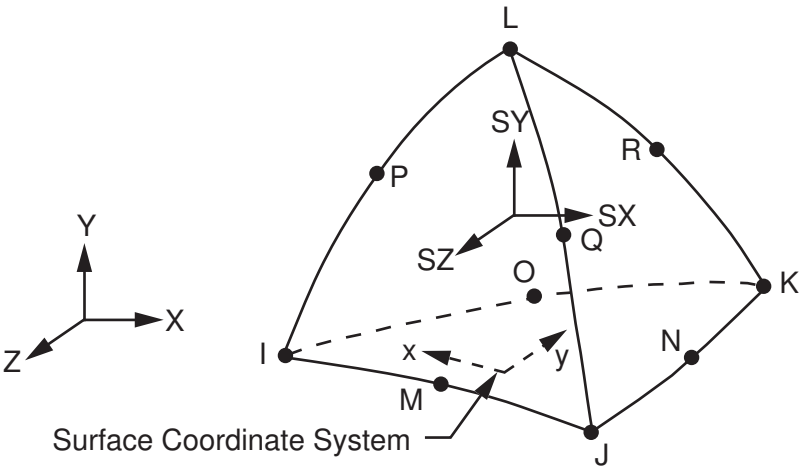
SOLID187 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID187 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID187 Stress Output"*. The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate system and are available for any face (KEYOPT(6)). The coordinate system for face JIK is shown in *Figure 2, "SOLID187 Stress Output"*. The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in *Section 2.2.2: Element Solution* are met. A general description of solution output is given in *Section 2.2.2.2: The Item and Sequence Number Table*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID187 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID187 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*3*
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [*6*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	*1*	*1*
EPTH: EQV	Equivalent thermal strains [*6*]	*1*	*1*
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [*7*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*6*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*6*]	*1*	*1*

Name	Definition	O	R
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND: ELASTIC, PLASTIC, CREEP	Strain energy density	-	*1*
LOCI:X, Y, Z	Integration point locations	-	*4*
SVAR:1, 2, ... , N	State variables	-	*5*

1. Nonlinear solution, output only if the element has a nonlinear material
2. Output only if element has a thermal load
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the USERMAT subroutine and **TB**,STATE are used.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 2, "SOLID187 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID187 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "SOLID187 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,R
sequence number for data at nodes I, J, ..., R

Table 2 SOLID187 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-

See *Section 2.2.2.5: Surface Solution* in this manual for the item and sequence numbers for surface output for **ETABLE**.

SOLID187 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in *Figure 1, "SOLID187 Geometry"* or may have node L below the I, J, K plane.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for information about using midside nodes.
- When mixed formulation is used (KEYOPT(6) = 1 or 2), no midside nodes can be missed.
- If you use the mixed formulation (KEYOPT(6) = 1 or 2), you must use either the sparse solver (default) or the frontal solver.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

SOLID187 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is stress stiffening.

BEAM188

3-D Linear Finite Strain Beam

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

BEAM188 Element Description

BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.

BEAM188 is a linear (2-node) or a quadratic beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node, with the number of degrees of freedom depending on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

BEAM188 includes stress stiffness terms, by default, in any analysis with **NLGEOM**,ON. The provided stress stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling or collapse studies with arc length methods).

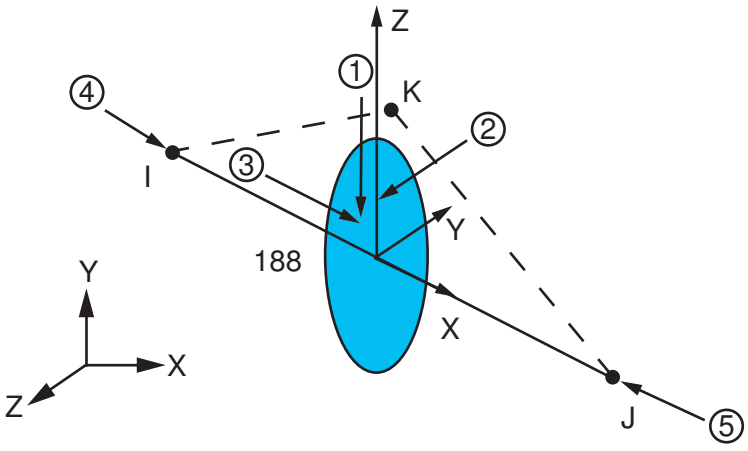
BEAM188 can be used with any beam cross-section defined via **SECTYPE**, **SECDATA**, **SECOFFSET**, **SECWRITE**, and **SECREAD**. The cross-section associated with the beam may be linearly tapered.

Elasticity, creep, and plasticity models are supported (irrespective of cross-section subtype). A cross-section associated with this element type can be a built-up section referencing more than one material.

BEAM188 ignores any real constant data beginning with Release 6.0. See the **SECCONTROLS** command for defining the transverse shear stiffness, and added mass.

For BEAM188, the element coordinate system (**/PSYMB**,ESYS) is not relevant.

Figure 1 BEAM188 Geometry



BEAM188 Input Data

The geometry, node locations, and coordinate system for this element are shown in *Figure 1, "BEAM188 Geometry"*. BEAM188 is defined by nodes I and J in the global coordinate system.

Node K is a preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. See the **LMESH** and **LATT** command descriptions for details on generating the K node automatically.

BEAM188 may also be defined without the orientation node. In this case, the element x-axis is oriented from node I (end 1) toward node J (end 2). For the two-node option, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element x-axis, use the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large deflection analysis, it should be noted that the location of the third node (K) is used only to *initially* orient the element.

The beam elements are one-dimensional line elements in space. The cross-section details are provided separately using the **SECTYPE** and **SECDATA** commands (see Beam Analysis and Cross Sections in the *Structural Analysis Guide* for details). A section is associated with the beam elements by specifying the section ID number (**SECNUM**). A section number is an independent element attribute. In addition to a constant cross-section, you can also define a tapered cross-section by using the TAPER option on the **SECTYPE** command (see Defining a Tapered Beam).

The beam elements are based on Timoshenko beam theory, which is a first order shear deformation theory: transverse shear strain is constant through the cross-section; that is, cross-sections remain plane and undistorted after deformation. BEAM188 is a first order Timoshenko beam element which uses one point of integration along the length with default KEYOPT(3) setting. Therefore, when SMISC quantities are requested at nodes I and J, the centroidal values are reported for *both* end nodes. With KEYOPT(3) set to 2, two points of integration are used resulting in linear variation along the length.

BEAM188/BEAM189 elements can be used for slender or stout beams. Due to the limitations of first order shear deformation theory, only moderately "thick" beams may be analyzed. The slenderness ratio of a beam structure ($GAL^2/(EI)$) may be used in judging the applicability of the element, where:

G
: Shear modulus

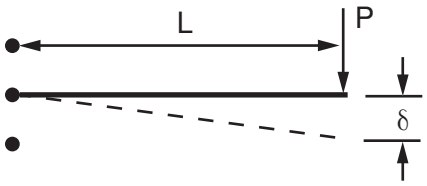
A
: Area of the cross section

L
: Length of the member

EI
: Flexural rigidity

It is important to note that this ratio should be calculated using some global distance measures, and not based on individual element dimensions. The following graphic provides an estimate of transverse shear deformation in a cantilever beam subjected to a tip load. Although the results cannot be extrapolated to any other application, the example serves well as a general guideline. We recommend that the slenderness ratio should be greater than 30.

Figure 2 Transverse Shear Deformation Estimation



Slenderness Ratio ($GAL^2/(EI)$)	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

These elements support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses using the **SECCONTROLS** command.

The St. Venant warping functions for torsional behavior are determined in the undeformed state, and are used to define shear strain even after yielding. ANSYS does not provide options to recalculate in deformed configuration the torsional shear distribution on cross-sections during the analysis and possible partial plastic yielding of cross-sections. As such, large inelastic deformation due to torsional loading should be treated and verified with caution. Under such circumstances, alternative modeling using solid or shell elements is recommended.

BEAM188/BEAM189 elements support "restrained warping" analysis by making available a seventh degree of freedom at each beam node. By default, BEAM188 elements assume that the warping of a cross-section is small enough that it may be neglected (KEYOPT(1) = 0). You can activate the warping degree of freedom by using KEYOPT(1) = 1. With the warping degree of freedom activated, each node has seven degrees of freedom: UX, UY, UZ, ROTX, ROTY, ROTZ, and WARP. With KEYOPT(1) = 1, bimoment and bicurvature are output.

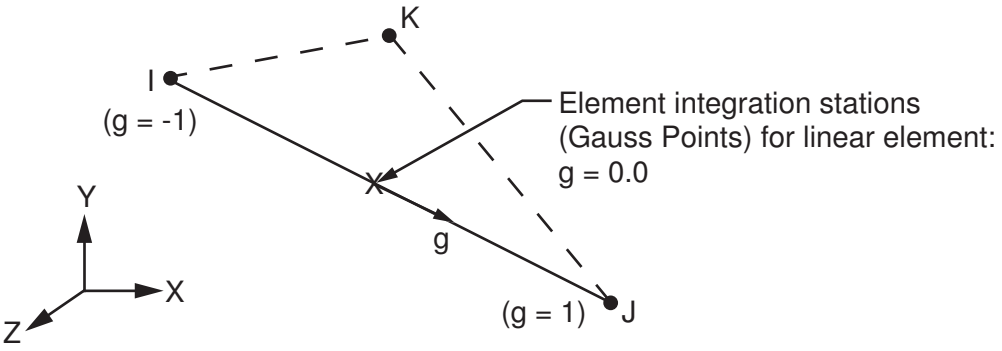
In practice, when two elements with "restrained warping" come together at a sharp angle, you need to couple the displacements and rotations, but leave the out-of-plane warping decoupled. This is normally accomplished by having two nodes at a physical location and using appropriate constraints. This process is made easier (or automated) by the **ENDRELEASE** command, which decouples the out-of plane warping for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

BEAM188 allows change in cross-sectional inertia properties as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you can choose to keep the cross-section constant or rigid. Scaling is not an option for nonlinear general beam sections (**SECTYPE**,,GENB).

Element output is available at element integration stations and at section integration points.

Integration stations (Gauss points) along the length of the beam are shown in *Figure 3, "BEAM188 Element Integration Stations"*.

Figure 3 BEAM188 Element Integration Stations



The section strains and forces (including bending moments) may be obtained at these integration stations. The element supports output options to extrapolate such quantities to the nodes of the element.

BEAM188/BEAM189 can be associated with either of these cross section types:

- Generalized beam cross sections (**SECTYPE**,,GENB), where the relationships of generalized stresses to generalized strains are input directly.
- Standard library section types or user meshes which define the geometry of the beam cross section (**SECTYPE**,,BEAM). The material of the beam is defined either as an element attribute (MAT), or as part of section buildup (for multi-material cross sections).

Generalized Beam Cross Sections

When using nonlinear general beam sections, neither the geometric properties nor the material is explicitly specified. *Generalized stress* implies the axial force, bending moments, torque, and transverse shear forces. Similarly, *generalized strain* implies the axial strain, bending curvatures, twisting curvature, and transverse shear strains. (For more information, see *Section 16.4: Using Nonlinear General Beam Sections*.) This is an abstract method for representing cross section behavior; therefore, input often consists of experimental data or the results of other analyses.

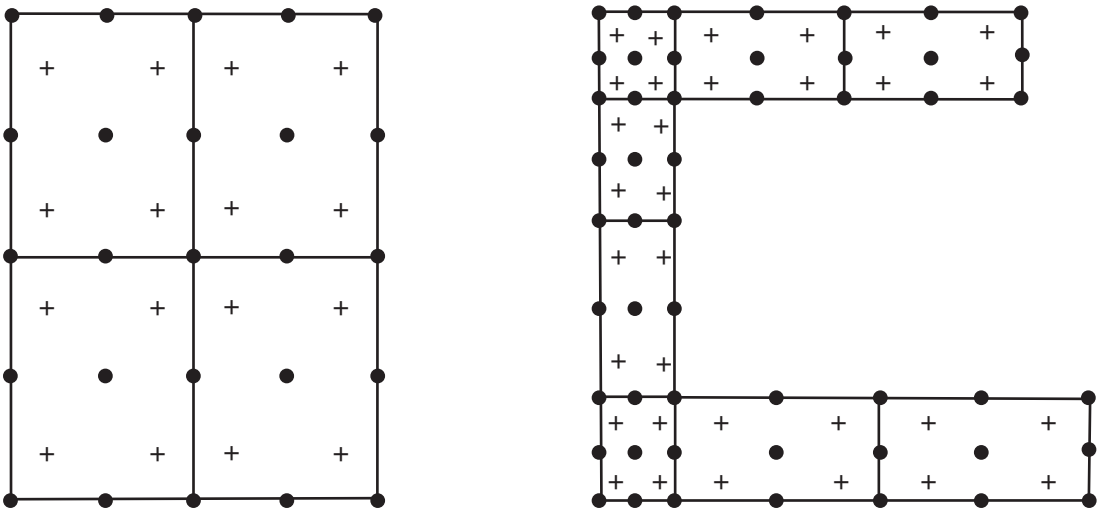
The BEAM188/BEAM189 elements, in general, support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses via the **SECCONTROLS** command.

When the beam element is associated with a generalized beam (**SECTYPE**,,GENB) cross section type, the relationship of transverse shear force to the transverse shear strain can be nonlinear elastic or plastic, an especially useful capability when flexible spot welds are modeled. In such a case, the **SECCONTROLS** command does not apply.

Standard Library Sections

BEAM188/BEAM189 are provided with section-relevant quantities (area of integration, position, Poisson function, function derivatives, etc.) automatically at a number of section points using **SECTYPE** and **SECDATA**. Each section is assumed to be an assembly of a predetermined number of 9-node cells. The following graphic illustrates models using the rectangular section subtype and the channel section subtype. Each cross-section cell has 4 integration points and each may be associated with an independent material type.

Figure 4 Cross-Section Cells



(a) Rectangular section

(b) Channel section

● Section Nodes
+ Section Integration Points

BEAM188/BEAM189 provide options for output at the section integration points and/or section nodes. You can request output only on the exterior boundary of the cross-section. (**PRSSOL** prints the section nodal and section integration point results. Stresses and strains are printed at section nodes, and plastic strains, plastic work, and creep strains are printed at section integration points.)

When the material associated with the elements has inelastic behavior or when the temperature varies across the section, constitutive calculations are performed at the section integration points. For more common elastic applications, the element uses precalculated properties of the section at the element integration points. However, the stresses and strains are calculated in the output pass at the section integration points.

If the section is assigned the subtype ASEC, only the generalized stresses and strains (axial force, bending moments, transverse shears, curvatures, and shear strains) are available for output. 3-D contour plots and deformed shapes are not available. The ASEC subtype can be displayed only as a thin rectangle to verify beam orientation.

BEAM188/BEAM189 allow for the analysis of built-up beams, (i.e., those fabricated of two or more pieces of material joined together to form a single, solid beam). The pieces are assumed to be perfectly bonded together. Therefore, the beam behaves as a single member.

The multi-material cross-section capability is applicable only where the assumptions of a beam behavior (Timoshenko or Bernoulli-Euler beam theory) holds.

In other words, what is supported is a simple extension of a conventional Timoshenko beam theory. It may be used in applications such as:

- bimetallic strips
- beams with metallic reinforcement
- sensors where layers of a different material has been deposited

BEAM188/BEAM189 do not account for coupling of bending and twisting at the section stiffness level. The transverse shears are also treated in an uncoupled manner. This may have a significant effect on layered composite and sandwich beams if the layup is unbalanced.

BEAM188/BEAM189 do not use higher order theories to account for variation in distribution of shear stresses. Use ANSYS solid elements if such effects must be considered.

Always validate the application of BEAM188/BEAM189 for particular applications, either with experiments or other numerical analysis. Use the restrained warping option with built-up sections after due verification.

For the mass matrix and evaluation of consistent load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Use **LUMPM**,ON to activate lumped mass matrix. Consistent mass matrix is used by default. An added mass per unit length may be input with the ADDMAS section controls. See *BEAM188 Input Summary*.

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause bending. Applied shear forces will cause torsional strains and moment if the centroid and shear center of the cross-section are different. The nodes should therefore be located at the desired points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the **SECOFFSET** command appropriately. By default, ANSYS uses the centroid as the reference axis for the beam elements.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM188 Geometry"*. Positive normal pressures act into the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

When KEYOPT(3) = 0 (default), BEAM188 is based on linear polynomials, unlike other Hermitian polynomial-based elements (for example, BEAM4). Refinement of the mesh is recommended in general.

When KEYOPT(3) = 2, ANSYS adds an internal node in the interpolation scheme, effectively making this a Timoshenko beam element based on quadratic shape functions. This option is highly recommended unless this element is used as a stiffener and you must maintain compatibility with a first order shell element. Linearly varying bending moments are represented exactly. The quadratic option is similar to BEAM189, with the following differences:

- The initial geometry is always a straight line with BEAM188 with or without the quadratic option.
- Because you cannot access the internal nodes, you cannot define boundary/loading/initial conditions on those nodes; therefore, you will notice discrepancies between BEAM188 and BEAM189 results if both midside and end nodes have specified boundary/loading/initial conditions in a BEAM189 model.

Offsets in specification of distributed loads are not allowed. Non-nodal concentrated forces are not supported. Use the quadratic option (KEYOPT(3) = 2) when the element is associated with tapered cross-sections.

Temperatures may be input as element body loads at three locations at each end node of the beam. At each end, the element temperatures are input at the element x-axis (T(0,0)), at one unit from the x-axis in the element y-direction (T(1,0)), and at one unit from the x-axis in the element z-direction (T(0,1)). The first coordinate temperature T(0,0) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other input pattern, unspecified temperatures default to TUNIF.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10)

= 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *BEAM188 Input Summary*.

BEAM188 Input Summary

Nodes

I, J, K (K, the orientation node, is optional but recommended)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0
UX, UY, UZ, ROTX, ROTY, ROTZ, WARP if KEYOPT(1) = 1

Section Controls

TXZ, TXY, ADDMAS (See **SECCONTROLS**)
(TXZ and TXY default to A*GXZ and A*GXY, respectively, where A = cross-sectional area)

Material Properties

EX, (PRXY or NUXY), ALPX, DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressure --

face 1 (I-J) (-z normal direction),
face 2 (I-J) (-y normal direction),
face 3 (I-J) (+x tangential direction),
face 4 (J) (+x axial direction),
face 5 (I) (-x direction).
(use a negative value for loading in the opposite direction)
I and J denote the end nodes.

Body Loads

Temperatures --

T(0,0), T(1,0), T(0,1) at each end node

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Birth and death (requires KEYOPT(11) = 1)
Automatic selection of element technology
Generalized cross section (nonlinear elastic, elasto-plastic, temperature-dependent)

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)

Warping degree of freedom:

0 --

Default; six degrees of freedom per node, unrestrained warping

1 --

Seven degrees of freedom per node (including warping). Bimoment and bicurvature are output.

KEYOPT(2)

Cross-section scaling, applies only if **NLGEOM**,ON has been invoked:

0 --

Default; cross-section is scaled as a function of axial stretch

1 --

Section is assumed to be rigid (classical beam theory)

KEYOPT(3)

Interpolation scheme:

0 --

Default; linear polynomial. Mesh refinement is recommended.

2 --

Quadratic shape functions (effectively a Timoshenko beam element); uses an internal node (inaccessible to users) to enhance element accuracy, allowing exact representation of linearly varying bending moments

KEYOPT(4)

Shear stress output:

0 --

Default; output only torsion-related shear stresses

1 --

Output only flexure-related transverse shear stresses

2 --

Output a combined state of the previous two types

KEYOPT(6) through KEYOPT(9) are active only when **OUTPR**,ESOL is active. When KEYOPT(6), (7), (8), and (9) are active, the strains reported in the element output are total strains. "Total" implies the inclusion of thermal strains. When the material associated with the element has plasticity, plastic strain and plastic work are also provided. Alternatively, use **PRSSOL** in /POST1.

KEYOPT(6)

Output control at element integration point:

0 --

Default; output section forces, strains, and bending moments

1 --
Same as KEYOPT(6) = 0 plus current section area

2 --
Same as KEYOPT(6) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments and strains/curvatures extrapolated to element nodes

KEYOPT(7)
Output control at section integration point (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(7) = 1 plus stresses and strains at each section point

KEYOPT(8)
Output control at section nodes (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(8) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(8) = 1 plus stresses and strains at each section node

KEYOPT(9)
Output control for extrapolated values at element nodes and section nodes (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(9) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(9) = 1 plus stresses and strains at all section nodes

KEYOPT(10)
User-defined initial stresses:

0 --
No user subroutine to provide initial stresses (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

KEYOPT(11)
Set section properties:

0 --
Automatically determine if pre-integrated section properties can be used (default)

1 --
Use numerical integration of section (required for birth/death functionality)

KEYOPT(12)
Tapered section treatment:

0 --
Linear tapered section analysis; cross section properties are evaluated at each Gauss point (default). This is more accurate, but computationally intense.

1 --
Average cross section analysis; for elements with tapered sections, cross section properties are evaluated at the centroid only. This is an approximation of the order of the mesh size; however, it is faster.

BEAM188 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in *Table 1, "BEAM188 Element Output Definitions"*

Where necessary, ANSYS recommends KEYOPT(8) = 2 and KEYOPT(9) = 2. See the *Basic Analysis Guide* for ways to view results.

To view 3-D deformed shapes for BEAM188, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Linearized Stress

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately. Therefore, BEAM188 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR = FX/A, where FX is the axial load (SMISC quantities 1 and 14) and A is the area of the cross section.

SBYT and SBYB are bending stress components.

SBYT = -MZ * y_{max} / Izz
SBYB = -MZ * y_{min} / Izz
SBZT = MY * z_{max} / Iyy
SBZB = MY * z_{min} / Iyy

where MY, MZ are bending moments (SMISC quantities 2,15,3,16). Coordinates y_{max}, y_{min}, z_{max}, and z_{min} are the maximum and minimum y, z coordinates in the cross section measured from the centroid. Values Iyy and Izz are moments of inertia of the cross section. Except for the ASEC type of beam cross section, ANSYS uses the maximum and minimum cross section dimensions. For the ASEC type of cross section, the maximum and minimum in each of Y and Z direction is assumed to be +0.5 to -0.5, respectively.

Corresponding definitions for the component strains are:

EPELDIR = EX
EPELBYT = -KZ * y_{max}
EPELBYB = -KZ * y_{min}
EPELBZT = KY * z_{max}
EPELBZB = KY * z_{min}

where EX, KY, and KZ are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. BEAM188 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses may at best be regarded as linearized approximations and should be interpreted with caution.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 BEAM188 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Center of gravity	Y	Y
AREA	Area of cross-section	*1*	Y
SF:Y, Z	Section shear forces	*1*	Y
SE:Y, Z	Section shear strains	*1*	Y
S:XX, XZ, XY	Section point stresses	*2*	Y
EPTO:XX, XZ, XY	Section point total strains (EPEL + EPPL + EPTH)	*2*	Y
EPPL:XX, XZ, XY	Section point plastic strains	*2*	Y
EPCR:XX, XZ, XY	Section point creep strains	*2*	Y
EPTH:XX, XZ, XY	Section point thermal strains	*2*	Y
MX	Torsional moment	Y	Y
KX	Torsional strain	Y	Y
KY, KZ	Curvature	Y	Y
EX	Axial strain	Y	Y
FX	Axial force	Y	Y
MY, MZ	Bending moments	Y	Y
BM	Bimoment	*3*	*3*
BK	Bicurvature	*3*	*3*
SDIR	Axial direct stress	-	*1*
SBYT	Bending stress on the element +Y side of the beam	-	*1*

Name	Definition	O	R
SBYB	Bending stress on the element -Y side of the beam	-	*1*
SBZT	Bending stress on the element +Z side of the beam	-	*1*
SBZB	Bending stress on the element -Z side of the beam	-	*1*
EPELDIR	Axial strain at the end	-	*1*
EPELBYT	Bending strain on the element +Y side of the beam.	-	*1*
EPELBYB	Bending strain on the element -Y side of the beam.	-	*1*
EPELBZT	Bending strain on the element +Z side of the beam.	-	*1*
EPELBZB	Bending strain on the element -Zside of the beam.	-	*1*
TEMP	Temperatures T0, T1(1,0), T2(0,1)	-	*1*

Note

More output is described via the **PRSSOL** command in **/POST1**.

1. See KEYOPT(6) description.
2. See KEYOPT(7), KEYOPT(8), KEYOPT(9) descriptions.
3. See KEYOPT(1) description.

Table 2, "BEAM188 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. *Table 2, "BEAM188 Item and Sequence Numbers"* uses the following notation:

Name
: output quantity as defined in the *Table 1, "BEAM188 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE**

I,J
: sequence number for data at nodes I and J

Table 2 BEAM188 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
FX	SMISC	1	14
MY	SMISC	2	15
MZ	SMISC	3	16
MX	SMISC	4	17
SFZ	SMISC	5	18
SFY	SMISC	6	19
EX	SMISC	7	20
KY	SMISC	8	21
KZ	SMISC	9	22
KX	SMISC	10	23
SEZ	SMISC	11	24
SEY	SMISC	12	25

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
Area	SMISC	13	26
BM	SMISC	27	29
BK	SMISC	28	30
SDIR	SMISC	31	36
SBYT	SMISC	32	37
SBYB	SMISC	33	38
SBZT	SMISC	34	39
SBZB	SMISC	35	40
EPELDIR	SMISC	41	46
EPELBYT	SMISC	42	47
EPELBYB	SMISC	43	48
EPELBZT	SMISC	44	49
EPELBZB	SMISC	45	50
TEMP	SMISC	51-53	54-56
S:XX, XZ, XY	LS	i	3 * $nNode + i$
EPTO:XX, XZ, XY	LEPEL	i	3 * $nNode + i$
EPTH:XX, XZ, XY	LEPTH	i	3 * $nNode + i$
EPPL:XX, XZ, XY	LEPPL	i	3 * $nIntg + i$
EPCR:XX, XZ, XY	LEPCR	i	3 * $nIntg + i$

Note

The value i in *Table 2, "BEAM188 Item and Sequence Numbers"* refers to the order of the node or integration point of the beam where $1 \leq i \geq (3 * nNode)$ (number of nodes), or $1 \leq i \geq (3 * nIntg)$ (number of integration points).

For more usage details, see *Section 16.6.13: Plot and Review the Section Results* and *Section 16.7: Sample Problem with Cantilever Beams, Command Method*.

Transverse Shear Stress Output

BEAM188/BEAM189 formulation is based on three stress components:

- one axial
- two shear stress components

The shear stresses are caused by torsional and transverse loads. BEAM188/BEAM189 are based on first order shear deformation theory, also popularly known as Timoshenko Beam theory. The transverse shear strain is constant for the cross section, and hence the shear energy is based on a transverse shear force. This shear force is redistributed by predetermined shear stress distribution coefficients across the beam cross-section, and made available for output purposes. By default, ANSYS will only output the shear stresses caused by torsional loading. KEYOPT(4) of BEAM188/BEAM189 may be used to activate output of shear stresses caused by flexure or transverse loading.

The accuracy of transverse shear distribution is directly proportional to the mesh density of cross-section modeling (for determination of warping, shear center and other section geometric properties). The traction free state at the edges of cross-section, is met only in a well-refined model of the cross-section.

By default, ANSYS uses a mesh density (for cross-section model) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear center determination. The default mesh employed is also appropriate for nonlinear material calculations. However, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Note that increasing cross-section mesh size, does not imply larger computational cost if the associated material is linear. **SECTYPE** and **SECDATA** command descriptions allow specification of cross-section mesh density.

The transverse shear distribution calculation neglects the effects of Poisson's ratio. The Poisson's ratio affects the shear correction factor and shear stress distribution slightly.

BEAM188 Assumptions and Restrictions

- The beam must not have zero length.
- By default (KEYOPT(1) = 0), the effect of warping restraint is assumed to be negligible.
- Cross-section failure or folding is not accounted for.
- Rotational degrees of freedom are not included in the lumped mass matrix if offsets are present.
- It is a common practice in civil engineering to model the frame members of a typical multi-storied structure using a single element for each member. Because of cubic interpolation of lateral displacement, BEAM4 and BEAM44 are well-suited for such an approach. However, if BEAM188 is used in that type of application, be sure to use several elements for each frame member. BEAM188 includes the effects of transverse shear.
- This element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control). For nonlinear problems that are dominated by large rotations, we recommend that you do *not* use **PRED**,ON.
- Note that only moderately "thick" beams may be analyzed. See *BEAM188 Input Data* for more information.
- When a cross-section has multiple materials and you issue the **/ESHAPE** command (which displays elements with shapes determined from the real constants or section definition) to produce contour plots of stresses (and other quantities), the element averages the stresses across material boundaries. To limit this behavior, use small cross-section cells around the material boundaries. There are no input options to bypass this behavior.
- For this element, the **/ESHAPE** command supports visualization of stresses, but not of plastic strains.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.
- When the element is associated with nonlinear general beam sections (**SECTYPE**,,GENB), additional restrictions apply. For more information, see *Section 16.4.2: Considerations for Employing Nonlinear General Beam Sections*.

BEAM188 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

BEAM189

3-D Quadratic Finite Strain Beam

MP ME ST PR PRN DS DSS <> <> <> <> PP VT
Product Restrictions

BEAM189 Element Description

BEAM189 is an element suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.

BEAM189 is a quadratic (3-node) beam element in 3-D. BEAM189 has six or seven degrees of freedom at each node, with the number of degrees of freedom depending on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

BEAM189 includes stress stiffness terms, by default, in any analysis with **NLGEOM**,ON. The provided stress stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling or collapse studies with arc length methods).

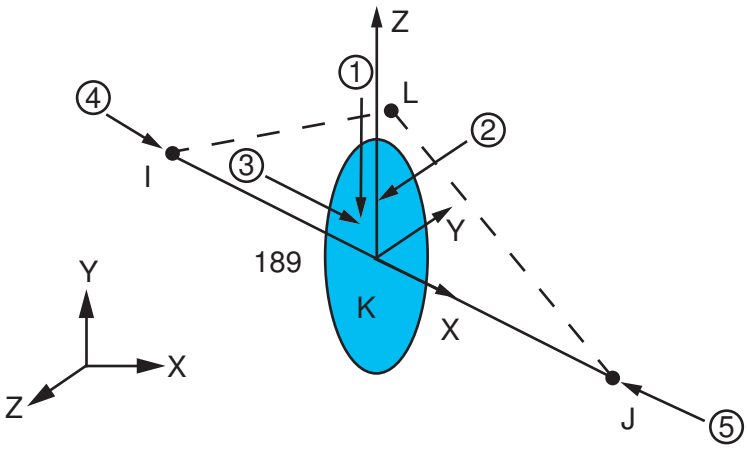
BEAM189 can be used with any beam cross-section that was defined using **SECTYPE**, **SECDATA**, **SECOFFSET**, **SECWRITE**, and **SECREAD**. The cross-section associated with the beam may be linearly tapered.

Elasticity, creep, and plasticity models are supported (irrespective of cross-section subtype). A cross-section associated with this element type can be a built-up section referencing more than one material.

BEAM189 will ignore any real constant data beginning with Release 6.0. See **SECCONTROLS** command for defining the transverse shear stiffness, and added mass.

For BEAM189, the element coordinate system (**/PSYMB**,ESYS) is not relevant.

Figure 1 BEAM189 Geometry



BEAM189 Input Data

The geometry, node locations, and coordinate system for this element are shown in *Figure 1, "BEAM189 Geometry"*. BEAM189 is defined by nodes I, J, and K in the global coordinate system. .

Node L is a preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. Also, see Quadratic Elements (Midside Nodes) in the same manual for the use of midside nodes. See the **LMESH** and **LATT** command descriptions for details on generating the L node automatically. For a description of the low-order beam, see BEAM188.

BEAM189 may also be defined without the orientation node L. In this case, the element x-axis is oriented from node I (end 1) toward node J (end 2). For the two-node option, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element x-axis, use the "L" node option. If both are defined, the orientation node option takes precedence. The orientation node (L), if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large deflection analysis, it should be noted that the location of the orientation node (L) is used only to *initially* orient the element.

The beam elements are one-dimensional line elements in space. The cross-section details are provided separately using the **SECTYPE** and **SECDATA** commands (see Beam Analysis and Cross Sections in the *Structural Analysis Guide* for details). A section is associated with the beam elements by specifying the section ID number (**SECNUM**). A section number is an independent attribute. In addition to a constant cross-section, you can also define a tapered cross-section by using the TAPER option on the **SECTYPE** command (see Defining a Tapered Beam).

The beam elements are based on Timoshenko beam theory, which is a first order shear deformation theory: transverse shear strain is constant through the cross-section; that is, cross-sections remain plane and undistorted after deformation.

BEAM188/BEAM189 elements can be used for slender or stout beams. Due to the limitations of first order shear deformation theory, only moderately "thick" beams may be analyzed. The slenderness ratio of a beam structure ($GAL^2/(EI)$) may be used in judging the applicability of the element, where:

G
: Shear modulus

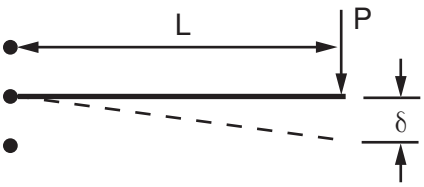
A
: Area of the cross section

L
: Length of the member

EI
: Flexural rigidity

It is important to note that this ratio should be calculated using some global distance measures, and not based on individual element dimensions. The following graphic provides an estimate of transverse shear deformation in a cantilever beam subjected to a tip load. Although the results cannot be extrapolated to any other application, the example serves well as a general guideline. We recommend that the slenderness ratio should be greater than 30.

Figure 2 Transverse Shear Deformation Estimation



Slenderness Ratio ($GAL^2/(EI)$)	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

These elements support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses using the **SECCONTROLS** command.

The St. Venant warping functions for torsional behavior are determined in the undeformed state, and are used to define shear strain even after yielding. ANSYS does not provide options to recalculate in deformed configuration the torsional shear distribution on cross-sections during the analysis and possible partial plastic yielding of cross-sections. As such, large inelastic deformation due to torsional loading should be treated and verified with caution. Under such circumstances, alternative modeling using solid or shell elements is recommended.

BEAM188/BEAM189 elements support "restrained warping" analysis by making available a seventh degree of freedom at each beam node. By default, BEAM189 elements assume that the warping of a cross-section is small enough that it may be neglected (KEYOPT(1) = 0). You can activate the warping degree of freedom by using KEYOPT(1) = 1. With the warping degree of freedom activated, each node has seven degrees of freedom: UX, UY, UZ, ROTX, ROTZ, ROTY, and WARP. With KEYOPT(1) = 1, bimoment and bicurvature are output.

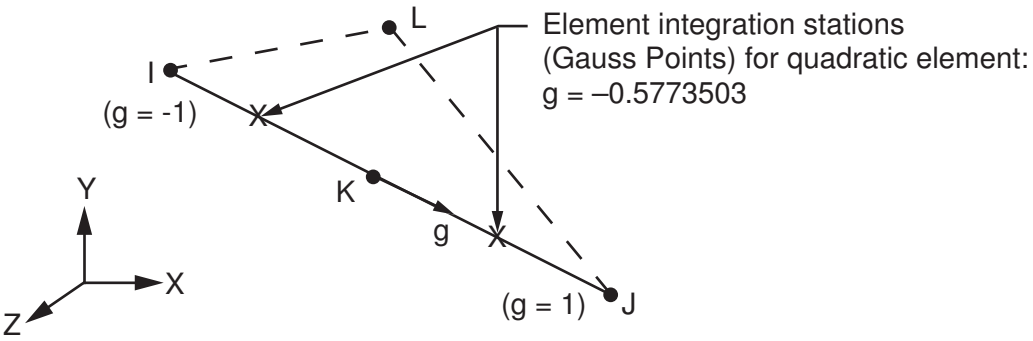
In practice, when two elements with "restrained warping" come together at a sharp angle, you need to couple the displacements and rotations, but leave the out-of-plane warping decoupled. This is normally accomplished by having two nodes at a physical location and using appropriate constraints. This process is made easier (or automated) by the **ENDRELEASE** command, which decouples the out-of plane warping for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

BEAM189 allows change in cross-sectional inertia properties as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you can choose to keep the cross-section constant or rigid. Scaling is not an option for nonlinear general beam sections (**SECTYPE**,,GENB).

Element output is available at element integration stations and at section integration points.

Integration stations (Gauss points) along the length of BEAM189 are shown in *Figure 3, "BEAM189 Element Integration Stations"*.

Figure 3 BEAM189 Element Integration Stations



The section strains and forces (including bending moments) may be obtained at these integration stations. The element supports output options to extrapolate such quantities to the nodes of the element.

BEAM188/BEAM189 can be associated with either of these cross section types:

- Generalized beam cross sections (**SECTYPE**,,GENB), where the relationships of generalized stresses to generalized strains are input directly.
- Standard library section types or user meshes which define the geometry of the beam cross section (**SECTYPE**,,BEAM). The material of the beam is defined either as an element attribute (MAT), or as part of section buildup (for multi-material cross sections).

Generalized Beam Cross Sections

When using nonlinear general beam sections, neither the geometric properties nor the material is explicitly specified. *Generalized stress* implies the axial force, bending moments, torque, and transverse shear forces. Similarly, *generalized strain* implies the axial strain, bending curvatures, twisting curvature, and transverse shear strains. (For more information, see *Section 16.4: Using Nonlinear General Beam Sections*.) This is an abstract method for representing cross section behavior; therefore, input often consists of experimental data or the results of other analyses.

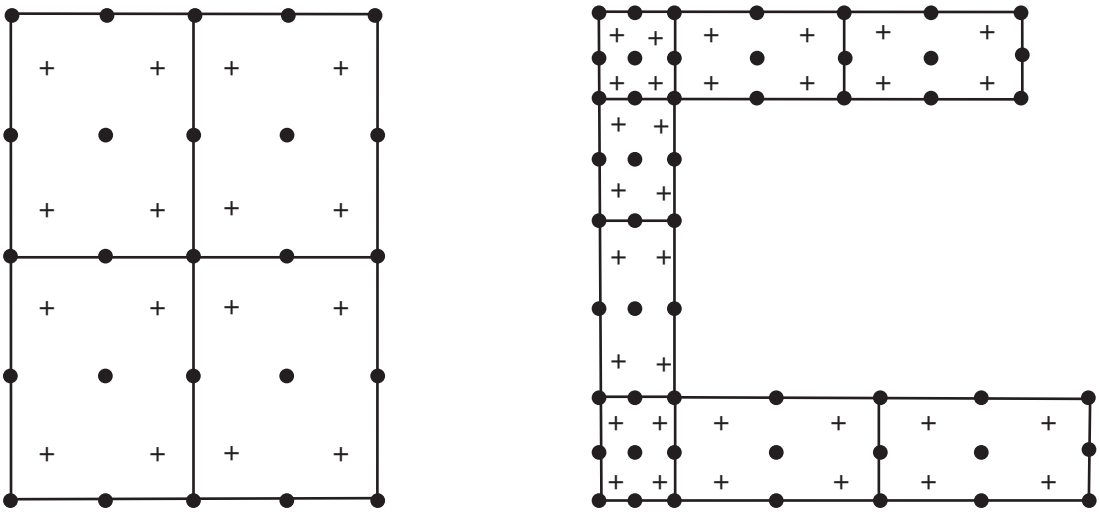
The BEAM188/BEAM189 elements, in general, support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses via the **SECCONTROLS** command.

When the beam element is associated with a generalized beam (**SECTYPE**,,GENB) cross section type, the relationship of transverse shear force to the transverse shear strain can be nonlinear elastic or plastic, an especially useful capability when flexible spot welds are modeled. In such a case, the **SECCONTROLS** command does not apply.

Standard Library Sections

BEAM188/BEAM189 are provided with section-relevant quantities (area of integration, position, Poisson function, function derivatives, etc.) automatically at a number of section points using **SECTYPE** and **SECDATA**. Each section is assumed to be an assembly of a predetermined number of 9-node cells. The following graphic illustrates models using the rectangular section subtype and the channel section subtype. Each cross-section cell has 4 integration points and each may be associated with an independent material type.

Figure 4 Cross-Section Cells



(a) Rectangular section

(b) Channel section

● Section Nodes
+ Section Integration Points

BEAM188/BEAM189 provide options for output at the section integration points and/or section nodes. You can request output only on the exterior boundary of the cross-section. (**PRSSOL** prints the section nodal and section integration point results. Stresses and strains are printed at section nodes, and plastic strains, plastic work, and creep strains are printed at section integration points.)

When the material associated with the elements has inelastic behavior or when the temperature varies across the section, constitutive calculations are performed at the section integration points. For more common elastic applications, the element uses precalculated properties of the section at the element integration points. However, the stresses and strains are calculated in the output pass at the section integration points.

If the section is assigned the subtype ASEC, only the generalized stresses and strains (axial force, bending moments, transverse shears, curvatures, and shear strains) are available for output. 3-D contour plots and deformed shapes are not available. The ASEC subtype can be displayed only as a thin rectangle to verify beam orientation.

BEAM188/BEAM189 allow for the analysis of built-up beams, (i.e., those fabricated of two or more pieces of material joined together to form a single, solid beam). The pieces are assumed to be perfectly bonded together. Therefore, the beam behaves as a single member.

The multi-material cross-section capability is applicable only where the assumptions of a beam behavior (Timoshenko or Bernoulli-Euler beam theory) holds.

In other words, what is supported is a simple extension of a conventional Timoshenko beam theory. It may be used in applications such as:

- bimetallic strips
- beams with metallic reinforcement
- sensors where layers of a different material has been deposited

BEAM188/BEAM189 do not account for coupling of bending and twisting at the section stiffness level. The transverse shears are also treated in an uncoupled manner. This may have a significant effect on layered composite and sandwich beams if the layup is unbalanced.

BEAM188/BEAM189 do not use higher order theories to account for variation in distribution of shear stresses. Use ANSYS solid elements if such effects must be considered.

Always validate the application of BEAM188/BEAM189 for particular applications, either with experiments or other numerical analysis. Use the restrained warping option with built-up sections after due verification.

For the mass matrix and evaluation of consistent load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Avoid using **LUMPM**,ON as BEAM189 is a higher-order element. Consistent mass matrix is used by default. An added mass per unit length may be input with the ADDMAS section controls. See *BEAM189 Input Summary*.

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause bending. Applied shear forces will cause torsional strains and moment if the centroid and shear center of the cross-section are different. The nodes should therefore be located at the points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the **SECOFFSET** command appropriately.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "BEAM189 Geometry"*. Positive normal pressures act into the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

BEAM189 is based on quadratic polynomials, unlike other Hermitian polynomial-based elements in ANSYS (for example, BEAM4). Due to this, the offsets in specification of distributed loads are not allowed. In addition, non-nodal concentrated forces are not supported. BEAM189 has the same linear bending moment variation as BEAM4. Refinement of the mesh is recommended in order to accommodate such loading. BEAM189 is computationally efficient and has super-convergence properties with respect to mesh refinement. For example, the quadratic beam with a two point Gaussian integration is known to be of same accuracy as a Hermitian element.

Temperatures may be input as element body loads at three locations at each end node of the beam. At each end, the element temperatures are input at the element x-axis (T(0,0)), at one unit from the x-axis in the element y-direction (T(1,0)), and at one unit from the x-axis in the element z-direction (T(0,1)). The first coordinate temperature T(0,0) defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other input pattern, unspecified temperatures default to TUNIF.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input follows in *BEAM189 Input Summary*.

BEAM189 Input Summary

Nodes

I, J, K, L (L, the orientation node, is optional but recommended)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0
UX, UY, UZ, ROTX, ROTY, ROTZ, WARP if KEYOPT(1) = 1

Section Controls

TXZ, TXY, ADDMAS (See **SECCONTROLS**)
(TXZ and TXY default to A*GXZ and A*GXY, respectively, where A = cross-sectional area)

Material Properties
EX, (PRXY or NUXY), ALPX, DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressure --

face 1 (I-J) (-z normal direction),
face 2 (I-J) (-y normal direction),
face 3 (I-J) (+x tangential direction),
face 4 (J) (+x axial direction),
face 5 (I) (-x direction).
(use a negative value for loading in the opposite direction)
I and J denote the end nodes.

Body Loads

Temperatures --
T(0,0), T(1,0), T(0,1) at each end node

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Other material (USER)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Birth and death (requires KEYOPT(11) = 1)
Automatic selection of element technology
Generalized cross section (nonlinear elastic, elasto-plastic, temperature-dependent)

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Warping degree of freedom:

0 --
Default; six degrees of freedom per node, unrestrained warping

1 --
Seven degrees of freedom per node (including warping). Bimoment and bicurvature are output.

KEYOPT(2)
Cross-section scaling:

0 --
Default; cross-section is scaled as a function of axial stretch; applies only if **NLGEOM**,ON has been invoked

1 --
Section is assumed to be rigid (classical beam theory)

KEYOPT(4)
Shear stress output:

0 --
Default; output only torsion-related shear stresses

1 --
Output only flexure-related transverse shear stresses

2 --
Output a combined state of the previous two types.

KEYOPT(6) through KEYOPT(9) are active only when **OUTPR**,ESOL is active. When KEYOPT(6), (7), (8), and (9) are active, the strains reported in the element output are total strains. "Total" implies the inclusion of thermal strains. When the material associated with the element has plasticity, plastic strain and plastic work are also provided. Alternatively, use **PRSSOL** in /POST1.

KEYOPT(6)
Output control at element integration point:

0 --
Default; output section forces, strains, and bending moments

1 --
Same as KEYOPT(6) = 0 plus current section area

2 --
Same as KEYOPT(6) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments and strains/curvatures extrapolated to element nodes

KEYOPT(7)
Output control at section integration point (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(7) = 1 plus stresses and strains at each section point

KEYOPT(8)
Output control at section nodes (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(8) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(8) = 1 plus stresses and strains at each section node

KEYOPT(9)
Output control for extrapolated values at element nodes and section nodes (not available when section subtype = ASEC):

0 --
Default; none

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(9) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(9) = 1 plus stresses and strains at all section nodes

KEYOPT(10)
User-defined initial stresses:

0 --
No user subroutine to provide initial stresses (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

KEYOPT(11)
Set section properties:

0 --
Automatically determine if pre-integrated section properties can be used (default)

1 --
Use numerical integration of section (required for birth/death functionality)

KEYOPT(12)
Tapered section treatment:

0 --
Linear tapered section analysis; cross section properties are evaluated at each Gauss point (default). This is more accurate, but computationally intense.

1 --
Average cross section analysis; for elements with tapered sections, cross section properties are evaluated at the centroid only. This is an approximation of the order of the mesh size; however, it is faster.

BEAM189 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in *Table 1, "BEAM189 Element Output Definitions"*

Where necessary, we recommend KEYOPT(8) = 2 and KEYOPT(9) = 2. See the *Basic Analysis Guide* for ways to view results.

To view 3-D deformed shapes for BEAM189, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the **MXPAND** command's *Elcalc* = YES option).

Linearized Stress

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately. Therefore, BEAM189 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR = FX/A, where FX is the axial load (SMISC quantities 1 and 14) and A is the area of the cross section.

SBYT and SBYB are bending stress components.

SBYT = -MZ * y_{max} / Izz
SBYB = -MZ * y_{min} / Izz
SBZT = MY * z_{max} / Iyy
SBZB = MY * z_{min} / Iyy

where MY, MZ are bending moments (SMISC quantities 2,15,3,16). Coordinates y_{max}, y_{min}, z_{max}, and z_{min} are the maximum and minimum y, z coordinates in the cross section measured from the centroid. Values Iyy and Izz are moments of inertia of the cross section. Except for the ASEC type of beam cross section, ANSYS uses the maximum and minimum cross section dimensions. For the ASEC type of cross section, the maximum and minimum in each of Y and Z direction is assumed to be +0.5 to -0.5, respectively.

Corresponding definitions for the component strains are:

EPELDIR = EX
EPELBYT = -KZ * y_{max}
EPELBYB = -KZ * y_{min}
EPELBZT = KY * z_{max}
EPELBZB = KY * z_{min}

where EX, KY, and KZ are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. BEAM189 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses may at best be regarded as linearized approximations and should be interpreted with caution.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 BEAM189 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Center of gravity	Y	Y
AREA	Area of cross-section	*1*	Y
SF:Y, Z	Section shear forces	*1*	Y
SE:Y, Z	Section shear strains	*1*	Y
S:XX, XZ, XY	Section point stresses	*2*	Y
EPTO:XX, XZ, XY	Section point total strains (EPEL + EPPL + EPTH)	*2*	Y
EPPL:XX, XZ, XY	Section point plastic strains	*2*	Y
EPCR:XX, XZ, XY	Section point creep strains	*2*	Y
EPTH:XX, XZ, XY	Section point thermal strains	*2*	Y
MX	Torsional moment	Y	Y
KX	Torsional strain	Y	Y
KY, KZ	Curvature	Y	Y
EX	Axial strain	Y	Y
FX	Axial force	Y	Y
MY, MZ	Bending moments	Y	Y
BM	Bimoment	*3*	*3*
BK	Bicurvature	*3*	*3*
SDIR	Axial direct stress	-	*1*
SBYT	Bending stress on the element +Y side of the beam	-	*1*
SBYB	Bending stress on the element -Y side of the beam	-	*1*
SBZT	Bending stress on the element +Z side of the beam	-	*1*
SBZB	Bending stress on the element -Z side of the beam	-	*1*
EPELDIR	Axial strain at the end	-	*1*
EPELBYT	Bending strain on the element +Y side of the beam.	-	*1*
EPELBYB	Bending strain on the element -Y side of the beam.	-	*1*
EPELBZT	Bending strain on the element +Z side of the beam.	-	*1*
EPELBZB	Bending strain on the element -Zside of the beam.	-	*1*
TEMP	Temperatures T0, T1(1,0), T2(0,1)	-	*1*

Note

More output is described via the **PRSSOL** command in **/POST1**.

1. See KEYOPT(6) description
2. See KEYOPT(7), KEYOPT(8), KEYOPT(9) descriptions
3. See KEYOPT(1) description

Table 2, "BEAM 189 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. *Table 2, "BEAM 189 Item and Sequence Numbers"* uses the following notation:

Name
output quantity as defined in the *Table 1, "BEAM189 Element Output Definitions"*

Item
predetermined Item label for **ETABLE**

I,J
sequence number for data at nodes I and J

Table 2 BEAM 189 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
FX	SMISC	1	14
MY	SMISC	2	15
MZ	SMISC	3	16
MX	SMISC	4	17
SFZ	SMISC	5	18
SFY	SMISC	6	19
EX	SMISC	7	20
KY	SMISC	8	21
KZ	SMISC	9	22
KX	SMISC	10	23
SEZ	SMISC	11	24
SEY	SMISC	12	25
Area	SMISC	13	26
BM	SMISC	27	29
BK	SMISC	28	30
SDIR	SMISC	31	36
SBYT	SMISC	32	37
SBYB	SMISC	33	38
SBZT	SMISC	34	39
SBZB	SMISC	35	40
EPELDIR	SMISC	41	46
EPELBYT	SMISC	42	47
EPELBYB	SMISC	43	48
EPELBZT	SMISC	44	49
EPELBZB	SMISC	45	50
TEMP	SMISC	51-53	54-56
S:XX, XZ, XY	LS	*i*	3 * *nNode* + *i*
EPTO:XX,XZ,XY	LEPEL	*i*	3 * *nNode* + *i*
EPTH:XX,XZ,XY	LEPTH	*i*	3 * *nNode* + *i*
EPPL:XX,XZ,XY	LEPPL	*i*	3 * *nIntg* + *i*

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
EPCR:XX,XZ,XY	LEPCR	*i*	3 * *nIntg* + *i*

Note

The value *i* in *Table 2, "BEAM 189 Item and Sequence Numbers"* refers to the order of the node or integration point of the beam where 1 ≤ *i* ≥ (3 * *nNode*) (number of nodes), or 1 ≤ *i* ≥ (3 * *nIntg*) (number of integration points).

For more usage details, see *Section 16.6.13: Plot and Review the Section Results* and *Section 16.7: Sample Problem with Cantilever Beams, Command Method*.

Transverse Shear Stress Output

BEAM188/BEAM189 formulation is based on three stress components:

- one axial
- two shear stress components

The shear stresses are caused by torsional and transverse loads. BEAM188/BEAM189 are based on first order shear deformation theory, also popularly known as Timoshenko Beam theory. The transverse shear strain is constant for the cross section, and hence the shear energy is based on a transverse shear force. This shear force is redistributed by predetermined shear stress distribution coefficients across the beam cross-section, and made available for output purposes. By default, ANSYS will only output the shear stresses caused by torsional loading. KEYOPT(4) of BEAM188/BEAM189 may be used to activate output of shear stresses caused by flexure or transverse loading.

The accuracy of transverse shear distribution is directly proportional to the mesh density of cross-section modeling (for determination of warping, shear center and other section geometric properties). The traction free state at the edges of cross-section, is met only in a well-refined model of the cross-section.

By default, ANSYS uses a mesh density (for cross-section model) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear center determination. The default mesh employed is also appropriate for nonlinear material calculations. However, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Note that increasing cross-section mesh size, does not imply larger computational cost if the associated material is linear. **SECTYPE** and **SECDATA** command descriptions allow specification of cross-section mesh density.

The transverse shear distribution calculation neglects the effects of Poisson's ratio. The Poisson's ratio affects the shear correction factor and shear stress distribution slightly.

BEAM189 Assumptions and Restrictions

- The beam must not have zero length.
- By default (KEYOPT(1) = 0), the effect of warping restraint is assumed to be negligible.
- Cross-section failure or folding is not accounted for.
- Rotational degrees of freedom are not included in the lumped mass matrix if offsets are present.
- It is a common practice in civil engineering to model the frame members of a typical multi-storied structure using a single element for each member. Because of cubic interpolation of lateral displacement, BEAM4 and BEAM44 are well-suited for such an approach. BEAM189, under most circumstances, may provide

accuracy similar to that of the cubic elements, since the linear bending moment variation is accounted for.

- BEAM189 includes the effects of transverse shear and accounts for the initial curvature of the beams.
- This element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control). For nonlinear problems that are dominated by large rotations, we recommend that you do *not* use **PRED**,ON.
- Note that only moderately "thick" beams may be analyzed. See the*BEAM189 Input Data* section for more information.
- When a cross-section has multiple materials and you issue the **/ESHAPE** command (which displays elements with shapes determined from the real constants or section definition) to produce contour plots of stresses (and other quantities), the element averages the stresses across material boundaries. To limit this behavior, use small cross-section cells around the material boundaries. There are no input options to bypass this behavior.
- For this element, the **/ESHAPE** command supports visualization of stresses, but not of plastic strains.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.
- When the element is associated with nonlinear general beam sections (**SECTYPE**,,GENB), additional restrictions apply. For more information, see *Section 16.4.2: Considerations for Employing Nonlinear General Beam Sections*.

BEAM189 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

SOLSH190

3-D 8-Node Layered Solid Shell

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

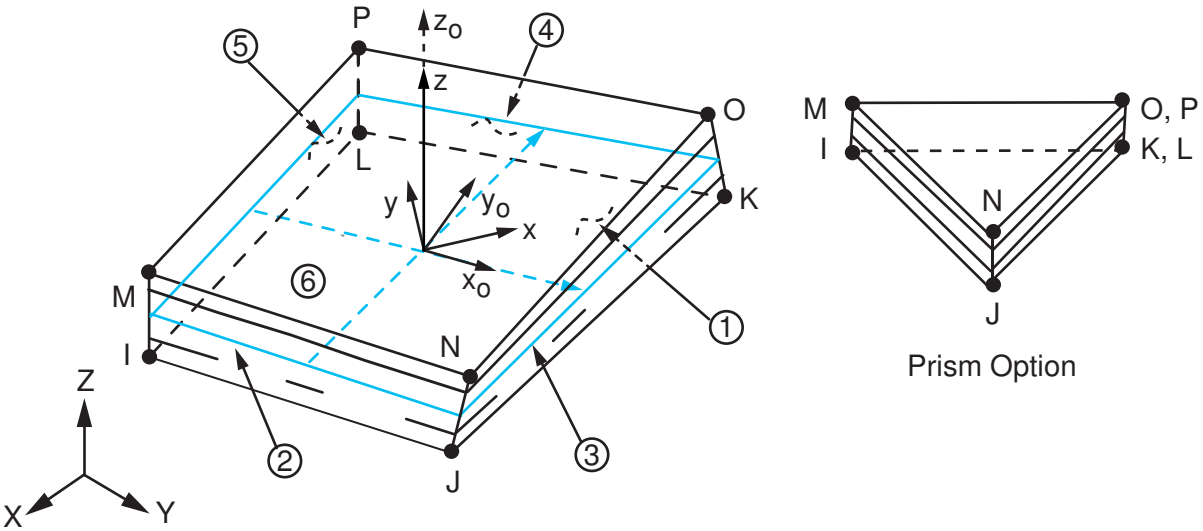
SOLSH190 Element Description

SOLSH190 is used for simulating shell structures with a wide range of thickness (from thin to moderately thick). The element possesses the continuum solid element topology and features eight-node connectivity with three degrees of freedom at each node: translations in the nodal x, y, and z directions. Thus, connecting SOLSH190 with other continuum elements requires no extra efforts. A degenerate prism option is available, but should only be used as filler elements in mesh generation. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed u-P formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. The element formulation is based on logarithmic strain and true stress measures.

You can use SOLSH190 for layered applications such as modeling laminated shells or sandwich construction. The layered section definition is given by ANSYS section (SEC*xxx*) commands. The element allows up to 250 different material layers. Accuracy in modeling composite shells is governed by the first order shear deformation theory (also known as Mindlin-Reissner shell theory).

See SOLSH190 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLSH190 Geometry



x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLSH190 Input Data

The geometry, node locations, and the element coordinate system for this element are shown in *Figure 1, "SOLSH190 Geometry"*. The element is defined by eight nodes. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The node ordering must follow the convention that the I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command as you would for shell elements

(as described in *Section 2.3: Coordinate Systems*). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLSH190 with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered SOLSH190 element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLSH190 does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element. The axis can be defined as shown:

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where:

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{8}\right)\left[-\{x\}^I + \{x\}^J + \{x\}^K - \{x\}^L - \{x\}^M + \{x\}^N + \{x\}^O - \{x\}^P\right]$$

$\{x\}^I, \{x\}^J, \ldots, \{x\}^P$ = global nodal coordinates

You can reorient the default first surface direction S1 in the element reference plane (see Figure 190.1) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See *Section 2.3: Coordinate Systems* for details.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SOLSH190 Geometry"*. Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, you define temperatures with commands other than **BFE**--**SOLSH190** adopts an element-wise temperature pattern and requires only eight temperatures for the eight element nodes. Unspecified nodal temperatures default to the assigned uniform temperature (**TUNIF**). ANSYS computes all layer interface temperatures by interpolating nodal temperatures T1 ~ T8.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-1024 maximum). In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used

for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NL. If you input exactly NL + 1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

You can use the **MP** command to define the isotropic or orthotropic elastic material properties and the **ANEL** command to define anisotropic elastic material properties. Other material properties include density, damping ratios, and coefficients of thermal expansion. You may also use the **TB** command to define nonlinear material behavior such as plasticity, hyperelasticity, viscoelasticity, creep, and viscoplasticity.

KEYOPT(6) = 1 sets the element for using u-P mixed formulation. For details on the use of mixed formulation, see *Section 2.16.3: Applications of Mixed u-P Formulations* in the *Elements Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

SOLSH190 Input Summary contains a summary of element input. For a general description of element input, see *Section 2.1: Element Input*.

SOLSH190 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

Element-wise pattern (no element body load command issued): T1, T2, T3, T4, T5, T6, T7, T8 for 8 element nodes. Temperatures at layer interface corners are computed by interpolating nodal temperatures. Layer-wise pattern (element body load command issued): T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for temperatures between subsequent layers, ending with temper-

atures at top of layer NL (4 * (NL + 1) maximum). For a one-layer element, therefore, 8 temperatures are used.

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ELASTIC, ANEL)
Other material (USER, SDAMP, SMA, CAST, EDP, GURSON)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details about the material models.

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default).

1 --
Use mixed u-P formulation.

KEYOPT(8)
Storage of layer data:

0 --
For multilayer elements, store data for bottom of bottom layer and top of top layer (default).

1 --
For multilayer elements, store data for top and bottom for all layers. (Before using this option, be aware that the amount of data involved can be very large.)

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stresses (default).

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

SOLSH190 Element Technology

SOLSH190 employs incompatible modes to enhance the accuracy in in-plane bending situations. The satisfaction of the in-plane patch test is ensured. A separate set of incompatible modes is adopted to overcome the thickness

locking in bending dominant problems. The incompatible modes introduce seven internal DOFs that are inaccessible to users and condensed out at the element level.

SOLSH190 utilizes a suite of special kinematic formulations to avoid locking when the shell thickness becomes extremely small. However, due to its shell-like behavior, SOLSH190 fails to pass the patch test if the element is distorted in the thickness direction.

SOLSH190 is fully compatible with 3-D constitutive relations. Compared to classical shell elements that are based on plane stress assumptions, SOLSH190 usually gives more accurate predictions when the shell is thick.

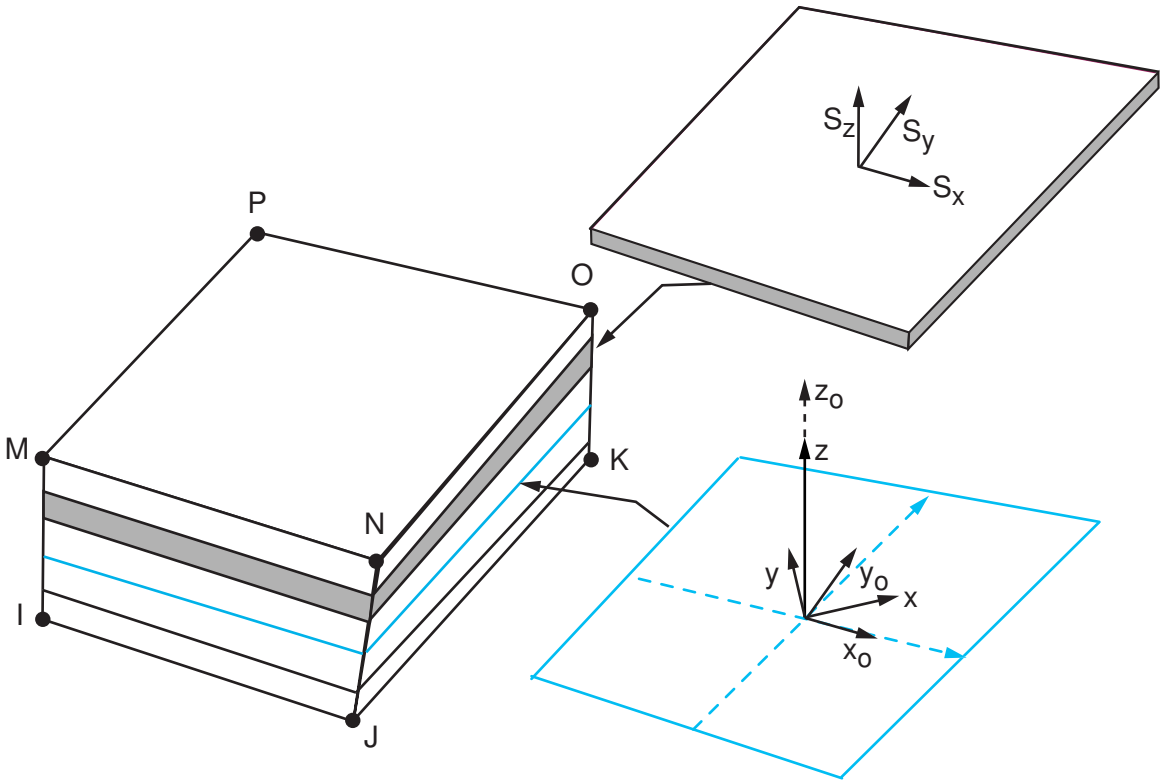
SOLSH190 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLSH190 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLSH190 Stress Output"*. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information.

Figure 2 SOLSH190 Stress Output



x_o = Element x-axis if ESYS is not supplied.
x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLSH190 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	*2*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [*5*]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	Y	Y
EPTH:EQV	Equivalent thermal strains [*5*]	Y	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains[*6*]	*1*	*1*
EPPL:EQV	Equivalent plastic strains [*5*]	*1*	*1*
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	*1*	*1*
EPCR:EQV	Equivalent creep strains [*5*]	*1*	*1*
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	*1*	*1*
NL:CREQ	Accumulated equivalent creep strain	*1*	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	*1*	*1*
NL:HPRES	Hydrostatic pressure	*1*	*1*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*1*
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	Y
Q13, Q23	Transverse shear forces (per unit length)	-	Y
LOCI:X, Y, Z	Integration point locations	-	*3*
SVAR:1, 2, ... , N	State variables	-	*4*

Name	Definition	O	R
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. Nonlinear solution, output only if the element has a nonlinear material
2. Available only at centroid as a ***GET** item
3. Available only if **OUTRES**,LOCI is used
4. Available only if the USERMAT subroutine and **TB**,STATE are used
5. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,NUXY); for plastic and creep this value is set at 0.5.
6. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 2, "SOLSH190 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLSH190 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SOLSH190 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 2 SOLSH190 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	4	3	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-
P4	SMISC	-	-	-	13	14	-	-	16	15
P5	SMISC	-	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	-	21	22	23	24
THICK	SMISC	27	-	-	-	-	-	-	-	-
N11	SMISC	28	-	-	-	-	-	-	-	-
N22	SMISC	29	-	-	-	-	-	-	-	-
N12	SMISC	30	-	-	-	-	-	-	-	-
M11	SMISC	31	-	-	-	-	-	-	-	-
M22	SMISC	32	-	-	-	-	-	-	-	-
M12	SMISC	33	-	-	-	-	-	-	-	-
Q13	SMISC	34	-	-	-	-	-	-	-	-
Q23	SMISC	35	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SOLSH190 Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in *Figure 1, "SOLSH190 Geometry"* or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- If you use the mixed u-P formulation (KEYOPT(6) = 1), you must use either the sparse solver (default) or the frontal solver.
- The maximum number of layers is 250.
- If the material of a layer is hyperelastic, the layer orientation angle has no effect.
- Using both hyperelastic and elastoplastic layers in the same element can produce unpredictable results and is not recommended.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

SOLSH190 Product Restrictions

There are no product-specific restrictions for this element.

SOLID191

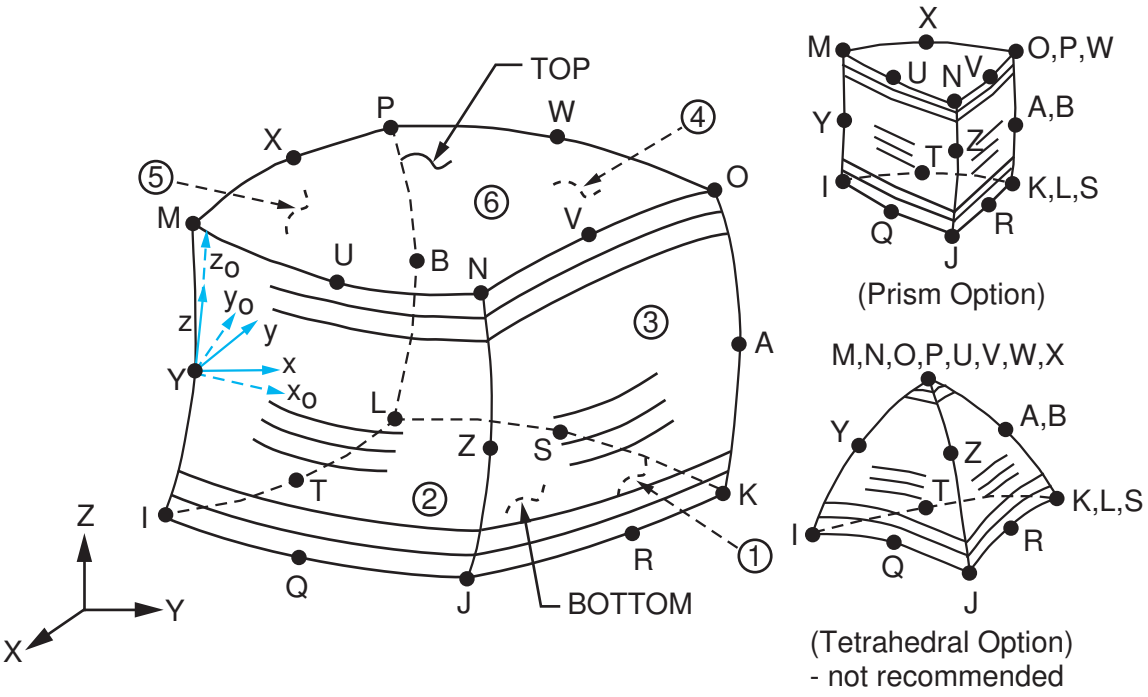
3-D 20-Node Layered Structural Solid

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

SOLID191 Element Description

SOLID191 is a layered version of the 20-node structural solid (SOLID95) designed to model layered thick shells or solids. The element allows up to 100 different material layers. If more than 100 layers are required, the elements may be stacked. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. SOLID191 has stress stiffening capabilities. Various printout options are also available. See SOLID191 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details. A similar element with 8 nodes is SOLID46. A similar element for shells is SHELL91.

Figure 1 SOLID191 Geometry



x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID191 Input Data

The geometry, node locations, and the coordinate system for this element are shown in SOLID191. The element is defined by 20 nodes, layer thicknesses, layer material direction angles, and orthotropic material properties. For any material, shear moduli GXZ and GYZ must be within a factor of 10,000 of each other.

To reorient the elements (after automatic meshing) you should use **EORIENT**. With **EORIENT**, you can make SOLID191 elements match an element whose orientation is as desired, or set the orientation to be as parallel as possible to a defined axis. The element z-axis is defined to be normal to the reference plane, which may be curved, as shown in *Figure 2, "SOLID191 Stress Output"*. The default element x-axis is the average projection of

side I-J and side M-N onto the reference plane. The orientation within the plane of the layers may be changed using **ESYS** in the same way it is used for shell elements as described in *Section 2.3: Coordinate Systems*.

The total number of layers (up to 100) must be specified (NL). If the properties of the layers are symmetric about the midthickness of the element (LSYM = 1), only half the properties, up to and including those of the middle layer (if any), need to be entered. Otherwise (LSYM = 0), the properties of all layers should be entered.

The material properties of each layer may be orthotropic in the plane of the element. The real constant MAT is used to define the layer material number instead of the element material number applied with the **MAT** command. MAT defaults to 1 if not input. The material X direction corresponds to the local layer x' direction.

Use the **BETAD** command to supply the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to supply the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

Each layer of the layered solid element may have a variable thickness (TK). The thickness is assumed to vary bilinearly over the area of the layer, with the thickness input at the corner node locations. If a layer has a constant thickness, only TK(I) need be input using positive values. If the thickness is not constant, all four corner thicknesses must be input. Zero thickness layers may be used to model dropped plies.

The layer thicknesses used are computed by scaling the input real constant thicknesses to be consistent with the thicknesses between the nodes. The node locations may imply that the layers are tilted or warped. However, the local coordinate system for each layer is effectively reoriented parallel to the reference plane, as shown in *Figure 2, "SOLID191 Stress Output"*. In this local right-handed system, the x'-axis is rotated an angle THETA (LN) (in degrees) from the element x-axis toward the element y-axis.

The total number of layers must be specified with the NL real constant as described in *SOLID191 Input Summary*. The real constants, material properties, and layer thicknesses are also described in *SOLID191 Input Summary*. Failure criteria are delineated in SHELL99.

The failure criteria selection is input in the data table [**TB**], as described in *Table 2.2, "Orthotropic Material Failure Criteria Data"*. Three predefined criteria are available and up to six user-defined criteria may be entered with user subroutines. See Failure Criteria in the *Theory Reference for ANSYS and ANSYS Workbench* for an explanation of the three predefined failure criteria. See *Guide to ANSYS User Programmable Features* for an explanation of user subroutines. Failure criteria may also be computed in POST1 (using the **FC** commands). All references to failure criteria as part of element output data are based only on the **TB** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers on SOLID191. Positive pressures act into the element.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-404 maximum), as shown in *Figure 1, "SHELL91 Geometry"*. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SOLID191 Input Summary*. A general description of element input is given in *SOLID191 Input Summary*.

SOLID191 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

See *Table 1, "SOLID191 Real Constants"* for a description of the real constants

Material Properties

Supply the following 13*NM properties where NM is the number of materials (maximum is NL):

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
PRXY, PRYZ, PRXZ, (or NUXY, NUYZ, NUXZ),
DENS, GXY, GYZ, GXZ, for each of the NM materials

Supply DAMP only once for the element (use **MAT** to assign material property set). REFT may be supplied once for the element, or may be assigned on a per layer basis. See the discussion in *SOLID191 Input Data* for more details.

Surface Loads

Pressure --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer *NL*(4*(*NL*+1) maximum)

Special Features

Plasticity (BISO, MISO, BKIN, MKIN, KINH, DP, ANISO)
Creep (CREEP)
Elasticity (MELAS)
Other material (USER)
Stress stiffening
Adaptive descent

Note

Items in parentheses refer to data tables associated with the **TB** command.

KEYOPT(1)

The maximum number of layers used by this element type for storage in the .ESAV and .0SAV files; default = 16. The first real constant (NL) must be no greater than the value you specify. The maximum number of layers may be no greater than 100.

KEYOPT(2)

Form of input:

0 --
Constant thickness layer input

1 --
Tapered thickness layer input

KEYOPT(3)
Material property usage:

0 --
Use material properties as given

1 --
Adjust material properties to give nonvarying values of $\sigma_z, \sigma_{xz}, \sigma_{yz}$ through thickness of element (similar to SOLID46)

KEYOPT(4)
Element coordinate system:

0 --
No user subroutines to define element coordinate system

4 --
Element x-axis located by user subroutine USERAN

5 --
Element x-axis located by user subroutine USERAN and layer x-axes located by user subroutine USANLY

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(5)
Element output per layer:

0 --
Print average results at layer face farthest from element nodal plane

1 --
Print average results at layer middle

2 --
Print average results at layer top and bottom

3 --
Print results, including failure criterion, at layer top and bottom 4 integration points and averages

4 --
Print results at layer top and bottom 4 corner points and averages

KEYOPT(7)
Extra element output:

0 --
Basic element printout

2 --
Nodal force printout in element coordinates (member forces)

KEYOPT(8)
Storage of layer data:

0 --
Store data for the following locations: bottom of the bottom layer, top of the top layer, and data for the maximum failure criteria layer

1 --
Store data for all layers

Caution

Volume of data stored may be excessive.

KEYOPT(10)
Failure criteria print summary:

0 --
Print summary of the maximum of all failure criteria

1 --
Print summary of all the failure criteria

Table 1 SOLID191 Real Constants

No.	Name	Description
If KEYOPT(2) = 0, supply the following 12+(3*NL) constants:		
1	NL	Number of layers (100 maximum)
2	LSYM	Layer symmetry key
3 ... 12	(Blank)	
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK	Layer thickness for layer 1
16	MAT	Material number for layer 2
17	THETA	x-axis rotation for layer 2
18	TK	Layer thickness for layer 2
19 ... 12+(3*NL)	MAT, THETA, etc.	Repeat MAT, THETA, and TK for each layer (up to NL layers)
If KEYOPT(2) = 1, supply the following 12+(6*NL) constants:		
1	NL	Number of layers (100 maximum)
2	LSYM	Layer symmetry key
3 ... 12	(Blank)	
13	MAT	Material number for layer 1
14	THETA	x-axis rotation for layer 1
15	TK(I)	Layer thickness at node I for layer 1
16	TK(J)	Layer thickness at node J for layer 1
17	TK(K)	Layer thickness at node K for layer 1
18	TK(L)	Layer thickness at node L for layer 1
19 ... 12+(6*NL)	MAT, THETA, etc.	Repeat MAT, THETA, TK(I), TK(J) , TK(K), and TK(L) for each layer (up to NL layers)

For more information on real constants and other input data, see SHELL91. For more information on failure criteria, please refer to *Section 2.2.2.12: Failure Criteria*.

SOLID191 Output Data

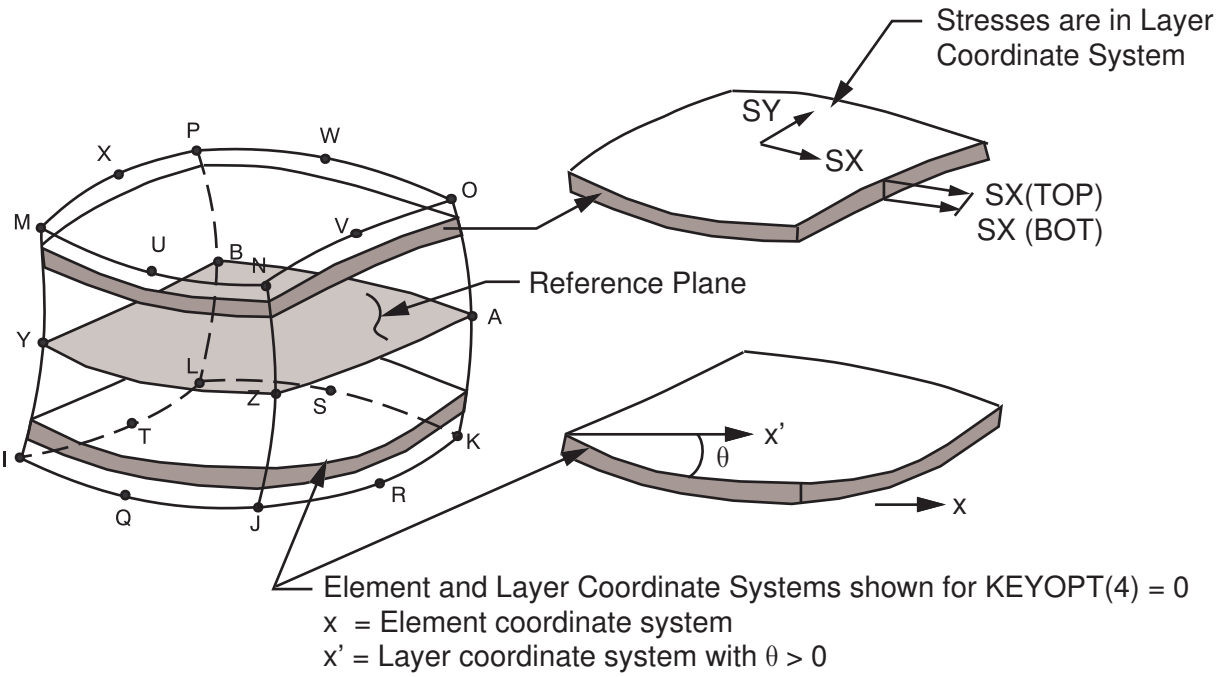
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SOLID191 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SOLID191 Stress Output"*.

The element stress directions correspond to the layer local coordinate directions. Various layer printout options are available. For integration point output, integration point 1 is nearest node I, 2 nearest J, 3 nearest K, and 4 nearest L. Failure criterion output is evaluated only at the in-plane integration points. (See SOLID191 in the *Theory Reference for ANSYS and ANSYS Workbench*.) KEYOPT(8) controls the amount of data output on the postdata file for processing with **LAYER** or **LAYERP26**. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

Figure 2 SOLID191 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SOLID191 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y

Name	Definition	O	R
VOLU:	Volume	Y	Y
TTOP, TBOT	Average temperatures at top and bottom faces	Y	Y
XC, YC, ZC	Location where results are reported	Y	*5*
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	*1*	Y
S:X, Y, Z, XY, YZ, XZ	Stresses (in layer local coordinates)	*2*	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains (in layer local coordinates)	*2*	Y
EPEL:EQV	Equivalent elastic strain (in layer local coordinates) [*6*]	*2*	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains (in layer local coordinates)	*2*	*2*
EPTH:EQV	Equivalent thermal strain (in layer local coordinates) [*6*]	*2*	*2*
LN	Layer number	Y	-
POS	Top (TOP), bottom (BOT), midthickness (MID) of layer	Y	-
LOC	Center location (AVG) [see KEYOPT(5) for control options]	Y	-
MAT	Material number of this layer	Y	-
THETA	Material direction angle for layer (THETA)	Y	-
AVE THICK	Average thickness of layer	Y	-
ACC AVE THICK	Accumulative average thickness (Thickness of element from layer 1 to this layer)	Y	-
AVE TEMP	Average temperature of layer	Y	-
NODE	Corner node number	Y	-
INT	Integration point number	Y	-
FC1, ..., FC6, FCMAX	Failure criterion values and maximum at each integration point	*3*	Y
FC	Failure criterion number (FC1 to FC6, FCMAX)	*3*	Y
VALUE	Maximum value for this criterion (if value exceeds 9999.999, 9999.999 will be printed)	*3*	Y
LN	Layer number where maximum occurs	*3*	Y
EPELF(X, Y, Z, XY, YZ, XZ)	Elastic strains (in layer local coordinates) causing the maximum value for this criterion in the element.	*3*	Y
SF(X, Y, Z, XY, YZ, XZ)	Stresses (in layer local coordinates) causing the maximum value for this criterion in the element.	*3*	Y
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SXZ interlaminar shear stress	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
LN1, LN2	Layer numbers which define location of maximum interlaminar shear stress (ILMAX)	*4*	Y
ILMAX	Maximum interlaminar shear stress (occurs between LN1 and LN2)	*4*	Y

1. If temperatures are present.
2. The strain and stress output is controlled with KEYOPT(5) and KEYOPT(8).

3. Summary of failure criteria calculation. Output of the elastic strains and/or stresses for each failure criterion and the maximum of all criteria (FCMAX).
4. Printed only if significant shear stress. If there is no clear maximum location, LN1 and LN2 are set to NL and zero, respectively.
5. Available only at centroid as a ***GET** item.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

The following notation is used in *Table 3, "SOLID191 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 2, "SOLID191 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 3 SOLID191 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	(2*i)-1	(2*NL)+1
ILSYZ	SMISC	(2*i)	(2*NL)+2
ILSUM	NMISC	(2*i)+5	(2*NL)+7
ILANG	NMISC	(2*i)+6	(2*NL)+8

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	(2*NL)+4	(2*NL)+3	(2*NL)+6	(2*NL)+5
P2	SMISC	(2*NL)+7	(2*NL)+8	-	-
P3	SMISC	-	(2*NL)+11	(2*NL)+12	-
P4	SMISC	-	-	(2*NL)+15	(2*NL)+16
P5	SMISC	(2*NL)+20	-	-	(2*NL)+19
P6	SMISC	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	M	N	O	P
P1	SMISC	-	-	-	-
P2	SMISC	(2*NL)+10	(2*NL)+9	-	-
P3	SMISC	-	(2*NL)+14	(2*NL)+13	-
P4	SMISC	-	-	(2*NL)+18	(2*NL)+17
P5	SMISC	(2*NL)+21	-	-	(2*NL)+22

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	M	N	O	P
P6	SMISC	(2*NL)+23	(2*NL)+24	(2*NL)+25	(2*NL)+26

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FCMAX (over all layers)	NMISC	1
VALUE	NMISC	2
LN	NMISC	3
ILMAX	NMISC	4
LN1	NMISC	5
LN2	NMISC	6
ILSUM (between layers i-1 and i)	NMISC	2i+7
ILANG	NMISC	2i+8
FCMAX (at layer i)	NMISC	(2*(NL+i))+7
VALUE (at layer i)	NMISC	(2*(NL+i))+8
FC	NMISC	(4*NL)+8+15(N-1)+1
VALUE	NMISC	(4*NL)+8+15(N-1)+2
LN	NMISC	(4*NL)+8+15(N-1)+3
EPELFX	NMISC	(4*NL)+8+15(N-1)+4
EPELFY	NMISC	(4*NL)+8+15(N-1)+5
EPELFZ	NMISC	(4*NL)+8+15(N-1)+6
EPELFXY	NMISC	(4*NL)+8+15(N-1)+7
EPELFYZ	NMISC	(4*NL)+8+15(N-1)+8
EPELFXZ	NMISC	(4*NL)+8+15(N-1)+9
SFX	NMISC	(4*NL)+8+15(N-1)+10
SFY	NMISC	(4*NL)+8+15(N-1)+11
SFZ	NMISC	(4*NL)+8+15(N-1)+12
SFXY	NMISC	(4*NL)+8+15(N-1)+13
SFYZ	NMISC	(4*NL)+8+15(N-1)+14
SFXZ	NMISC	(4*NL)+8+15(N-1)+15

Note

The i in *Table 3, "SOLID191 Item and Sequence Numbers"* (where i = 1, 2, 3, ..., NL) refers to the layer number of the element. NL is the maximum layer number as input for real constant NL ($1 \leq NL \leq 100$). N is the failure number as stored on the results file in compressed form, e.g., only those failure criteria requested will be written to the results file. For example, if only the maximum strain and the Tsai-Wu failure criteria are requested, the maximum strain criteria will be stored first (N = 1) and the Tsai-Wu failure criteria will be stored second (N = 2). In addition, if more than one criteria is requested, the maximum value over all criteria is stored last (N = 3 for this example).

SOLID191 Assumptions and Restrictions

- Zero volume elements are not allowed. Usually, this occurs if the elements are not numbered properly.

- All elements must have at least eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see *Section 2.9: Triangle, Prism and Tetrahedral Elements*).
- A tetrahedron shape is also available.
- No slippage is assumed between the element layers. All material orientations are parallel to the reference plane.
- It has been observed that large differences (factors greater than 1000) between different moduli of the same material can cause large differences between the equation solver maximum and minimum pivots, and can even cause "NEGATIVE PIVOT..." messages to appear. If this occurs, you should consider whether the material properties are realistic. The element matrices are reformed every iteration unless option 1 of **KUSE** is active.
- Because of the nature of the in-plane integration, spurious rigid body motions are possible when using SOLID191. These spurious motions are not seen when using meshes of at least two elements in at least two different directions.
- Interlaminar shear stresses for SHELL91 and SHELL99 shell elements are based on the premise that there are no interlaminar (transverse) shear stresses at the outer surface of the shell. This assumption cannot be used for a solid element. Thus, SOLID191 has two forms of shear stress calculations:
  - Those based on nodal forces (labeled "average transverse shear stress components").
  - Those based on the strain-displacement relationships, averaged across layers when applicable (labeled "maximum interlaminar shear stress").

  Neither one of these is exact, but ideally they will agree with each other. In both situations, the given values are averages, which will be less than the peak value. The differences between the average and the peak will be small in most cases; however, differences up to a factor of two have been seen.
- Additional elements in the thickness direction will improve the interlaminar shear stress calculation.
- When brick (rectangular prism) elements are used, both calculations result in constant stresses over the volume of the element.
- In all cases, the values are constant in the plane of the layer and may, therefore, be thought of as centroidal values. Hence, one should consider using solid-to-solid submodeling to get accurate shear stress values at a free edge. These shear stresses are discussed further with SOLID46 in the *Theory Reference for ANSYS and ANSYS Workbench*. The *Structural Analysis Guide* contains additional information on composite elements.

SOLID191 Product Restrictions

There are no product-specific restrictions for this element.

INTER192

2-D 4-Node Gasket

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

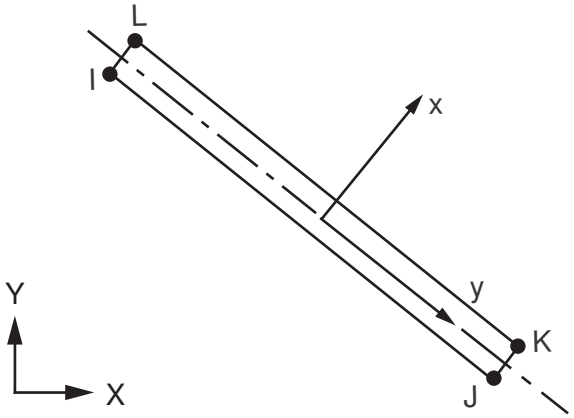
INTER192 Element Description

INTER192 is a 2-D 4-node linear interface element used for the 2-D modeling of structural assemblies. When used in conjunction with 2-D linear structural elements (PLANE42, VISCO106, and PLANE182), INTER192 is used to simulate gasket joints. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Gasket Material and INTER192 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on the gasket capability in ANSYS.

Figure 1 INTER192 Geometry



INTER192 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER192 Geometry"*. The element geometry is defined by 4 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J; and the top line is defined by nodes K, L. The element connectivity is defined as I, J, K, L. This element has 2 integration points. The Gauss integration scheme is used for the numerical integration.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See the *Section 2.1: Element Input* section in the *Elements Reference* for a general description of element input.

INTER192 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
DAMP, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features
Gasket material associated with **TB**,GASKET.

Note

See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER192 Output Data

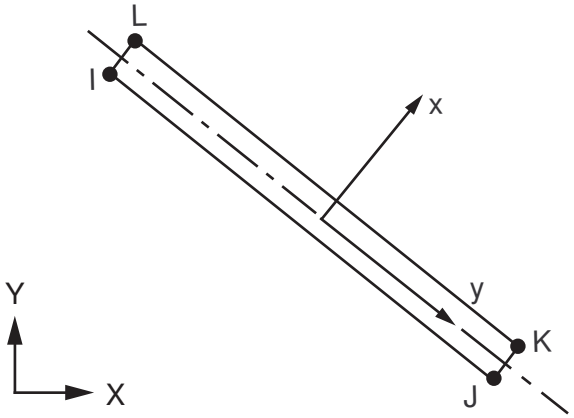
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in *Table 1, "INTER192 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 2,"INTER192 Stress Output"*. See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER192 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER192 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
GKS:X, (XY)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY)	Total closure	Y	Y
GKDI:X, (XY)	Total inelastic closure	Y	Y
GKTH:X, (XY)	Thermal closure	Y	Y

INTER192 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER192 Product Restrictions

There are no product-specific restrictions on this element.

INTER193

2-D 6-Node Gasket

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

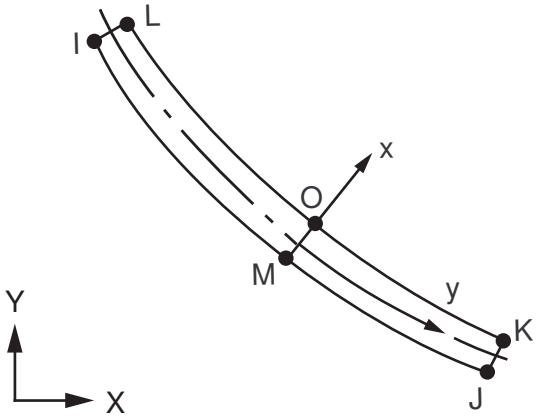
INTER193 Element Description

INTER193 is a 2-D 6-node quadratic interface element used for the 2-D modeling of structural assemblies. When used in conjunction with 2-D quadratic structural elements (PLANE82, VISCO88 and PLANE183), INTER193 is used to simulate gasket joints. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Gasket Material and INTER193 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on the gasket capability in ANSYS.

Figure 1 INTER193 Geometry



INTER193 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER193 Geometry"*. The element geometry is defined by 6 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J, M; and the top line is defined by nodes K, L, O. The element connectivity is defined as I, J, K, L, M, O. This element has 2 integration points. Dropping mid side nodes M or O is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER193 Input Summary

Nodes
I, J, K, L, M, O

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
DAMP, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(O)

Special Features
Gasket material associated with **TB**,GASKET.

Note

See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER193 Output Data

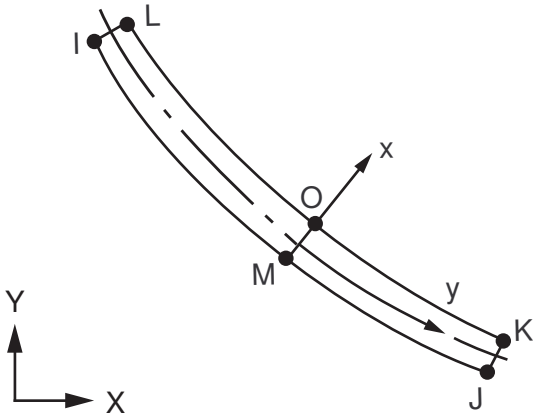
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in *Table 1, "INTER193 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 2,"INTER193 Stress Output"*. See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER193 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER193 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, O	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(O)	-	Y
GKS:X, (XY)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY)	Total closure	Y	Y
GKDI:X, (XY)	Total inelastic closure	Y	Y
GKTH:X, (XY)	Thermal closure	Y	Y

INTER193 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER193 Product Restrictions

There are no product-specific restrictions on this element.

INTER194

3-D 16-Node Gasket

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

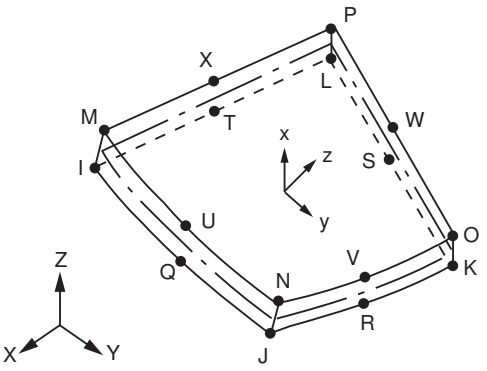
INTER194 Element Description

INTER194 is a 3-D 16-node quadratic interface element. When used in conjunction with 3-D quadratic structural elements (VISCO89, SOLID92, SOLID95, SOLID96, SOLID186, and SOLID187), INTER194 is used to simulate gasket joints. It is defined by 16 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Gasket Material and INTER194 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on the gasket capability in ANSYS.

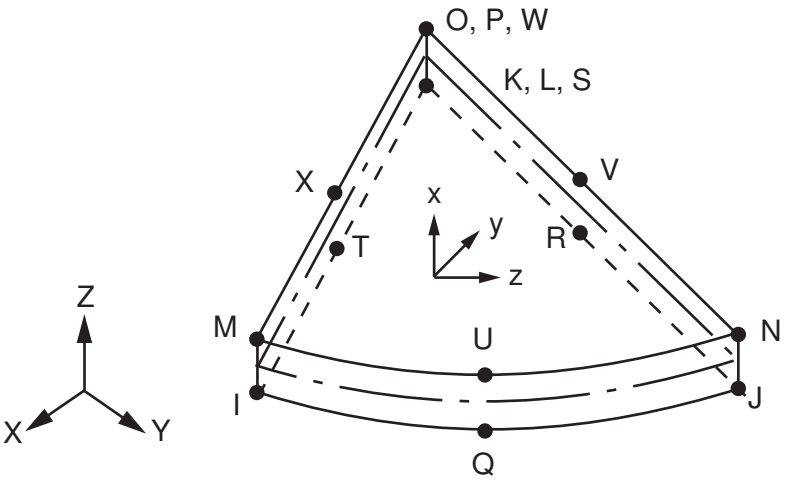
Figure 1 INTER194 Geometry



INTER194 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER194 Geometry"*. The element geometry is defined by 16 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L, Q, R, S, T; and the top surface is defined by nodes, M, N, O, P, U, V, W, X. As shown, the element connectivity is defined as I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X. The element is degenerated to a wedge (prism) element, when K=L=S and O=P=W, as shown in *Figure 2, "INTER194 3-D 16-Node Degenerated Quadratic Interface"*.

Figure 2 INTER194 3-D 16-Node Degenerated Quadratic Interface



For the degenerated element, 3 integration points are used for numerical integration. The degenerated element can be used in conjunction with the 10-node solid tetrahedral elements SOLID92 and SOLID187. Dropping any or some of midside nodes, Q, R, S, T, U, V, W, X is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER194 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
DAMP, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L) T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)

Special Features
Gasket material associated with **TB**,GASKET.

Note

See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation

INTER194 Output Data

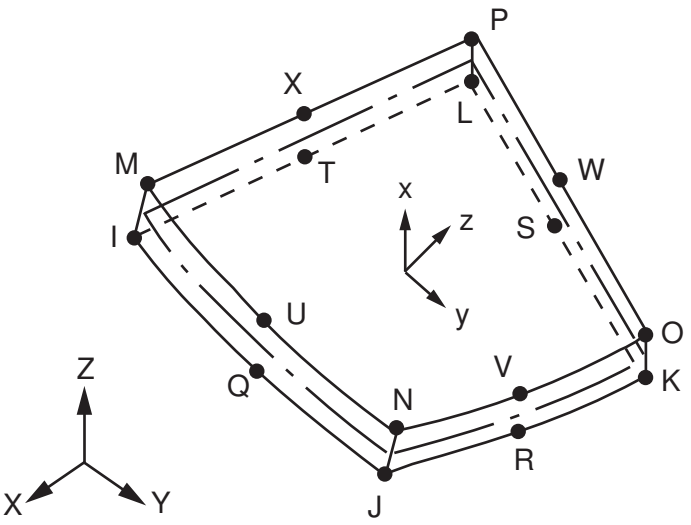
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in *Table 1, "INTER194 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 3, "INTER194 Stress Output"*. See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 3 INTER194 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER194 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y

Name	Definition	O	R
NODES	Node connectivity - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)	-	Y
GKS:X, (XY, XZ)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY, XZ)	Total closure	Y	Y
GKDI:X, (XY, XZ)	Total inelastic closure	Y	Y
GKTH:X, (XY, XZ)	Thermal closure	Y	Y

INTER194 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER194 Product Restrictions

There are no product-specific restrictions on this element.

INTER195

3-D 8-Node Gasket

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

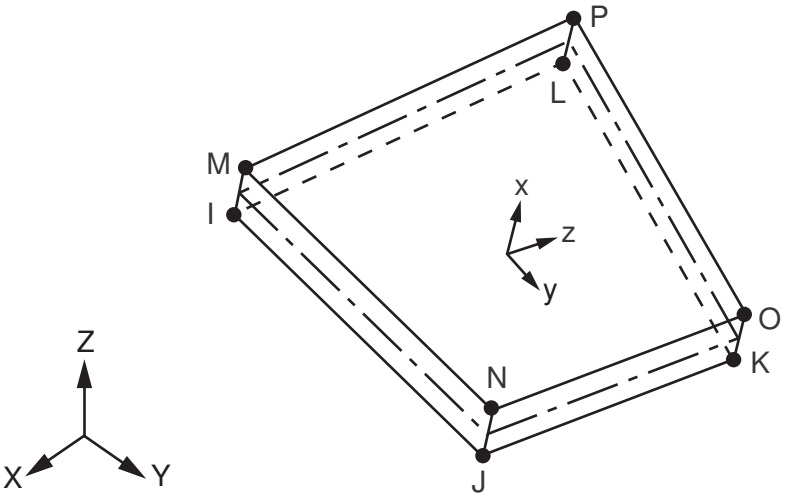
INTER195 Element Description

INTER195 is a 3-D 8-node linear interface element. When used in conjunction with 3-D linear structural elements (SOLID45, SOLID46, SOLID62, SOLID65, SOLID185, and SOLSH190), INTER195 is used to simulate gasket joints. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Gasket Material and INTER195 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on this ANSYS capability.

Figure 1 INTER195 Geometry



INTER195 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER195 Geometry"*. The element geometry is defined by 8 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L; and the top surface is defined by nodes, M, N, O, P. As shown, the element connectivity is defined as I, J, K, L, M, N, O, P.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER195 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
DAMP, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features
Gasket material associated with **TB**,GASKET.

Note

See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation

INTER195 Output Data

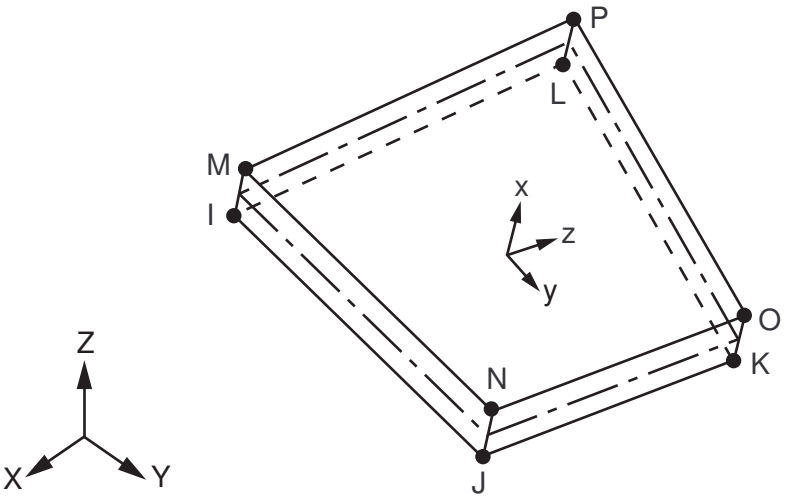
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in *Table 1, "INTER195 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 2, "INTER195 Stress Output"*. See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER195 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER195 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
GKS:X, (XY, XZ)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY, XZ)	Total closure	Y	Y
GKDI:X, (XY, XZ)	Total inelastic closure	Y	Y
GKTH:X, (XY, XZ)	Thermal closure	Y	Y

INTER195 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER195 Product Restrictions

There are no product-specific restrictions on this element.

MESH200

Meshing Facet

MP ME ST PR PRN <> <> <> EM EH DY PP <>
Product Restrictions

MESH200 Element Description

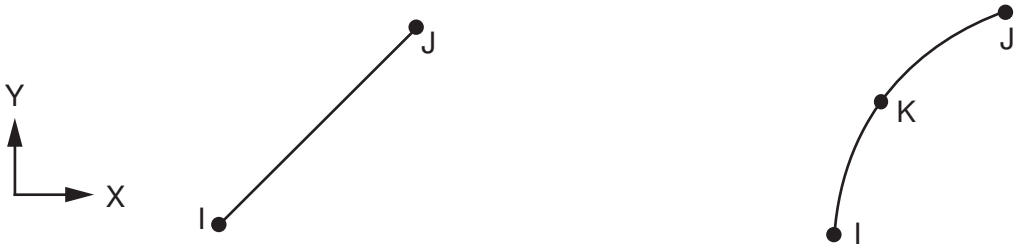
MESH200 is a "mesh-only" element, contributing nothing to the solution. This element can be used for the following types of operations:

- Multistep meshing operations, such as extrusion, that require a lower dimensionality mesh be used for the creation of a higher dimensionality mesh
- Line-meshing in 2-D or 3-D space with or without midside nodes,
- Area-meshing or volume-meshing in 3-D space with triangles, quadrilaterals, tetrahedra, or bricks, with or without midside nodes.
- Temporary storage of elements when the analysis physics has not yet been specified.

MESH200 may be used in conjunction with any other ANSYS element types. After it is no longer needed, it can be deleted (cleared), or can be left in place. Its presence will not affect solution results.

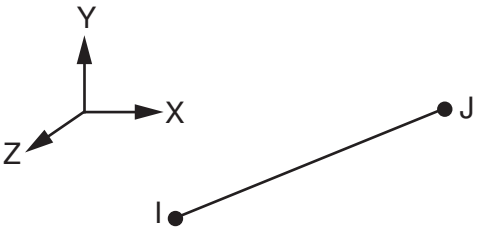
MESH200 elements can be changed into other element types using **EMODIF**.

Figure 1 MESH200 Geometry

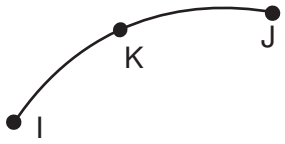


KEYOPT (1) = 0
2-D line with 2 nodes

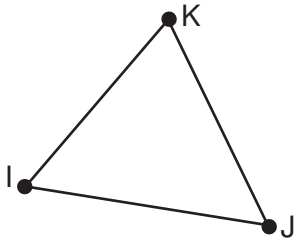
KEYOPT (1) = 1
2-D line with 3 nodes



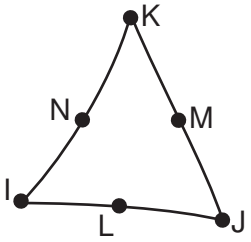
KEYOPT (1) = 2
3-D line with 2 nodes



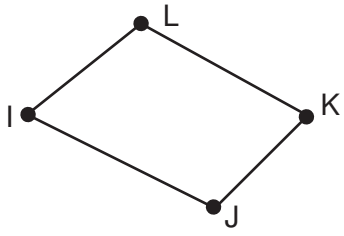
KEYOPT (1) = 3
3-D line with 3 nodes



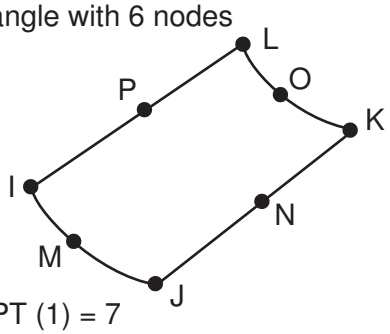
KEYOPT (1) = 4
3-D triangle with 3 nodes



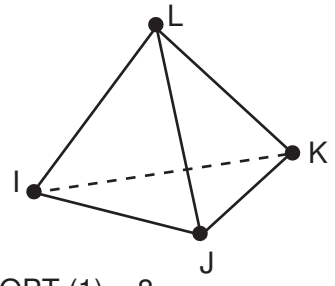
KEYOPT (1) = 5
3-D triangle with 6 nodes



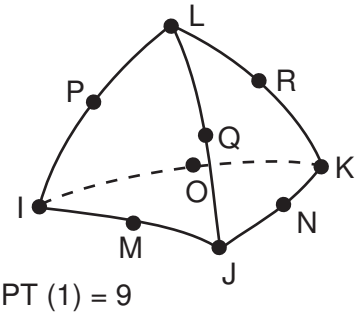
KEYOPT (1) = 6
3-D quadrilateral with 4 nodes



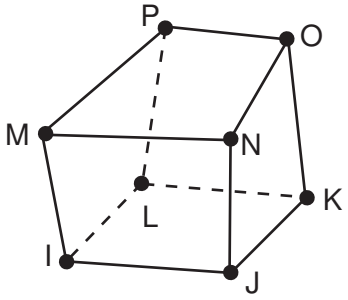
KEYOPT (1) = 7
3-D quadrilateral with 8 nodes



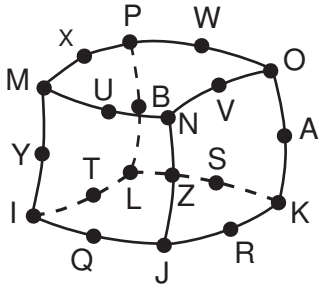
KEYOPT (1) = 8
tetrahedron with 4 nodes



KEYOPT (1) = 9
tetrahedron with 10 nodes



KEYOPT (1) = 10
brick with 8 nodes



KEYOPT (1) = 11
brick with 20 nodes

MESH200 Input Data

The permissible geometry and node locations for this element are shown in *Figure 1, "MESH200 Geometry"*. The element is defined by two to twenty nodes. It has no degrees of freedom, material properties, real constants, or loadings.

MESH200 Input Summary summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

MESH200 Input Summary

Nodes

I, J if KEYOPT (1) = 0, 2-D line with 2 nodes
I, J, K if KEYOPT (1) = 1, 2-D line with 3 nodes
I, J if KEYOPT (1) = 2, 3-D line with 2 nodes
I, J, K if KEYOPT (1) = 3, 3-D line with 3 nodes
I, J, K if KEYOPT (1) = 4, 3-D triangle with 3 nodes
I, J, K, L, M, N if KEYOPT (1) = 5, 3-D triangle with 6 nodes
I, J, K, L if KEYOPT (1) = 6, 3-D quadrilateral with 4 nodes
I, J, K, L, M, N, O, P if KEYOPT (1) = 7, 3-D quadrilateral with 8 nodes
I, J, K, L if KEYOPT (1) = 8, tetrahedron with 4 nodes
I, J, K, L, M, N, O, P, Q, R if KEYOPT (1) = 9, tetrahedron with 10 nodes
I, J, K, L, M, N, O, P if KEYOPT (1) = 10, brick with 8 nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B if KEYOPT (1) = 11, brick with 20 nodes

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Element shape and number of nodes:

0 --
2-D line with 2 nodes

1 --
2-D line with 3 nodes

2 --
3-D line with 2 nodes

3 --
3-D line with 3 nodes

4 --
3-D triangle with 3 nodes

5 --
3-D triangle with 6 nodes

6 --
3-D quadrilateral with 4 nodes

7 --
3-D quadrilateral with 8 nodes

8 --
tetrahedron with 4 nodes

9 --
tetrahedron with 10 nodes

10 --
brick with 8 nodes

11 --
brick with 20 nodes

KEYOPT(2)
Element shape testing:

0 --
Shape testing is done (default)

1 --
No shape testing is done for this element

MESH200 Output Data

This element has no output data.

MESH200 Assumptions and Restrictions

- When this element is a triangle or quadrilateral, it is shape-tested in the same manner as an equivalent "non-structural shell". When it is a tetrahedron or brick, it is shape-tested like a SOLID92, SOLID45, or SOLID95. This is so that meshing will work to create well-shaped elements. If KEYOPT(2) = 1, no shape testing is done for this element type.
- MESH200 elements may not be active during result contour plotting (**/POST1**, **PLNSOL**, or **PLESOL**). The elements are automatically unselected during either operation.

MESH200 Product Restrictions

There are no product-specific restrictions for this element.

FOLLW201

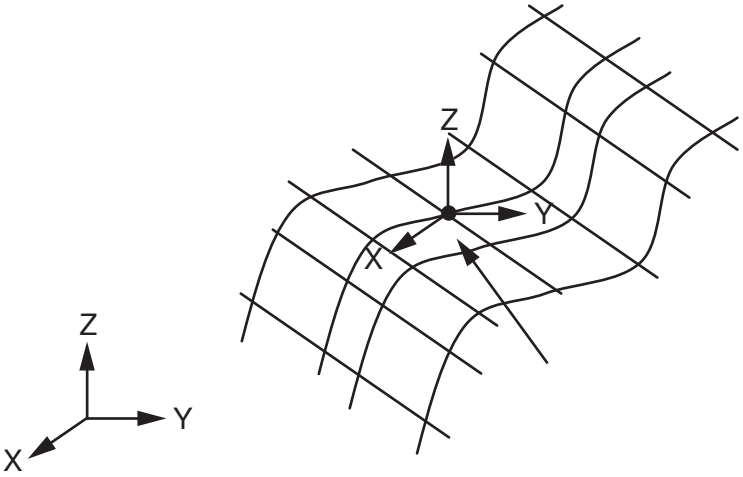
Follower Load

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

FOLLW201 Element Description

FOLLW201 is a one-node 3-D element that can be overlaid onto an existing node with physical rotation degrees of freedom. The element specifies external forces and moments which follow the deformation of a structure in a nonlinear analysis. FOLLW201 contributes follower load stiffness terms in a geometrically nonlinear analysis (**NLGEOM**,ON).

Figure 1 FOLLW201 Geometry



FOLLW201 overlaid on a node shared by shell or beam elements. The element has two faces: face 1 for specifying magnitude of force and face 2 for specifying magnitude of moment.

FOLLW201 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "FOLLW201 Geometry"*. The element is defined by a single node. The node has three translational and rotational degrees of freedom each. The element may be defined only at those nodes which are associated with structural elements having three translational and rotational degrees of freedom; a singularity will result if the element is used in any other way.

Real constants of the element specify the direction of the force/moment vectors, and the element load command **SFE** specifies the magnitude of force/moment.

Element loads are described in *Section 2.8: Node and Element Loads*. The vectors defined by real constants will evolve with deformation (follow the displacements) in a geometrically nonlinear analysis.

KEYOPT(1) provides a means of specifying complex point loads via an element specification. You can consider it as a nodal point equivalent of surface-effect elements (such as SURF154). When KEYOPT(1) = 1, the direction of the load is not updated. The element is also renamed to (and appears in the output as) CLOAD201. This option is intended primarily for use in the ANSYS Workbench interface.

The KEYOPT(2) = 1 setting, when used *along with* KEYOPT(1) = 1, enables use of the CLOAD201 element on nodes that may or may not have active rotation degrees of freedom. This option is intended primarily for use in the ANSYS Workbench interface.

With the exception of follower load effects, the element contributes nothing to the stiffness matrix. By default, follower load stiffness effects are included in a geometrically nonlinear analysis. The stiffness contribution is usually unsymmetrical and may require an unsymmetrical solution option (**NROPT**,UNSYM).

FOLLW201 Input Summary contains a summary of the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

FOLLW201 Input Summary

Nodes
I

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

FX - Cosine of the angle between force vector and global X direction
FY - Cosine of the angle between force vector and global Y direction
FZ - Cosine of the angle between force vector and global Z direction
MX - Cosine of the angle between moment vector and global X direction
MY - Cosine of the angle between moment vector and global Y direction
MZ - Cosine of the angle between moment vector and global Z direction

Material Properties
None

Surface Loads

face 1 (force magnitude)
face 2 (moment magnitude)

Body Loads
None

Special Features

Large deflection
Birth and death

KEYOPT(1)
Direction load:

0 --
Updated direction load (default)

1 --
Constant direction load

Note

This option is intended primarily for use in the ANSYS Workbench interface.

KEYOPT(2)
Degrees of freedom control:

0 --
Use all (UX, UY, UZ, ROTX, ROTY, ROTZ) degrees of freedom (default)

1 --
Use UX, UY, and UZ degrees of freedom only

Note

This option is intended primarily for use in the ANSYS Workbench interface.

FOLLW201 Output Data

The Element Outputs consist of updated direction cosines of the force/moment vectors as Miscellaneous quantities (SMISC). No other output is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

The following table lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information.

Name
output quantity as defined above

Item
predetermined item label for **ETABLE** command

I
sequence number for data at node I

Table 1 FOLLW201 Item and Sequence Numbers for the ETABLE and ESOL Commands

Name	Item	Location
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6

FOLLW201 Assumptions and Restrictions

- The element must be overlaid on a node having existing physical stiffness contributions (from other shell or beam elements).
- Follower load stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF), which is equivalent to the normal specification of forces and moments (**F**).

- Follower load effects are nonconservative. They often introduce dynamics instability issues (such as flutter) which may cause convergence difficulties.

FOLLW201 Product Restrictions

There are no product-specific restrictions for this element.

INTER202

2-D 4-Node Cohesive Zone

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

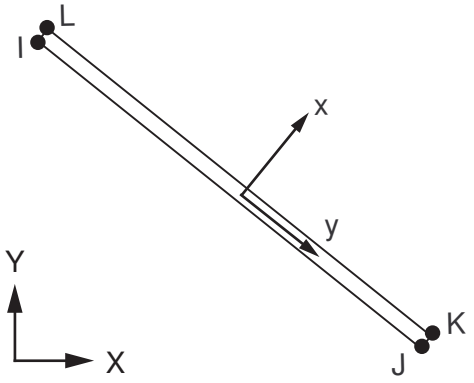
INTER202 Element Description

INTER202 is a 2-D 4-node linear interface element used to model 2-D structural assemblies. When used in conjunction with 2-D linear structural elements (PLANE42 and PLANE182), INTER202 simulates the interface surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself, that are initially coincident. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Cohesive Zone Material Model and INTER202 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details on the interface failure/delamination capability in ANSYS.

Figure 1 INTER202 Geometry



INTER202 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER202 Geometry"*. The element geometry is defined by 4 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J; and the top line is defined by nodes K, L. The element connectivity is defined as I, J, K, L. This element has 2 integration points. The Gauss integration scheme is used for the numerical integration.

INTER202 is used to simulate the separation along an interface defined by this element. At the outset of your simulation, nodes I,L and J,K are coincident, both with each other. and with the corresponding nodes in the adjacent structural elements. The subsequent separation of the adjacent elements (usually defined contiguously as components) is represented by an increasing displacement between the nodes within this element.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See the *Section 2.1: Element Input* section in the *Elements Reference* for a general description of element input.

INTER202 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Note

The temperature is used only to evaluate the material properties.

Special Features
Interface material associated with **TB**,CZM.

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER202 Output Data

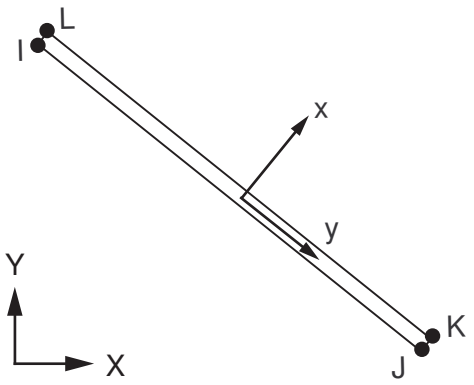
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in *Table 1, "INTER202 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane, as illustrated in *Figure 2, "INTER202 Stress Output"*. See Cohesive Zone Model in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER202 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER202 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
SS:X, (XY)	Interface Traction (stress)	Y	Y
SD:X, (XY)	Interface Separation (displacement)	Y	Y

INTER202 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER202 Product Restrictions

There are no product-specific restrictions on this element.

INTER203

2-D 6-Node Cohesive Zone

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

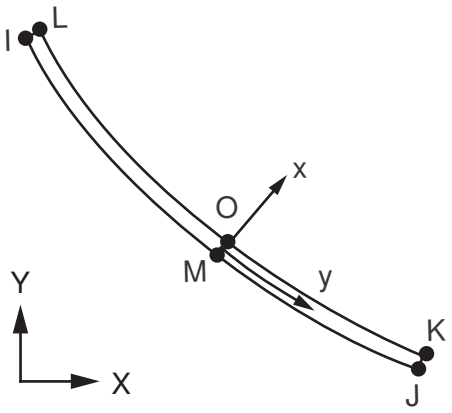
INTER203 Element Description

INTER203 is a 2-D 6-node quadratic interface element used for the 2-D modeling of structural assemblies. When used in conjunction with 2-D quadratic structural elements (PLANE82 and PLANE183), INTER203 simulates the interface surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself, that are initially coincident. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Cohesive Zone Material Model and INTER203 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details on the interface failure/delamination capability in ANSYS.

Figure 1 INTER203 Geometry



INTER203 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER203 Geometry"*. The element geometry is defined by 6 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J, M; and the top line is defined by nodes K, L, O. This element has 2 integration points. Dropping mid side nodes M or O is not permitted.

INTER203 is used to simulate a separation along an interface defined by this element. At the outset of your simulation, nodes I,L, nodes M,O and nodes J,K are coincident, with each other, and with the corresponding nodes in the adjacent structural elements. The subsequent separation of the adjacent elements (usually defined contiguously as components) is represented by an increasing displacement between the initially coincident nodes within this element.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER203 Input Summary

Nodes
I, J, K, L, M, , O

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(O)

Note

Temperature is used only to evaluate the material properties.

Special Features
Interface material associated with **TB**,CZM.

Note

See Cohesive Zone Material Model and INTER203 in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER203 Output Data

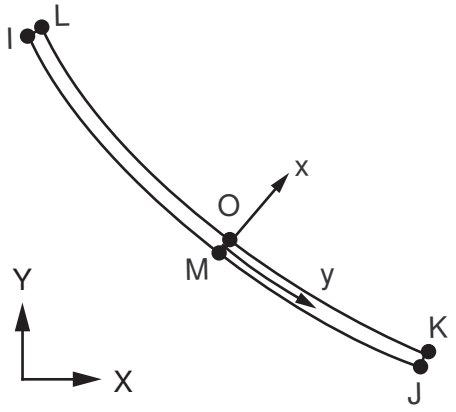
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in *Table 1,"INTER203 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 2, "INTER203 Stress Output"*. See Cohesive Zone Material Model and INTER203 in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER203 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER203 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, O	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(O)	-	Y
SS:X, (XY)	Interface Traction (Stress)	Y	Y
SD:X, (XY)	Interface Sepatation	Y	Y

INTER203 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER203 Product Restrictions

There are no product-specific restrictions on this element.

INTER204

3-D 16-Node Cohesive Zone

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

INTER204 Element Description

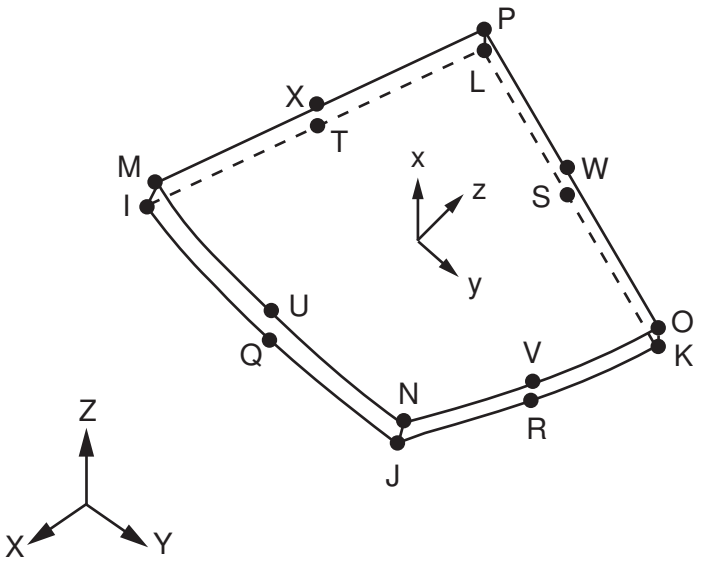
INTER204 is a 3-D 16-node quadratic interface element. When used in conjunction with 3-D quadratic structural elements (SOLID92, SOLID95, SOLID186, and SOLID187), INTER204 simulates an interface between two surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself. The nodes are initially coincident.

INTER204 is defined by 16 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Cohesive Zone Material Model and INTER204 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details on the interface failure/delamination capability in ANSYS.

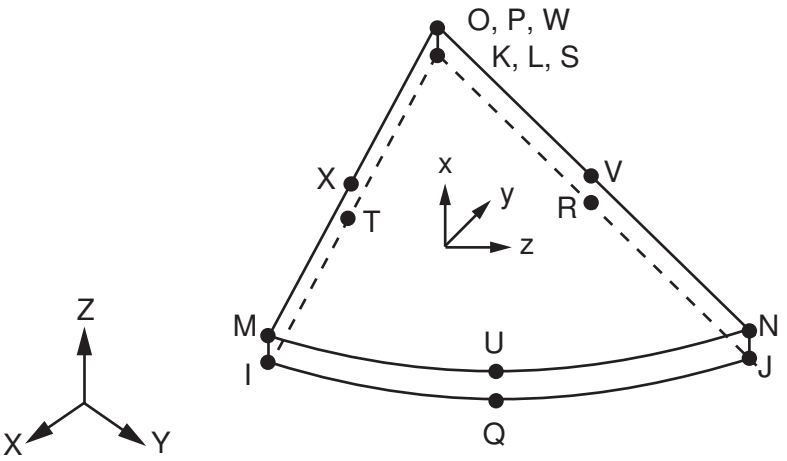
Figure 1 INTER204 Geometry



INTER204 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER204 Geometry"*. The element geometry is defined by 16 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L, Q, R, S, T; and the top surface is defined by nodes, M, N, O, P, U, V, W, X. The element may be degenerated to a wedge (prism) element, by setting nodes K=L=S and O=P=W, as shown in *Figure 2, "INTER204 3-D 16-Node Degenerated Quadratic Interface"*.

Figure 2 INTER204 3-D 16-Node Degenerated Quadratic Interface



For the degenerated element, 3 integration points are used for numerical integration. The degenerated element can be used in conjunction with the 10-node solid tetrahedral elements SOLID92 and SOLID187. Dropping any or some of the midside nodes, Q, R, S, T, U, V, W, X is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER204 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Body Loads

Temperatures --
T(I), T(J), T(K), T(L) T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)

Special Features
Cohesive zone material associated with **TB**,CZM.

Note

See Cohesive Zone Material Model and INTER204 in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

INTER204 Output Data

The solution output associated with the element is in two forms:

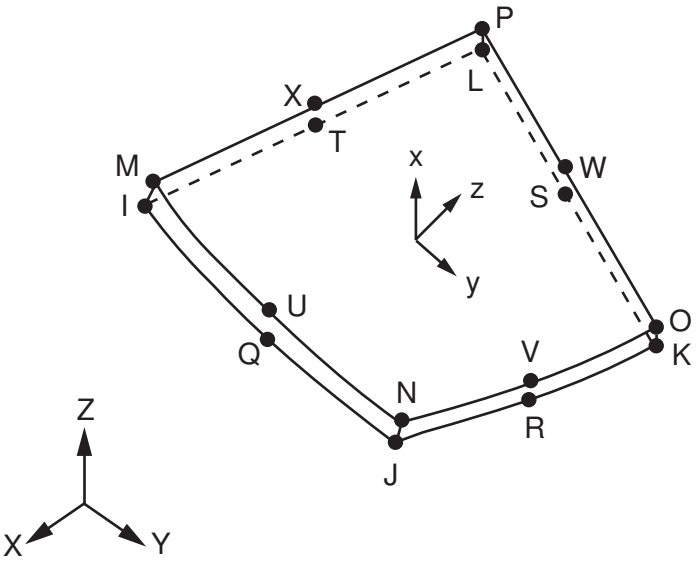
- Nodal items such as nodal displacements are included in the overall nodal solution.

- Element items such as tractions and separations are element outputs as shown in *Table 1,"INTER204 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 3,"INTER204 Stress Output"*. See Cohesive Zone Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 3 INTER204 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER204 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)	-	Y
SS:X, (XY, XZ)	Interface traction (stress)	Y	Y
SD:X, (XY, XZ)	Interface Separation	Y	Y

INTER204 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER204 Product Restrictions

There are no product-specific restrictions on this element.

INTER205

3-D 8-Node Cohesive Zone

MP ME ST <> <> <> <> <> <> <> <> PP <>
Product Restrictions

INTER205 Element Description

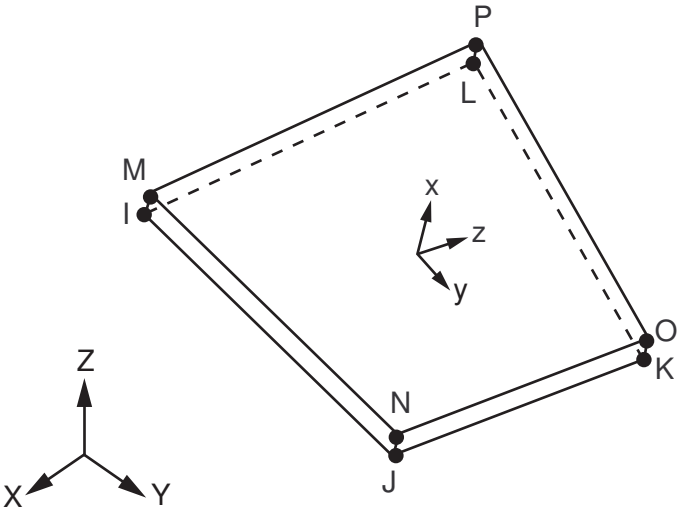
INTER205 is a 3-D 8-node linear interface element. When used in conjunction with 3-D linear structural elements (SOLID45, SOLID185, and SOLSH190), INTER205 simulates an interface between two surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself. The nodes are initially coincident.

It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Cohesive Zone Material Model and INTER204 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details on the interface failure/delamination capability in ANSYS.

Figure 1 INTER205 Geometry



INTER205 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in *Figure 1, "INTER205 Geometry"*. The element geometry is defined by 8 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L; and the top surface is defined by nodes, M, N, O, P.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See *Section 2.1: Element Input* in the *Elements Reference* for a general description of element input.

INTER205 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Note

The temperature is used only to evaluate the material properties.

Special Features
Cohesive zone material associated with **TB**,CZM.

Note

See Cohesive Zone Material Model and INTER205 in the *Theory Reference for ANSYS and ANSYS Workbench* for details on the material model.

INTER205 Output Data

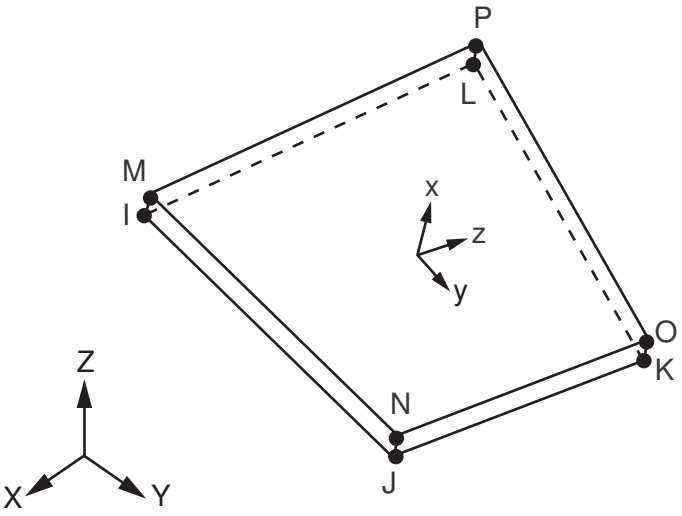
The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in *Table 1,"INTER205 Element Output Definitions"*.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in *Figure 2,"INTER205 Stress Output"*. See Gasket Material in the *Theory Reference for ANSYS and ANSYS Workbench* for details.

A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

Figure 2 INTER205 Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 INTER205 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
SS:X, (XY, XZ)	Interface traction (stress)	Y	Y
SD:X, (XY, XZ)	Interface separation	Y	Y

INTER205 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER205 Product Restrictions

There are no product-specific restrictions on this element.

SHELL208

2-Node Finite Strain Axisymmetric Shell

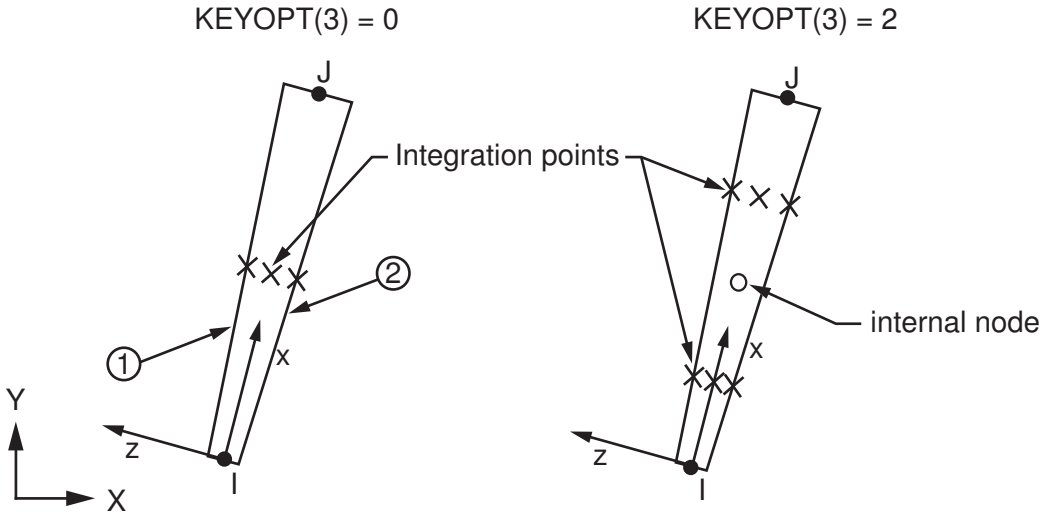
MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL208 Element Description

SHELL208 is suitable for modeling thin to moderately thick axisymmetric shell structures, such as oil tanks, pipes, and cooling towers. It is a 2-node element with three degrees of freedom at each node: translations in the x, and y directions, and rotation about the z-axis. A fourth translational degree of freedom in z direction can be included to model uniform torsion (KEYOPT(2) = 1). When membrane option is used, the rotational degree of freedom is excluded. You can add an extra internal node using KEYOPT(3) = 2. (SHELL209 incorporates this extra node by default.) Using SHELL208 allows you to account for large strain effects, transverse shear deformation, hyperelasticity and layers in your models. This element is intended to model finite strain with pure axisymmetric displacements; transverse shear strains are assumed to be small.

SHELL208 can be used for layered applications for modeling laminated composite shells or sandwich construction. See SHELL208 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL208 Geometry



SHELL208 Input Data

Figure 1, "SHELL208 Geometry" shows the geometry, node locations, and element coordinate system for SHELL208. The element is defined by two nodes. For material property labels, the local x-direction corresponds to the meridional direction of the shell element. The local y-direction is the circumferential. The local z-direction corresponds to the through-the-thickness direction. Element formulation is based on logarithmic strain and true stress measures. Element kinematics allows for finite membrane strains (stretching). However, the curvature changes within an increment are assumed to be small.

The element may have variable thickness. The shell thickness and more general properties (e.g., material and number of integration points through the thickness) are specified using section commands (see **SECTYPE**, **SECDATA** and **SECCONTROLS**). Shell section commands allow for both single-layered and composite shell definitions. You may designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. If you use only 1, the integration point is always located midway between the top and the bottom

surfaces. If you use 3 or more points, 2 points are located on the top and the bottom surfaces respectively and the remaining points are distributed evenly between these two points. The default for each layer is 3.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressure may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL208 Geometry"*. Positive pressures act into the element.

Temperatures may be input as element body loads at the corners of the outside faces of the element and the corners of the interfaces between layers (1-1024 maximum). The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and exactly NL+1 (where NL is the number of layers in the shell section) temperatures are input, one temperature is used for the bottom corners of each layer, and the last temperature is for the top corners of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1 and T2; T2 (as input) is used for T3 and T4, etc. For any other input patterns, unspecified temperatures default to TUNIF.

Nodal forces, if any, should be input on a full 360° basis.

KEYOPT(1) is the membrane option. When KEYOPT(1) = 1, the element uses one integration point through-the-thickness and accounts for only membrane stiffness (that is, the bending and transverse shear stiffness are ignored).

KEYOPT(2) controls the torsion capability. When KEYOPT(2) = 1, the element allows constant torsion by allowing a translational degree of freedom UZ in the circumferential direction.

KEYOPT(3) is used to include or suppress internal nodes. When KEYOPT(3) = 2, the element contains an extra internal node and adopts a two-point integration rule. By default, the element uses one-point integration scheme (see *Figure 1, "SHELL208 Geometry"*). Internal nodes are not accessible to users. Therefore, boundary conditions/loading can not be specified on those nodes.

SHELL208 includes the effects of transverse shear deformation. The transverse shear stiffness E11 can be specified using **SECCONTROLS**. For a single-layered shell with isotropic material, default transverse shear stiffness is kGh, in which k = 5/6, G is the shear modulus, and h is the thickness of the shell.

SHELL208 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties.

Set KEYOPT(8) = 2 to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate. Examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

Set KEYOPT(9) = 1 to read initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in *SHELL208 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL208 Input Summary

Nodes
I, J

Degrees of Freedom

UX, UY, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 0
UX, UY -- If KEYOPT(1) = 1 and KEYOPT(2) = 0
UX, UY, UZ, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 1
UX, UY, UZ -- If KEYOPT(1) = 1 and KEYOPT(2) = 1

Real Constants
None

Section Controls
E11, ADMSUA

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (*or* NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ,
DAMP

Surface Loads

Pressures --

face 1 (I-J) (top, in -N direction),
face 2 (I-J) (bottom, in +N direction)

Body Loads

Temperatures --

For KEYOPT(1) = 0:
T1, T2 (corresponding to nodes I and J) at bottom of layer 1, and T3, T4 (corresponding to nodes I and J) between layers 1-2. A similar relationship exists for all layers, ending with temperatures at the top of layer NL. Hence, for one-layer elements, four temperatures are used.
For KEYOPT(1) = 1:
T1, T2 for layer 1; T3, T4 for layer 2; similarly for all layers (2 * NL maximum). Hence, for one-layer elements, two temperatures are used.

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ANEL)
Other material (USER, SDAMP)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

Adaptive descent is *not* supported.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)

Element stiffness:

0 --

Bending and membrane stiffness (default).

1 --

Membrane stiffness only.

KEYOPT(2)

Torsion capability:

0 --

Excluded (default).

1 --

Included.

KEYOPT(3)

Extra internal node option:

0 --

Suppress extra internal node (default).

2 --

Include extra internal node.

KEYOPT(8)

Storage of layer data:

0 --

Store data for BOTTOM of bottom layer and TOP of top layer (default).

1 --

Store data for TOP and BOTTOM for all layers.

2 --

Store data for TOP, BOTTOM, and MID for all layers.

Caution

Volume of data may be excessive.

KEYOPT(9)

User-defined thickness:

0 --
No user subroutine to provide initial thickness (default).

1 --
Read initial thickness data from user subroutine UTHICK

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

SHELL208 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in *Table 1, "SHELL208 Element Output Definitions"*

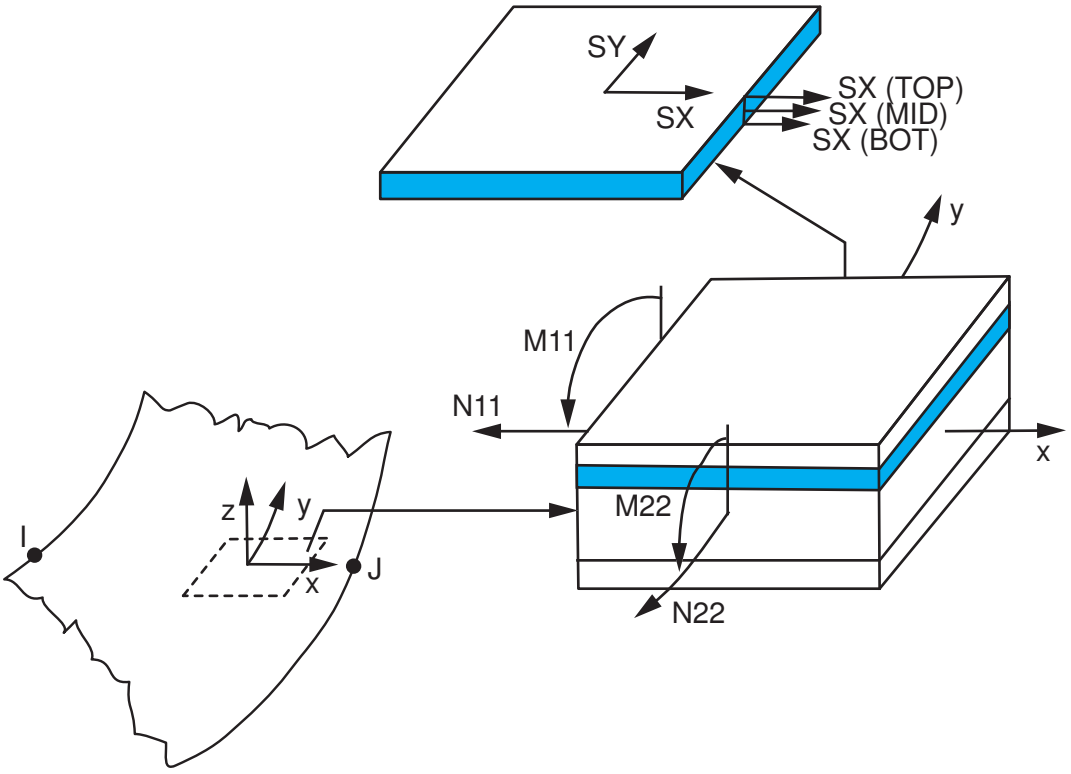
Several items are illustrated in *Figure 2, "SHELL208 Element Stress Output"*.

KEYOPT(8) controls the amount of data output on the result file for processing with the **LAYER** command. Interlaminar shear stress is available at the layer interfaces. Setting KEYOPT(8) = 1 or 2 is necessary for these stresses to be output in POST1. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear force Q13 is available only in resultant form: that is, use SMISC,5. Likewise, the transverse shear strain γ_{13} is constant through the thickness and only available as a SMISC item (SMISC,10).

SHELL208 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; you should use the **OUTRES** command to ensure that the required results are stored in the database.

Figure 2 SHELL208 Element Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL208 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*4*
PRES	Pressures P1 (top) at NODES I, J; P2 (bottom) at NODES I, J	-	Y
TEMP	Temperatures T1, T2 at bottom of layer 1, T3, T4 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL (2*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*3*	*1*
S:INT	Stress intensity	-	*1*
S:EQV	Equivalent stress	-	*1*

Name	Definition	O	R
EPEL:X, Y, Z, XY,YZ,XZ	Elastic strains	*3*	*1*
EPEL:EQV	Equivalent elastic strain	*3*	*1*
EPTH:X, Y, Z, XY,YZ,XZ	Thermal strains	*3*	*1*
EPTH:EQV	Equivalent thermal strain	*3*	*1*
EPPL:X, Y, Z, XY,YZ,XZ	Average plastic strains	*3*	*2*
EPPL:EQV	Equivalent plastic strain	*3*	*2*
EPCR:X, Y, Z, XY ,YZ,XZ	Average creep strains	*3*	*2*
EPCR:EQV	Equivalent creep strain	*3*	*2*
EPTO:X, Y, Z ,XY,YZ,XZ	Total mechanical strains (EPEL+EPPL+EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	*2*
NL:CREQ	Accumulated equivalent creep strain	-	*2*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	*2*
NL:PLWK	Plastic work	-	*2*
NL:HPRES	Hydrostatic pressure	-	*2*
SEND:Elastic, Plastic, Creep	Strain energy densities	-	*2*
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22	Out-of-plane moments (per unit length)	-	Y
Q13	Transverse shear forces (per unit length)	-	Y
E11, E22, E12	Membrane strains	-	Y
K11, K22	Curvatures	-	Y
γ_{13}	Transverse shear strain	-	Y
LOCI:X, Y, Z	Integration point locations	-	*5*
SVAR:1, 2, ... , N	State variables	-	*6*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all five section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the USERMAT subroutine and **TB**,STATE are used.

Table 2, "SHELL208 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL208 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL208 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J
sequence number for data at nodes I, J.

Table 2 SHELL208 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
N11	SMISC	1	-	-
N22	SMISC	2	-	-
N12	SMISC	3	-	-
M11	SMISC	4	-	-
M22	SMISC	5	-	-
Q13	SMISC	6	-	-
ε_{11}	SMISC	7	-	-
ε_{22}	SMISC	8	-	-
ε_{12}	SMISC	9	-	-
k_{11}	SMISC	10	-	-
k_{22}	SMISC	11	-	-
γ_{13}	SMISC	12	-	-
THICK	SMISC	13	-	-
P1	SMISC	-	14	15
P2	SMISC	-	16	17

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 31	8 * (NL - 1) + 32
ILSYZ	SMISC	8 * (i - 1) + 33	8 * (NL - 1) + 34
ILSUM	SMISC	8 * (i - 1) + 35	8 * (NL - 1) + 36
ILANG	SMISC	8 * (i - 1) + 37	8 * (NL - 1) + 38

SHELL208 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane with the Y-axis the axis of symmetry.
- The element must not have a zero length.
- Zero thickness elements or elements tapering to a zero thickness at any corner are not allowed (however, zero thickness layers are allowed).

- In a nonlinear analysis, the solution is terminated if the thickness at any integration point (defined with a nonzero thickness) vanishes (within a small numerical tolerance).
- For nonlinear applications, this element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,ON).
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers may not change between load steps.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- Transverse shear stiffness of the shell section is estimated by a energy equivalence procedure (of the generalized section forces and strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- Calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- A maximum of 250 layers is supported.
- Using KEYOPT(3) = 2 is recommended for most composite analysis (necessary to capture the stress gradients).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.
- SHELL208 with an internal node can not be used in substructures.

SHELL208 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

SHELL209

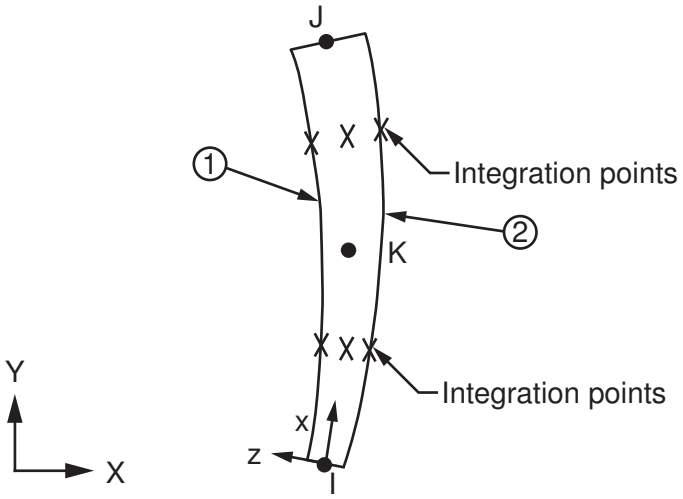
3-Node Finite Strain Axisymmetric Shell

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SHELL209 Element Description

SHELL209 is suitable for analyzing thin to moderately thick axisymmetric shell structures. It is a 3-node element with 3 DOFs at each node: translations in the X, Y directions, and a rotation about the Z-axis. A fourth translational degree of freedom in z direction can be included to model uniform torsion (KEYOPT(2) = 1). When membrane option is used, the rotational degree of freedom is excluded. For higher efficiency, the 2-node element SHELL208 may be more suitable. This element is well suited for linear, large rotation, and/or large strain nonlinear applications. Changes in shell thickness and follower effects of distributed pressures are accounted for in nonlinear analyses, and it can be used for layered applications for modeling laminated composite shells or sandwich construction. See SHELL209 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SHELL209 Geometry



SHELL209 Input Data

Figure 1, "SHELL209 Geometry" shows the geometry, node locations, and element coordinate system for this element. The element is defined by three nodes. For material property labels, the local x-direction corresponds to the meridional direction of the shell element. The local y-direction is the circumferential. The local z-direction corresponds to the through-the-thickness direction. Element formulation is based on logarithmic strain and true stress measures. Element kinematics allows for finite membrane strains (stretching). However, the curvature changes within an increment are assumed to be small.

The element may have variable thickness. The shell thickness and more general properties (e.g., material and number of integration points through the thickness) are specified using section commands (see **SECTYPE**, **SECDATA** and **SECCONTROLS**). Shell section commands allow for both single-layered and composite shell definitions. You may designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. If you use only 1, the integration point is always located midway between the top and the bottom surfaces. If you use 3 or more points, 2 points are located on the top and the bottom surfaces respectively and the remaining points are distributed evenly between these two points. The default for each layer is 3.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressure may be input as surface loads on the element faces as shown by the circled numbers on *Figure 1, "SHELL209 Geometry"*. Positive pressures act into the element.

Temperatures may be input as element body loads at the corners of the outside faces of the element and the corners of the interfaces between layers. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and exactly NL+1 (where NL is the number of layers in the shell section) temperatures are input, one temperature is used for the bottom corners of each layer, and the last temperature is for the top corners of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1, T2, and T3; T2 (as input) is used for T4, T5, and T6, etc. For any other input patterns, unspecified temperatures default to TUNIF.

Nodal forces, if any, should be input on a full 360° basis.

KEYOPT(1) is the membrane option. When KEYOPT(1) = 1, the element uses one integration point through-the-thickness and accounts for only membrane stiffness (that is, the bending and transverse shear stiffness are ignored).

KEYOPT(2) controls the torsion capability. When KEYOPT(2) = 1, the element allows constant torsion by allowing a translational degree of freedom UZ in the circumferential direction.

SHELL209 includes the effects of transverse shear deformation. The transverse shear stiffness E11 can be specified with **SECCONTROLS**. For a single-layered shell with isotropic material, default transverse shear stiffness is kGh, in which k = 5/6, G is the shear modulus, and h is the thickness of the shell.

SHELL209 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties.

Set KEYOPT(8) = 2 to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate. Examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

Set KEYOPT(9) = 1 to read initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

SHELL209 Input Summary gives a summary of the element input. A general description of element input is given in *Section 2.1: Element Input*

SHELL209 Input Summary

Nodes

I, J, K

Degrees of Freedom

UX, UY, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 0
UX, UY -- If KEYOPT(1) = 1 and KEYOPT(2) = 0

UX, UY, UZ, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 1
UX, UY, UZ -- If KEYOPT(1) = 1 and KEYOPT(2) = 1

Real Constants

None

Section Controls

E11, ADMSUA

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (*or* NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ,
GXZ, DAMP

Surface Loads

Pressures --

face 1 (I-J-K) (top, in -N direction),
face 2 (I-J-K) (bottom, in +N direction)

Body Loads

Temperatures --

For KEYOPT(1) = 0:
T1, T2 T3 (corresponding to nodes I and J) at bottom of layer 1, and T4, T5, T6 (corresponding to nodes I and J) between layers 1-2. A similar relationship exists for all layers, ending with temperatures at the top of layer NL. Hence, for one-layer elements, four temperatures are used.
For KEYOPT(1) = 1:
T1, T2 for layer 1; T3, T4, T5, T6 for layer 2; similarly for all layers (2 * NL maximum). Hence, for one-layer elements, three temperatures are used.

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ANEL)
Other material (USER, SDAMP)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details of the material models.

Note

Adaptive descent is *not* supported.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default).

1 --
Membrane stiffness only.

KEYOPT(2)
Torsion capability:

0 --
Excluded (default).

1 --
Included.

KEYOPT(8)
Storage of layer data:

0 --
Store data for BOTTOM of bottom layer and TOP of top layer (default).

1 --
Store data for TOP and BOTTOM for all layers.

2 --
Store data for TOP, BOTTOM, and MID for all layers.

Caution

Volume of data may be excessive.

KEYOPT(9)
User-defined thickness:

0 --
No user subroutine to provide initial thickness (default).

1 --
Read initial thickness data from user subroutine UTHICK

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines

SHELL209 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in *Table 1, "SHELL209 Element Output Definitions"*

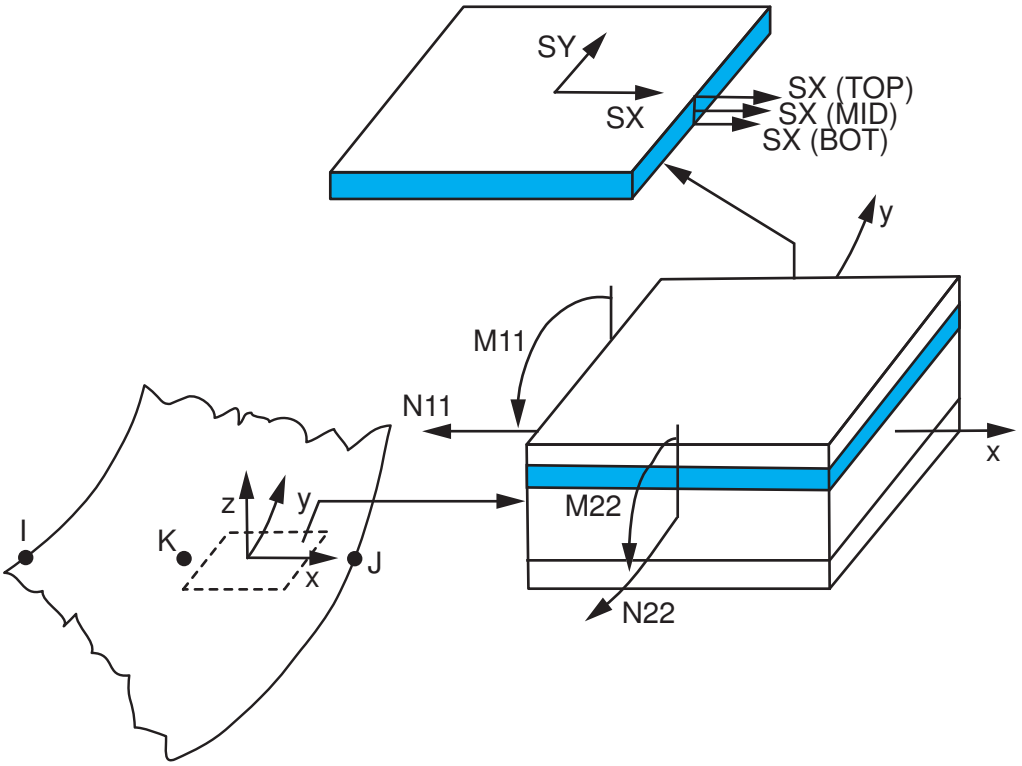
Several items are illustrated in *Figure 2, "SHELL209 Element Stress Output"*.

KEYOPT(8) controls the amount of data output on the result file for processing with the **LAYER** command. Interlaminar shear stress is available at the layer interfaces. Setting KEYOPT(8) = 1 or 2 is necessary for these stresses to be output in POST1. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear force Q13 are available only in resultant form: that is, use SMISC,5. Likewise, the transverse shear strain.

γ_{13} is constant through the thickness and only available as a SMISC item (SMISC,10). SHELL209 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, we suggest using **OUTRES** command to ensure that the required results are stored in the database.

Figure 2 SHELL209 Element Stress Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SHELL209 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	*4*
PRES	Pressures P1 (top) at NODES I, J; P2 (bottom) at NODES I, J	-	Y
TEMP	Temperatures T1, T2 at bottom of layer 1, T3, T4 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL (2*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*3*	*1*
S:INT	Stress intensity	-	*1*
S:EQV	Equivalent stress	-	*1*

Name	Definition	O	R
EPEL:X, Y, Z, XY,YZ,XZ	Elastic strains	*3*	*1*
EPEL:EQV	Equivalent elastic strain	*3*	*1*
EPTH:X, Y, Z, XY,YZ,XZ	Thermal strains	*3*	*1*
EPTH:EQV	Equivalent thermal strain	*3*	*1*
EPPL:X, Y, Z, XY,YZ,XZ	Average plastic strains	*3*	*2*
EPPL:EQV	Equivalent plastic strain	*3*	*2*
EPCR:X, Y, Z, XY,YZ,XZ	Average creep strains	*3*	*2*
EPCR:EQV	Equivalent creep strain	*3*	*2*
EPTO:X, Y, Z ,XY,YZ,XZ	Total mechanical strains (EPEL+EPPL+EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	*2*
NL:CREQ	Accumulated equivalent creep strain	-	*2*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	*2*
NL:PLWK	Plastic work	-	*2*
NL:HPRES	Hydrostatic pressure	-	*2*
SEND:Elastic, Plastic, Creep	Strain energy densities	-	*2*
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22	Out-of-plane moments (per unit length)	-	Y
Q13	Transverse shear forces (per unit length)	-	Y
E11, E22, E12	Membrane strains	-	Y
K11, K22	Curvatures	-	Y
γ_{13}	Transverse shear strain	-	Y
LOCI:X, Y, Z	Integration point locations	-	*5*
SVAR:1, 2, ... , N	State variables	-	*6*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all five section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the USERMAT subroutine and **TB**,STATE are used.

Table 2, "SHELL209 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 2, "SHELL209 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "SHELL208 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J, K
sequence number for data at nodes I, J, K.

Table 2 SHELL209 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
N11	SMISC	1	-	-	-
N22	SMISC	2	-	-	-
N12	SMISC	3	-	-	-
M11	SMISC	4	-	-	-
M22	SMISC	5	-	-	-
Q13	SMISC	6	-	-	-
ε_{11}	SMISC	7	-	-	-
ε_{22}	SMISC	8	-	-	-
ε_{12}	SMISC	9	-	-	-
k_{11}	SMISC	10	-	-	-
k_{22}	SMISC	11	-	-	-
γ_{13}	SMISC	12	-	-	-
THICK	SMISC	13	-	-	-
P1	SMISC	-	14	15	16
P2	SMISC	-	17	18	19

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 31	8 * (NL - 1) + 32
ILSYZ	SMISC	8 * (i - 1) + 33	8 * (NL - 1) + 34
ILSUM	SMISC	8 * (i - 1) + 35	8 * (NL - 1) + 36
ILANG	SMISC	8 * (i - 1) + 37	8 * (NL - 1) + 38

SHELL209 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane with the Y-axis the axis of symmetry.
- The element must not have a zero length.
- Zero thickness elements or elements tapering to a zero thickness at any corner are not allowed (however, zero thickness layers are allowed).

- In a nonlinear analysis, the solution is terminated if the thickness at any integration point (defined with a nonzero thickness) vanishes (within a small numerical tolerance).
- For nonlinear applications, this element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,ON).
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers may not change between load steps.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- Transverse shear stiffness of the shell section is estimated by a energy equivalence procedure (of the generalized section forces and strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- Calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- A maximum of 250 layers is supported.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified by **SSTIF**,ON. Prestress effects can be activated by the **PSTRES** command.

SHELL209 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

COMBI214

2-D Spring-Damper Bearing

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

COMBI214 Element Description

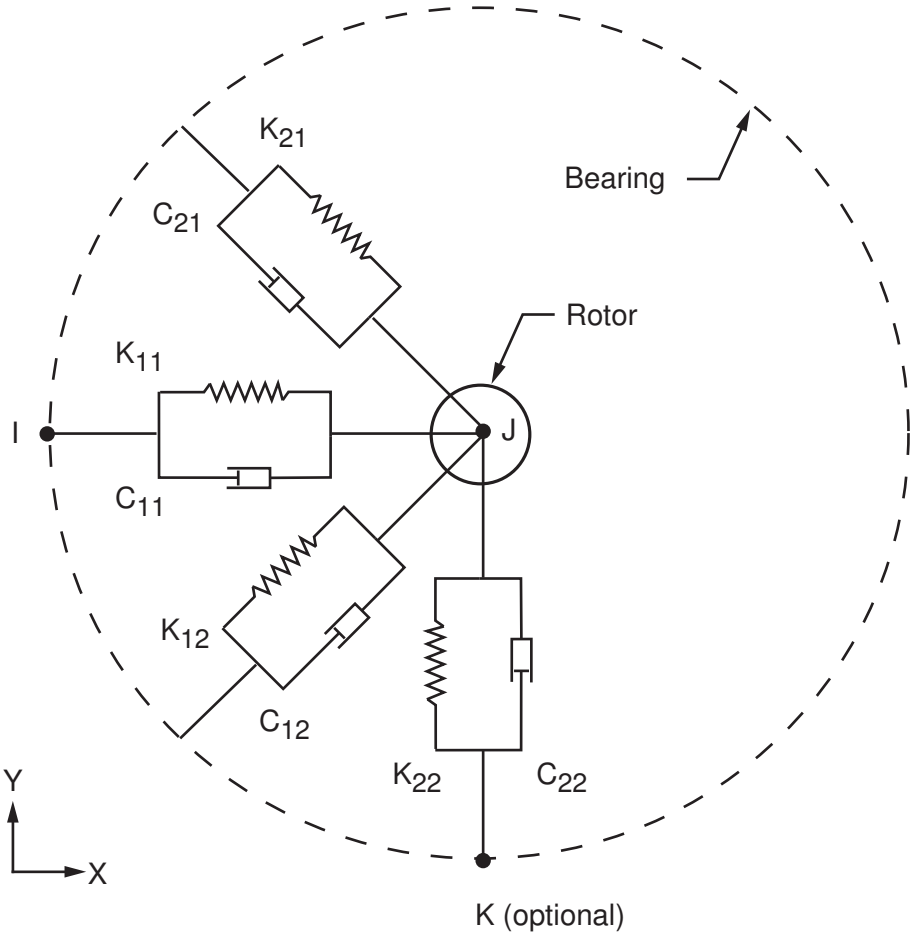
COMBI214 has longitudinal as well as cross-coupling capability in 2-D applications. It is a tension-compression element with up to two degrees of freedom at each node: translations in any *two* nodal directions (x, y, or z). COMBI214 has two nodes plus one optional orientation node. No bending or torsion is considered.

The spring-damper element has no mass. Masses can be added by using the appropriate mass element (MASS21). The spring or the damping capability may be removed from the element.

A longitudinal spring/damper with torsion capabilities is available via the COMBIN14 element. A general spring or damper is also available in the stiffness, damping or mass matrix element MATRIX27. Another spring-damper element having its direction of action determined by the nodal coordinate directions is COMBIN40.

For more information about this element, see *Section 14.214: COMBI214 - 2-D Spring-Damper Bearing* in the *Theory Reference for ANSYS and ANSYS Workbench*.

Figure 1 COMBI214 Geometry



For linear analyses, I and J can be coincident.

COMBI214 Input Data

The geometry, node locations, and coordinate system for this element are shown in *Figure 1, "COMBI214 Geometry"*. The element is defined by two nodes. It has stiffness characteristics K_{11}, K_{22}, K_{12} and K_{21} and damping characteristics C_{11}, C_{22}, C_{12} and C_{21}. The stiffness coefficients should have units of Force/Length, and the damping coefficient units are Force*Time/Length. (The damping capability is not used for static or undamped modal analyses.)

The third node is for orientation and applies to nonlinear analyses only.

For stiffness and damping real constants, either numerical values or tabular array inputs can be specified. If specifying tabular inputs, enclose the table name within "%" characters (%`tabname`%). These real constants can vary with the amplitude of the rotational velocity vector (defined via the **OMEGA** or **CMOMEGA** command). Use the ***DIM** command and the primary variable OMEGS to dimension the table and identify the variable. Because the amplitude of the rotational velocity vector is an absolute value, only positive values of OMEGS in the table parameter are valid. For more information about using tabular inputs, see *Section 3.10: Array Parameters* in the *ANSYS Parametric Design Language Guide*, *Section 2.5.14: Applying Loads Using TABLE Type Array Parameters* in the *Basic Analysis Guide*, and *Section 2.9: Performing a Thermal Analysis Using Tabular Boundary Conditions* in the *Thermal Analysis Guide*.

KEYOPT(2) = 0 through 2 options define the element plane. The element operates in the nodal coordinate system.

The KEYOPT(3) = 0 and 1 options specify whether or not the element is symmetric. When symmetric, cross-coupling terms in stiffness and damping coefficients are equal (that is, $K_{12} = K_{21}$ and $C_{12} = C_{21}$).

A summary of the element input is given in *COMBI214 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

COMBI214 Input Summary

Nodes

I, J, K (The K orientation node is optional and for nonlinear analyses only.)

Degrees of Freedom

UX, UY (KEYOPT (2) = 0)
UY, UZ (KEYOPT (2) = 1)
UX, UZ (KEYOPT (2) = 2)

Real Constants

K_{11}, K_{22}, K_{12}, K_{21} , C_{11}, C_{22}, C_{12}, C_{21}
K_{ij} - (i=1,2 j=1,2) Stiffness coefficients
C_{ij} - (i=1,2 j=1,2) Damping coefficients

Note

Real constants may be defined as table parameters as a function of omega (using primary variable OMEGS).

Material Properties

None

Surface Loads

None

Body Loads

None

Special Features

Stress stiffening

Large deflections

Birth and death

KEYOPT(2)

Degrees of freedom selection:

0 --

Element lies in a plane parallel to the XY plane. The degrees of freedom are UX and UY. This value option is the default.

1 --

Element lies in a plane parallel to the YZ plane. The degrees of freedom are UY and UZ.

2 --

Element lies in a plane parallel to the XZ plane. The degrees of freedom are UX and UZ.

KEYOPT(3)

Symmetry:

0 --

Element is symmetric: $K_{12} = K_{21}$ and $C_{12} = C_{21}$. This option is the default.

1 --

Element is not symmetric.

COMBI214 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "COMBI214 Element Output Definitions"*.

The Element Output Definitions table uses the following notation:

(1) and (2) indicate the first and second axis of the element plane defined by KEYOPT(2). For example, if KEYOPT(2) equals 0, then (1) is the X axis and (2) is the Y axis.

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 COMBI214 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
FORC1	Spring force along (1)	Y	Y
FORC2	Spring force along (2)	Y	Y

Name	Definition	O	R
STRETCH1	Stretch of spring along (1)	Y	Y
STRETCH2	Stretch of spring along (2)	Y	Y
VELOCITY1	Velocity along (1)	-	Y
VELOCITY2	Velocity along (2)	-	Y
DAMPING FORCE1	Damping force along (1) -- Zero unless this is a transient analysis (**ANTYPE**,TRANS) *and* damping is present	Y	Y
DAMPING FORCE2	Damping force along (2) -- Zero unless this is a transient analysis (**ANTYPE**,TRANS) *and* damping is present	Y	Y

1. Available only at centroid as a ***GET** item.

Table 2,"COMBI214 Item and Sequence Numbers" lists output available via the **ETABLE** command using the Sequence Number method. See *Chapter 5: The General Postprocessor (POST1)* in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information. The following notation is used in *Table 2,"COMBI214 Item and Sequence Numbers"*:

Name
Output quantity as defined in *Table 1,"COMBI214 Element Output Definitions"*

Item
Predetermined Item label for the **ETABLE** command

E
Sequence number for single-valued or constant element data

Output data for COMBI214 consists of the following:

Table 2 COMBI214 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORC1	SMISC	1
FORC2	SMISC	2
STRETCH1	NMISC	1
STRETCH2	NMISC	2
VELOCITY1	NMISC	3
VELOCITY2	NMISC	4
DAMPING FORCE1	NMISC	5
DAMPING FORCE2	NMISC	6

COMBI214 Assumptions and Restrictions

- Nodes must lie in the plane defined by KEYOPT(2).
- The following applies to a nonlinear analysis:

  -- The orientation node K is required.

  -- The length of the spring-damper element must not be zero (that is, nodes I, J and K should not be coincident because the node locations determine the spring orientation).
- Line (IJ) must be parallel to (1). Line (JK) must be parallel to (2).

- The element allows only a uniform stress in the springs.
- The following applies when KEYOPT(3) = 0 (symmetric):

  -- If K_{12} is non-zero and K_{21} is zero, then K_{21} is set to K_{12}.

  -- If C_{12} is non-zero and C_{21} is zero, then C_{21} is set to C_{12}.
- The spring or the damping capability may be deleted from the element by setting all K_{ij} (i=1,2 j=1,2) or all C_{ij} (i=1,2 j=1,2) equal to zero, respectively.
- The degrees of freedom are specified in the nodal coordinate system and are the same for both nodes. (For more information, see *Section 2.3.2: Elements that Operate in the Nodal Coordinate System*.) If the nodal coordinate systems are rotated relative to each other, the same degree of freedom may be in different directions (thereby giving possibly unexpected results).
- No moment effects are included; that is, if the nodes are offset from the lines of action, moment equilibrium may not be satisfied.
- The element is defined such that a positive displacement of node J relative to node I tends to stretch the spring. If, for a given set of conditions, nodes I and J are interchanged, a positive displacement of node J relative to node I tends to compress the spring.

COMBI214 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; C_{ij} (i=1,2 j=1,2) are not allowed.
- Only stress stiffening and large deflections are allowed.

PLANE223

2-D 8-Node Coupled-Field Solid

MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

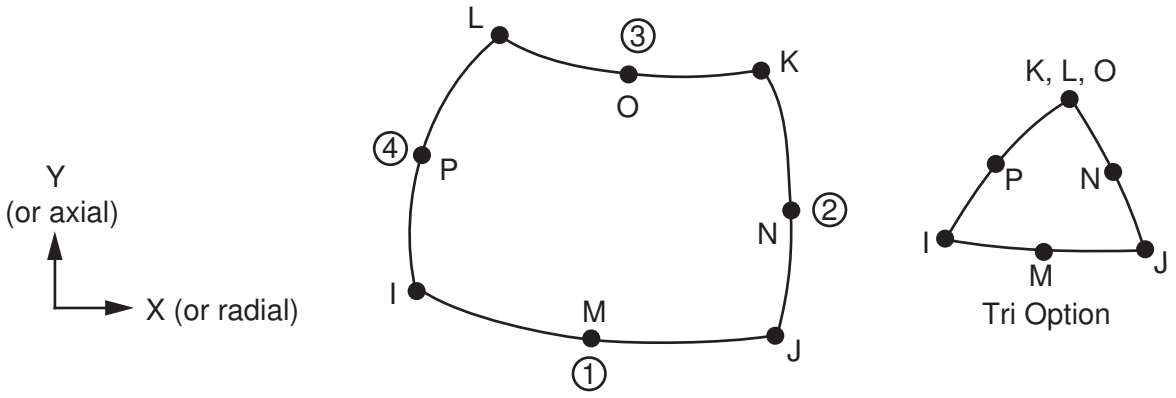
PLANE223 Element Description

PLANE223 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric

The element has eight nodes with up to four degrees of freedom per node. Structural capabilities are elastic only and include large deflection and stress stiffening. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. See PLANE223 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE223 Geometry



PLANE223 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE223 Geometry"*. The element input data includes eight nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in *Table 1, "PLANE223 Field Keys"*. For example, KEYOPT(1) is set to

11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 1 PLANE223 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY	FX, FY	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 2 PLANE223 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Structural-Thermal [*1*], [*2*]	11	UX, UY, TEMP	FX, FY, HEAT	Force, Heat Flow	Static Full Harmonic Full Transient
Piezoresistive	101	UX, UY, VOLT	FX, FY, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [*3*]	UX, UY, VOLT	FX, FY, CHRG	Force, Electric Charge (positive)	Static Full Transient
Piezoelectric	1001 [*3*]	UX, UY, VOLT	FX, FY, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [*1*]	111	UX, UY, TEMP, VOLT	FX, FY, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [*1*], [*2*]	1011	UX, UY, TEMP, VOLT	FX, FY, HEAT, CHRG	Force, Heat Flow, Electric Charge (negative)	Static Full Harmonic Full Transient

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling.
2. For full harmonic analyses, strong structural-thermal coupling only applies.

3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, or electrostatic) and those required for field coupling. Material properties are defined with the **MP**, **MPDATA** and **TB** commands.

Table 3 PLANE223 Material Properties

Coupled-Field Analysis	KEYOPT(1)	Material Properties
Structural-Thermal	11	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, DENS, C, ENTH **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ)
Piezoresistive	101	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** RSVX, RSVY, PERX, PERY **Coupling** PZRS
Electroelastic	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, DPER
Piezoelectric	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, LSST, DPER **Coupling** PIEZ
Thermal-Electric	110	**Thermal** KXX, KYY, DENS, C, ENTH **Electric** RSVX, RSVY, PERX, PERY

Coupled-Field Analysis	KEYOPT(1)	Material Properties
		Coupling SBKX, SBKY
Structural-Thermoelectric	111	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, DENS, C, ENTH **Electric** RSVX, RSVY, PERX, PERY **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, PZRS
Thermal-Piezoelectric	1011	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, DENS, C, ENTH **Electric** PERX, PERY, LSST, DPER **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. Nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in *Section 2.8: Node and Element Loads*. Surface loads may be input on the element faces indicated by the circled numbers in *Figure 1, "PLANE223 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

PLANE223 surface and body loads are given in the following table. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Table 4 PLANE223 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through P	HGEN

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Piezoresistive	101	Surface	Pressure	PRES
		Body	Temperature -- Nodes I through P	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS
		Body	Temperature -- Nodes I through P	TEMP
			Volume Charge Density -- Nodes I through P	CHRGD
Thermal-Electric	110	Surface	Pressure Surface Charge Density	PRES CHRGS
		Body	Heat Generation -- Nodes I through P	HGEN
Structural-Thermoelectric	111	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through P	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Density	PRES CHRGS
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through P	HGEN
			Volume Charge Density -- Nodes I through P	CHRGD

A summary of the element input is given in *PLANE223 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE223 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
Set by KEYOPT(1). See *Table 2, "PLANE223 Coupled-Field Analyses"*.

Real Constants
None

Material Properties
See *Table 3, "PLANE223 Material Properties"*.

Surface Loads
See *Table 4, "PLANE223 Surface and Body Loads"*.

Body Loads
See *Table 4, "PLANE223 Surface and Body Loads"*.

Special Features

Large deflection
Stress stiffening

KEYOPT(1)
Element degrees of freedom. See *Table 2, "PLANE223 Coupled-Field Analyses"*.

KEYOPT(2)
Structural-thermal coupling method (KEYOPT(1) = 11, 111, or 1011):

0 --
Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a strong coupled response is achieved after one iteration.

1 --
Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain

KEYOPT(4)
Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --
Applied to every element node.

1 --
Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --
Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

PLANE223 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 5, "PLANE223 Element Output Definitions"*.

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 5 PLANE223 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	-	*2*
STRUCTURAL-THERMAL (KEYOPT(1) = 11)			
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	*1*
UT	Total strain energy [*7*]	-	*1*
PIEZORESISTIVE (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	*1*
JC:X, Y, SUM	Conduction current density components (X, Y) and vector magnitude	-	*1*
JS:X, Y, SUM	Current density components (X, Y) and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*]	-	*1*
ELECTROELASTIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	*1*
D:X, Y, SUM	Electric flux density components (X, Y) and vector magnitude	-	*1*

Name	Definition	O	R
FMAG:X, Y, SUM	Electrostatic force components (X, Y) and vector magnitude	-	*1*
PIEZOELECTRIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	*1*
D:X, Y, SUM	Electric flux density components (X, Y) and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UE, UD, UM	Stored elastic, dielectric, and mutual energies	-	*1*
THERMAL-ELECTRIC (KEYOPT(1) = 110)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
STRUCTURAL-THERMOELECTRIC (KEYOPT(1) = 111)			
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UT	Total strain energy [*7*]	-	*1*
THERMAL-PIEZOELECTRIC (KEYOPT(1) = 1011)			
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*

Name	Definition	O	R
EPTH:X, Y, Z, XY	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, SUM	Electric field components and vector magnitude	-	*1*
D:X, Y, SUM	Electric flux density components and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UE, UD, UM	Stored elastic, dielectric, and mutual energies	-	*1*
UT	Total strain energy [*7*]	-	*1*

1. Solution values are output only if calculated (based on input values).
2. Available only at centroid as a ***GET** item.
3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).
4. JS represents the sum of element conduction and displacement current densities.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.
6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Theory Reference for ANSYS and ANSYS Workbench*.
7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Theory Reference for ANSYS and ANSYS Workbench*.

Table 6, "PLANE223 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 6, "PLANE223 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 5, "PLANE223 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 6 PLANE223 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
UE	NMISC	1
UD	NMISC	2
UM	NMISC	3
UT	NMISC	4

PLANE223 Assumptions and Restrictions

- PLANE 223 assumes a unit thickness.
- When NLGEOM is ON, SSTIF defaults to OFF.
- PLANE 223 uses 2 x 2 and 3 point integration rules to calculate the element matrices and load vectors for the quad and triangle geometries, respectively.
- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.
- The element must lie in a global X-Y plane as shown in *Figure 1, "PLANE223 Geometry"* and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

PLANE223 Product Restrictions

There are no product-specific restrictions for this element.

SOLID226

3-D 20-Node Coupled-Field Solid

MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

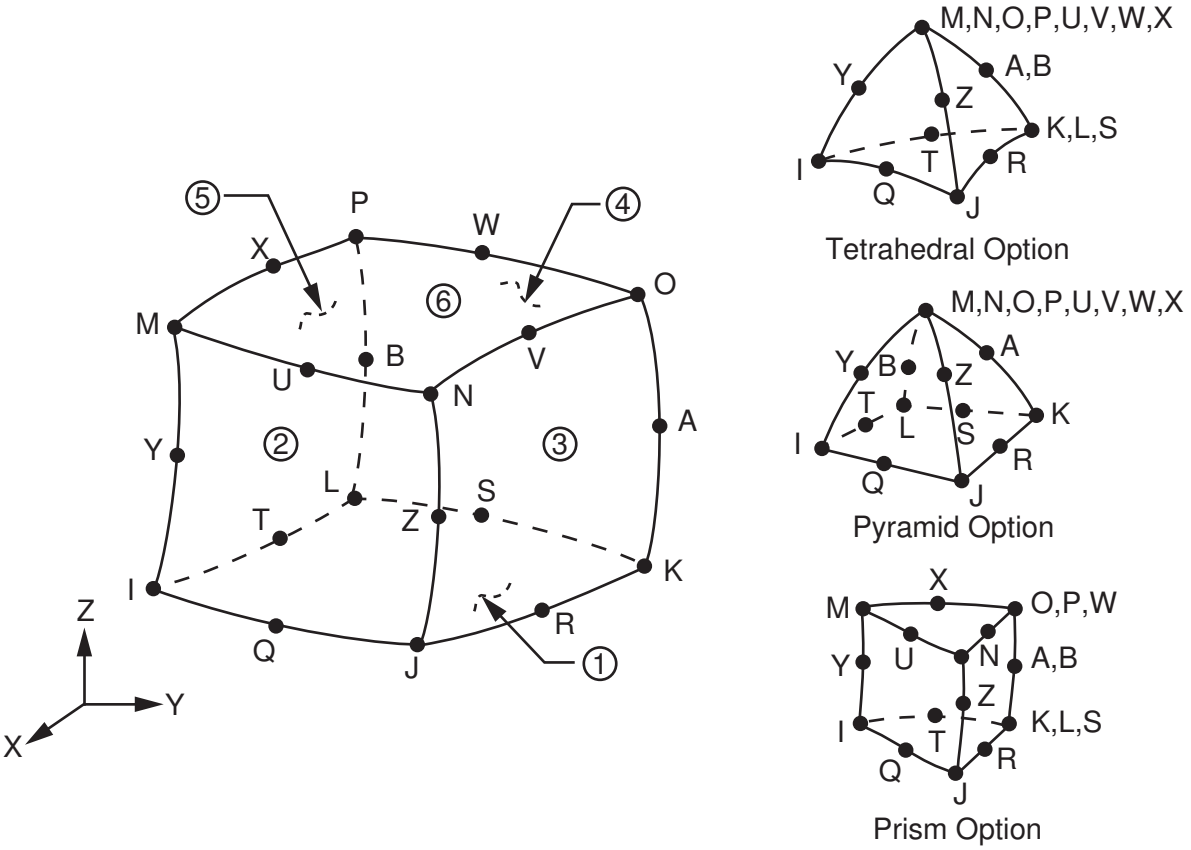
SOLID226 Element Description

SOLID226 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric

The element has twenty nodes with up to five degrees of freedom per node. Structural capabilities are elastic only and include large deflection and stress stiffening. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. See SOLID226 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID226 Geometry



SOLID226 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID226 Geometry"*. The element input data includes twenty nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in *Table 1, "SOLID226 Field Keys"*. For example, KEYOPT(1) is set to 11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 1 SOLID226 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY, UZ	FX, FY, FZ	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 2 SOLID226 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Structural-Thermal [*1*], [*2*]	11	UX, UY, UZ, TEMP	FX, FY, FZ, HEAT	Force, Heat Flow	Static Full Harmonic Full Transient
Piezoresistive	101	UX, UY, UZ, VOLT	FX, FY, FZ, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [*3*]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (positive)	Static Full Transient
Piezoelectric	1001 [*3*]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [*1*]	111	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [*1*], [*2*]	1011	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, CHRG	Force, Heat Flow, Electric Charge (negative)	Static Full Harmonic Full Transient

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling.
2. For full harmonic analyses, strong structural-thermal coupling only applies.
3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, or electrostatic) and those required for field coupling. Material properties are defined with the **MP**, **MPDATA** and **TB** commands.

Table 3 SOLID226 Material Properties

Coupled-Field Analysis	KEYOPT(1)	Material Properties
Structural-Thermal	11	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL

Coupled-Field Analysis	KEYOPT(1)	Material Properties
		Thermal KXX, KYY, KZZ, DENS, C, ENTH **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ)
Piezoresistive	101	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSZ, THSY, THSZ), DENS, DAMP, ANEL **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ **Coupling** PZRS
Electroelastic	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSZ, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, PERZ, DPER
Piezoelectric	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSZ, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, PERZ, DPER, LSST **Coupling** PIEZ
Thermal-Electric	110	**Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ **Coupling** SBKX, SBKY, SBKZ
Structural-Thermoelectric	111	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ **Coupling**

Coupled-Field Analysis	KEYOPT(1)	Material Properties
		ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, SBKZ, PZRS
Thermal-Piezoelectric	1011	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** PERX, PERY, PERZ, LSST, DPER **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Loads may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID226 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

SOLID226 surface and body loads are given in the following table. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Table 4 SOLID226 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I, J, ..., A, B	HGEN
Piezoresistive	101	Surface	Pressure	PRES
		Body	Temperature -- Nodes I, J, ..., A, B	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS
		Body	Temperature -- Nodes I, J, ..., A, B	TEMP
			Volume Charge Density -- Nodes I, J, ..., A, B	CHRGD
Thermal-Electric	110	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I, J, ..., A, B	HGEN
Structural-Thermoelectric	111	Surface	Pressure	PRES
			Convection Heat Flux	CONV HFLUX

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
			Radiation	RDSF
		Body	Heat Generation -- Nodes I, J, ..., A, B	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Density	PRES CHRGS
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I, J, ..., A, B	HGEN
			Volume Charge Density -- Nodes I, J, ..., A, B	CHRGD

A summary of the element input is given in *SOLID226 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID226 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
Set by KEYOPT(1). See *Table 2, "SOLID226 Coupled-Field Analyses"*.

Real Constants
None

Material Properties
See *Table 3, "SOLID226 Material Properties"*.

Surface Loads
See *Table 4, "SOLID226 Surface and Body Loads"*.

Body Loads
See *Table 4, "SOLID226 Surface and Body Loads"*.

Special Features

Large deflection
Stress stiffening

KEYOPT(1)
Element degrees of freedom. See *Table 2, "SOLID226 Coupled-Field Analyses"*.

KEYOPT(2)
Structural-thermal coupling method (KEYOPT(1) = 11, 111, or 1011):

0 --
Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a strong coupled response is achieved after one iteration.

1 --
Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(4)
Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --
Applied to every element node.

1 --
Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --
Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

SOLID226 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 5, "SOLID226 Element Output Definitions"*.

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 5 SOLID226 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	*2*
STRUCTURAL-THERMAL (KEYOPT(1) = 11)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
UT	Total strain energy [*7*]	-	*1*
PIEZORESISTIVE (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y

Name	Definition	O	R
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPEL:EQV	Equivalent elastic strains [*3*]	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components (X, Y, Z) and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*]	-	*1*
ELECTROELASTIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	*1*
FMAG:X, Y, Z, SUM	Electrostatic force components (X, Y, Z) and vector magnitude	-	*1*
PIEZOELECTRIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPEL:EQV	Equivalent elastic strains [*3*]	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*

Name	Definition	O	R
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UE, UD, UM	Stored elastic, dielectric, and mutual energies	-	*1*
THERMAL-ELECTRIC (KEYOPT(1) = 110)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
STRUCTURAL-THERMOELECTRIC (KEYOPT(1) = 111)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UT	Total strain energy [*7*]	-	*1*
THERMAL-PIEZOELECTRIC (KEYOPT(1) = 1011)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*

Name	Definition	O	R
UE, UD, UM	Stored elastic, dielectric, and mutual energies	-	*1*
UT	Total strain energy [*7*]	-	*1*

1. Solution values are output only if calculated (based on input values).
2. Available only at centroid as a ***GET** item.
3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).
4. JS represents the sum of element conduction and displacement current densities.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.
6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Theory Reference for ANSYS and ANSYS Workbench*.
7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Theory Reference for ANSYS and ANSYS Workbench*.

Table 5, "SOLID226 Element Output Definitions" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 6, "SOLID226 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 5, "SOLID226 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 6 SOLID226 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
UE	NMISC	1
UD	NMISC	2
UM	NMISC	3
UT	NMISC	4

SOLID226 Assumptions and Restrictions

- When NLGEOM is ON, SSTIF defaults to OFF.
- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.

- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

SOLID226 Product Restrictions

There are no product-specific restrictions for this element.

SOLID227

3-D 10-Node Coupled-Field Solid

MP <> <> <> <> <> <> <> <> <> <> PP <>
Product Restrictions

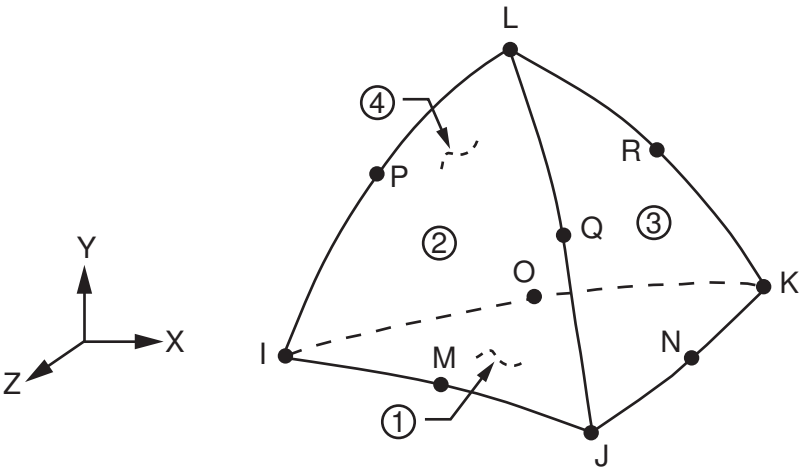
SOLID227 Element Description

SOLID227 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric

The element has ten nodes with up to five degrees of freedom per node. Structural capabilities are elastic only and include large deflection and stress stiffening. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. See SOLID227 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID227 Geometry



SOLID227 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID227 Geometry"*. The element input data includes ten nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in *Table 1, "SOLID227 Field Keys"*. For example, KEYOPT(1) is set to 11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 1 SOLID227 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY, UZ	FX, FY, FZ	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 2 SOLID227 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analyses Type
Structural-Thermal [*1*], [*2*]	11	UX, UY, UZ, TEMP	FX, FY, FZ, HEAT	Force, Heat Flow	Static Full Harmonic Full Transient
Piezoresistive	101	UX, UY, UZ, VOLT	FX, FY, FZ, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [*3*]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (positive)	Static Full Transient
Piezoelectric	1001 [*3*]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [*1*]	111	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [*1*], [*2*]	1011	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, CHRG	Force, Heat Flow, Electric Charge (negative)	Static Full Harmonic Full Transient

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling.

2. For full harmonic analyses, strong structural-thermal coupling only applies.
3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, or electrostatic) and those required for field coupling. Material properties are defined with the **MP**, **MPDATA** and **TB** commands.

Table 3 SOLID227 Material Properties

Coupled-Field Analysis	KEYOPT(1)	Material Properties
Structural-Thermal	11	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ)
Piezoresistive	101	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ **Coupling** PZRS
Electroelastic	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, PERZ, DPER
Piezoelectric	1001	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), DENS, DAMP, ANEL **Electric** PERX, PERY, PERZ, DPER, LSST **Coupling** PIEZ
Thermal-Electric	110	**Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ

Coupled-Field Analysis	KEYOPT(1)	Material Properties
		Coupling SBKX, SBKY, SBKZ
Structural-Thermoelectric	111	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** RSVX, RSVY, RSVZ, PERX, PERY, PERZ **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, SBKZ, PZRS
Thermal-Piezoelectric	1011	**Structural** EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, DAMP, ANEL **Thermal** KXX, KYY, KZZ, DENS, C, ENTH **Electric** PERX, PERY, PERZ, LSST, DPER **Coupling** ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands.

Element loads are described in *Section 2.8: Node and Element Loads*. Loads may be input on the element faces indicated by the circled numbers in *Figure 1, "SOLID227 Geometry"* using the **SF** and **SFE** commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

SOLID227 surface and body loads are given in the following table. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Table 4 SOLID227 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through R	HGEN
Piezoresistive	101	Surface	Pressure	PRES

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
		Body	Temperatures -- Nodes I through R	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS
		Body	Temperatures -- Nodes I through R	TEMP
			Volume Charge Density -- Nodes I through R	CHRGD
Thermal-Electric	110	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through R	HGEN
Structural-Thermoelectric	111	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through R	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Densitiy	PRES CHRGS
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through R	HGEN
			Volume Charge Density -- Nodes I through R	CHRGD

A summary of the element input is given in *SOLID227 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID227 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
Set by KEYOPT(1). See *Table 2, "SOLID227 Coupled-Field Analyses"*.

Real Constants
None

Material Properties
See *Table 3, "SOLID227 Material Properties"*.

Surface Loads
See *Table 4, "SOLID227 Surface and Body Loads"*.

Body Loads
See *Table 4, "SOLID227 Surface and Body Loads"*.

Special Features

Large deflection
Stress stiffening

KEYOPT(1)
Element degrees of freedom. See *Table 2, "SOLID227 Coupled-Field Analyses"*.

KEYOPT(2)
Structural-thermal coupling method (KEYOPT(1) = 11, 111, or 1011):

0 --
Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a strong coupled response is achieved after one iteration.

1 --
Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(4)
Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --
Applied to every element node.

1 --
Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --
Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

SOLID227 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 5, "SOLID227 Element Output Definitions"*.

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 5 SOLID227 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	*2*

Name	Definition	O	R
STRUCTURAL-THERMAL (KEYOPT(1) = 11)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
UT	Total strain energy [*7*]	-	*1*
PIEZORESISTIVE (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPEL:EQV	Equivalent elastic strains [*3*]	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components (X, Y, Z) and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*]	-	*1*
ELECTROELASTIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	*1*

Name	Definition	O	R
FMAG:X, Y, Z, SUM	Electrostatic force components (X, Y, Z) and vector magnitude	-	*1*
PIEZOELECTRIC (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPEL:EQV	Equivalent elastic strains [*3*]	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	*1*
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UE	Stored elastic energy	-	*1*
UD	Stored dielectric energy	-	*1*
UM	Stored mutual energy	-	*1*
THERMAL-ELECTRIC (KEYOPT(1) = 110)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
STRUCTURAL-THERMOELECTRIC (KEYOPT(1) = 111)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*4*]	*1*	*1*

Name	Definition	O	R
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UT	Total strain energy [*7*]	-	*1*
THERMAL-PIEZOELECTRIC (KEYOPT(1) = 1011)			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	*1*
S:1, 2, 3	Principal stresses	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	*1*
EPEL:1, 2, 3	Principal elastic strains	-	*1*
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	*1*
EPTH:EQV	Equivalent thermal strain [*3*]	-	*1*
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	*1*
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	*1*
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	*1*
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	*1*
JHEAT	Joule heat generation per unit volume [*5*], [*6*]	-	*1*
UE, UD, UM	Stored elastic, dielectric, and mutual energies	-	*1*
UT	Total strain energy [*7*]	-	*1*

1. Solution values are output only if calculated (based on input values).
2. Available only at centroid as a ***GET** item.
3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).
4. JS represents the sum of element conduction and displacement current densities.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.
6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Theory Reference for ANSYS and ANSYS Workbench*.
7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Theory Reference for ANSYS and ANSYS Workbench*.

Table 5, "SOLID227 Element Output Definitions" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* of this manual for more information. The following notation is used in *Table 6, "SOLID227 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 5, "SOLID227 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 6 SOLID227 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
UE	NMISC	1
UD	NMISC	2
UM	NMISC	3
UT	NMISC	4

SOLID227 Assumptions and Restrictions

- When NLGEOM is ON, SSTIF defaults to OFF.
- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same through reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

SOLID227 Product Restrictions

There are no product-specific restrictions for this element.

PLANE230

2-D 8-Node Electric Solid

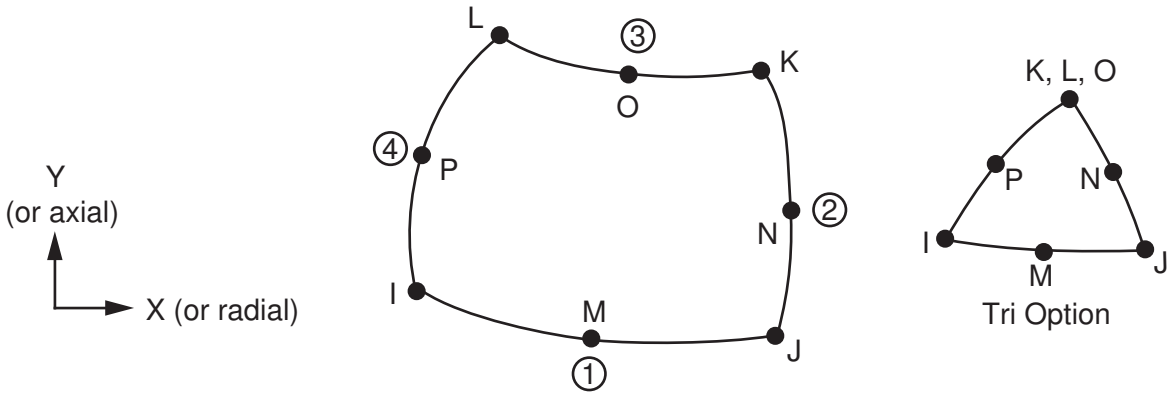
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

PLANE230 Element Description

PLANE230 is a 2-D, 8-node, current-based electric element. The element has one degree of freedom, voltage, at each node. The 8-node elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See *Section 14.230: PLANE230 - 2-D 8-Node Electric Solid* in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 PLANE230 Geometry



PLANE230 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "PLANE230 Geometry"*. The element is defined by eight nodes and orthotropic material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *PLANE230 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*. For axisymmetric applications see *Section 2.12: Axisymmetric Elements*.

PLANE230 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
RSVX, RSVY, PERX, PERY, LSST

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features
Birth and death

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

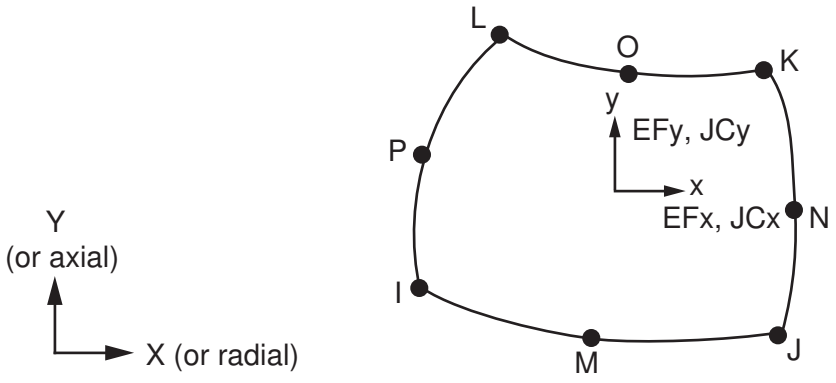
PLANE230 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in *Table 1, "PLANE230 Element Output Definitions"*.

Several items are illustrated in *Figure 2, "PLANE230 Output"*. The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results. The element output directions are parallel to the element coordinate system as shown in *Figure 2, "PLANE230 Output"*.

Figure 2 PLANE230 Output



The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 PLANE230 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	*1*	-
EF:X, Y, SUM	Electric field components and vector magnitude	*1*	*1*
JC:X, Y, SUM	Conduction current density components and vector magnitude	*1*	*1*
JS:X, Y, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*5*] [*6*]	*1*	*1*
SENE:	Stored electric energy [*6*]	*1*	*1*
D:X, Y, SUM	Electric flux density components and vector magnitude	*1*	*1*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT.

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Table 2, "PLANE230 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "PLANE230 Item and Sequence Numbers"*:

Name
output quantity as defined in the *Table 1, "PLANE230 Element Output Definitions"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 2 PLANE230 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DSUM	NMISC	3
JTX	NMISC	4
JTY	NMISC	5
JTSUM	NMISC	6

PLANE230 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in *Figure 2, "PLANE230 Output"*, and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.

PLANE230 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

SOLID231

3-D 20-Node Electric Solid

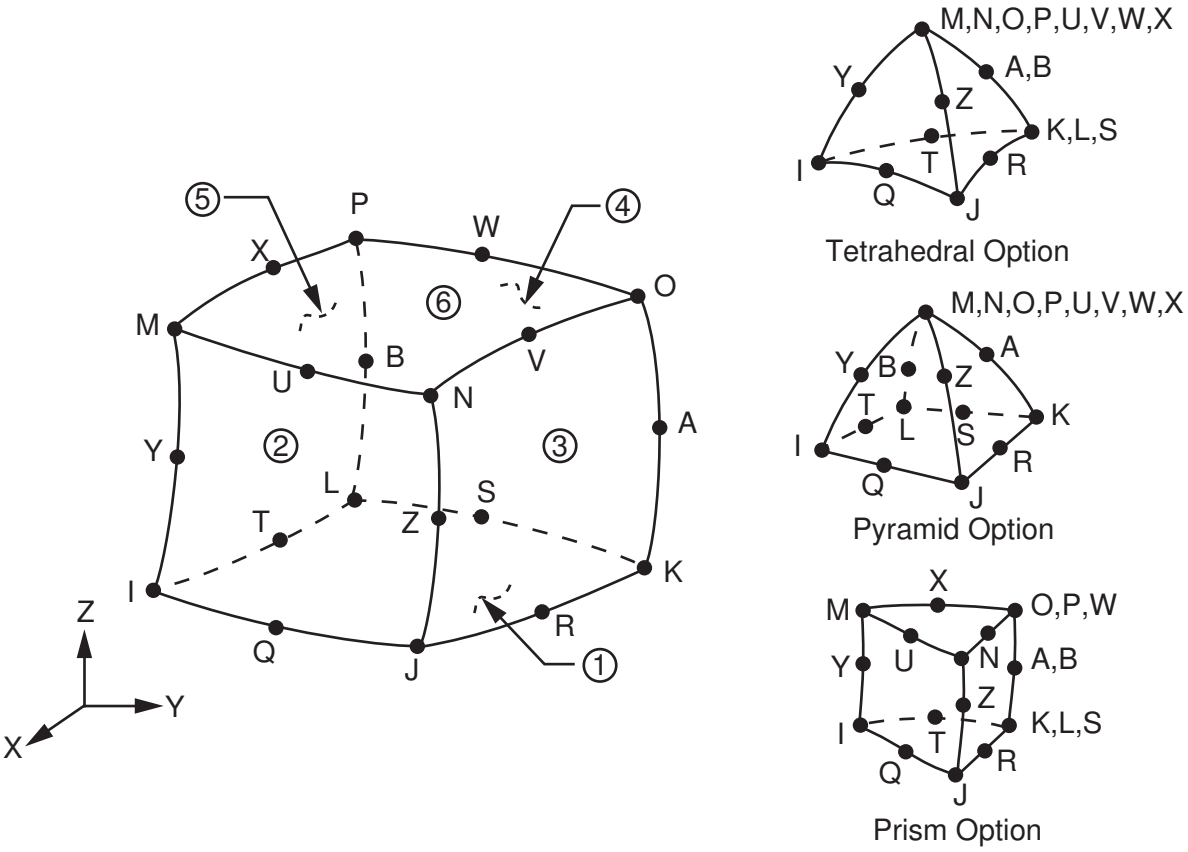
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID231 Element Description

SOLID231 is a 3-D 20-node, current-based electric element. The element has one degree of freedom, voltage, at each node. It can tolerate irregular shapes without much loss of accuracy. SOLID231 elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See *Section 14.231: SOLID231 - 3-D 20-Node Electric Solid* in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID231 Geometry



SOLID231 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID231 Geometry"*. The element is defined by 20 node points and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A pyramid-shaped element and a tetrahedral-shaped element may also be formed as shown in *Figure 1, "SOLID231 Geometry"*.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *SOLID231 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID231 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Special Features
Birth and death

KEYOPT
None

SOLID231 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID231 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID231 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*
TEMP	Temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
EF:X, Y, Z, SUM	Electric field components and vector magnitude	*1*	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	*1*	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, Z, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*5*] [*6*]	*1*	*1*
SENE:	Stored electric energy [*6*]	*1*	*1*
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	*1*	*1*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].
4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Table 2, "SOLID231 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID231 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "SOLID231 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 SOLID231 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DZ	NMISC	3
DSUM	NMISC	4
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID231 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID231 Geometry"* or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements in meshing transition zones.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.

SOLID231 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

SOLID232

3-D 10-Node Tetrahedral Electric Solid

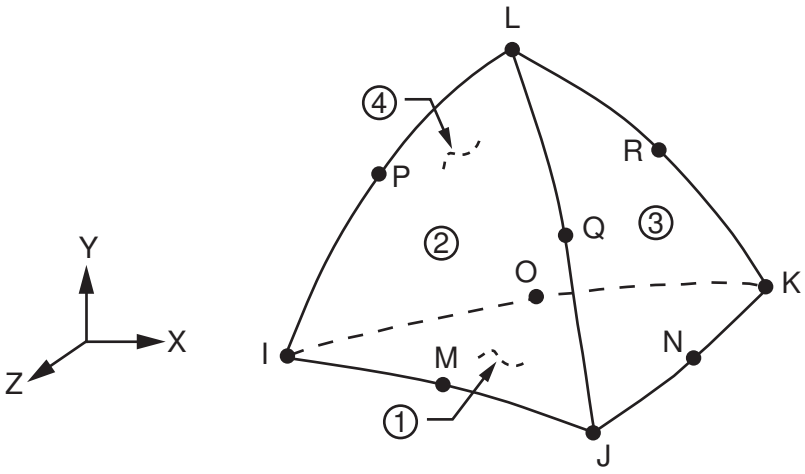
MP <> <> <> <> <> <> <> EM <> <> PP <>
Product Restrictions

SOLID232 Element Description

SOLID232 is a 3-D, 10-node, current-based electric element. It is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, voltage, at each node.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See *Section 14.232: SOLID232 - 3-D 10-Node Tetrahedral Electric Solid* in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

Figure 1 SOLID232 Geometry



SOLID232 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SOLID232 Geometry"*. The element is defined by 10 node points and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in *Section 2.3: Coordinate Systems*. Properties not input default as described in *Section 2.4: Linear Material Properties*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in *SOLID232 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SOLID232 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features
Birth and death

KEYOPTS
None

SOLID232 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in *Table 1, "SOLID123 Element Output Definitions"*

The element output directions are parallel to the element coordinate system. A general description of solution output is given in *Section 2.2: Solution Output* in the *Elements Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 SOLID123 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	*2*

Name	Definition	O	R
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
LOC	Output location (X, Y, Z)	*1*	-
EF:X, Y, Z, SUM	Electric field components and vector magnitude	*1*	*1*
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	*1*	*1*
JS:X, Y, Z, SUM	Current density components and vector magnitude [*3*]	*1*	*1*
JT:X, Y, Z, SUM	Conduction current density components and magnitude [*3*]	*1*	*1*
JHEAT:	Joule heat generation rate per unit volume [*4*] [*5*] [*6*]	*1*	*1*
SENE:	Stored electric energy [*6*]	*1*	*1*
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	*1*	*1*

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].
4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.
5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].
6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Table 2, "SOLID232 Item and Sequence Numbers" lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 2, "SOLID232 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 1, "SOLID123 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

Table 2 SOLID232 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DZ	NMISC	3

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DSUM	NMISC	4
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID232 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in *Figure 1, "SOLID232 Geometry"* or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.

SOLID232 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

SURF251

2-D Radiosity Surface

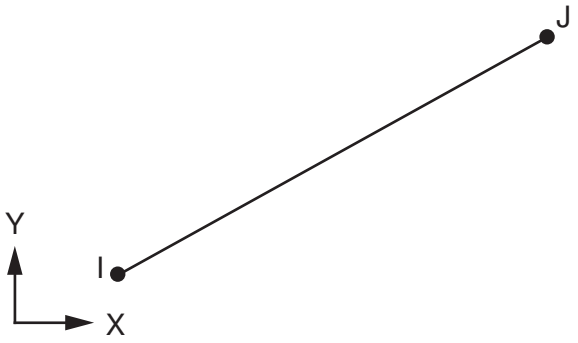
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SURF251 Element Description

SURF251 is used for radiation surface loads and can be used only with the radiosity solver method. It can be overlaid onto a face of any 2-D thermal solid element that supports temperature DOF, except FLUID141 elements. This element is applicable to 2-D thermal analyses (planar or axisymmetric). Various other loads and surface effects may exist simultaneously (e.g., SURF151 and SURF153 and SURF251 may be applied on the same solid element faces to support convection heat flux and radiation heat flux loads).

This element can be created only by the **RSURF** command. The underlying solid surface must also have the RDSF flag.

Figure 1 SURF251 Geometry

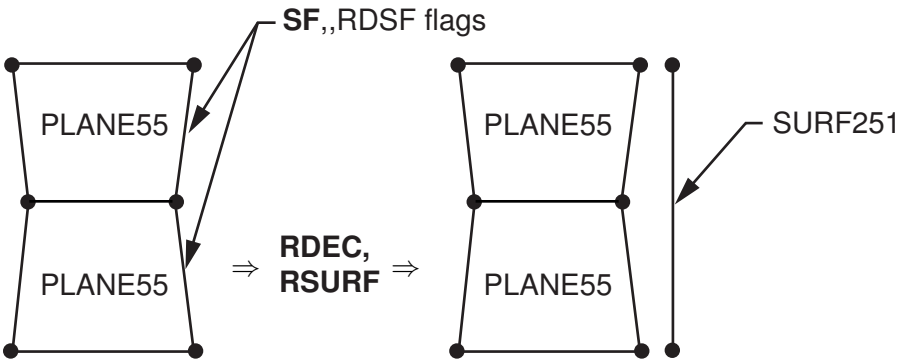


SURF251 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF251 Geometry"*. The element is defined by two nodes, regardless of the underlying solid element.

You would typically generate SURF251 elements via the **RSURF** command, creating elements which are coincident with the solid element surface. However, if you are using decimation (**RDEC**), then the surface elements created will not coincide with the underlying solid element topology. See *Figure 2, "SURF251 Elements Without Coincident Nodes"*. Symmetrical SURF251 elements (produced when using the symmetry options [**RSYMM**]) can have no underlying solid elements. The **RSURF** command always produces extra nodes to define the SURF251 topology as shown in *Figure 2, "SURF251 Elements Without Coincident Nodes"*, regardless if **RDEC** is used.

Figure 2 SURF251 Elements Without Coincident Nodes



You cannot apply any loads on this element. During solution, the element extracts the temperature of the solid element and computes the radiation heat flux, which is transferred back as a surface load to the solid element.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

SURF251 Input Summary

Element Name
SURF251

Nodes
I, J

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

SURF251 Output Data

*Table 1, "SURF251 Item and Sequence Numbers for the **ETABLE** and **ESOL** Commands"* lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SURF151 Item and Sequence Numbers"*:

Name
output quantity as defined in *Table 1, "SURF251 Item and Sequence Numbers for the **ETABLE** and **ESOL** Commands"*

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

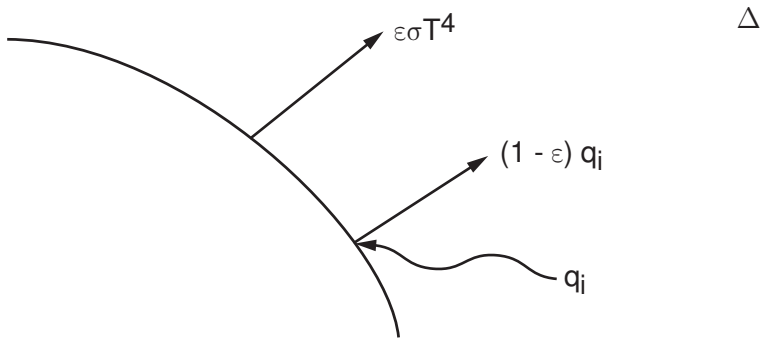
I,J
sequence number for data at nodes I and J

Table 1 SURF251 Item and Sequence Numbers for the ETABLE and ESOL Commands

Name	Item	E
CENTROID X	NMISC	1
CENTROID Y	NMISC	2
CENTROID Z	NMISC	3
AREA	NMISC	4
TEMP	NMISC	5
EMISSIVITY	NMISC	6
Net radiation heat flux	NMISC	7
Emitted radiation heat flux	NMISC	8
Reflected radiation heat flux	NMISC	9
Incident radiant heat flux	NMISC	10
Enclosure No.	NMISC	18

The net radiation heat flux is the sum of the directly emitted radiation flux [$\varepsilon\sigma T^4$] plus the reflected radiation flux [$(1-\varepsilon)q_i$] minus the incoming radiation [q_i], as shown in *Figure 3, "Net Radiation Heat Flux"*.

Figure 3 Net Radiation Heat Flux



Net outgoing radiant heat flux = $\underline{\varepsilon\sigma T^4} + \underline{(1-\varepsilon)q_i} - \underline{q_i}$

emitted, reflected, incident

SURF251 Assumptions and Restrictions

- The element must not have a zero length.

SURF251 Product Restrictions

None

SURF252

3-D Thermal Radiosity Surface

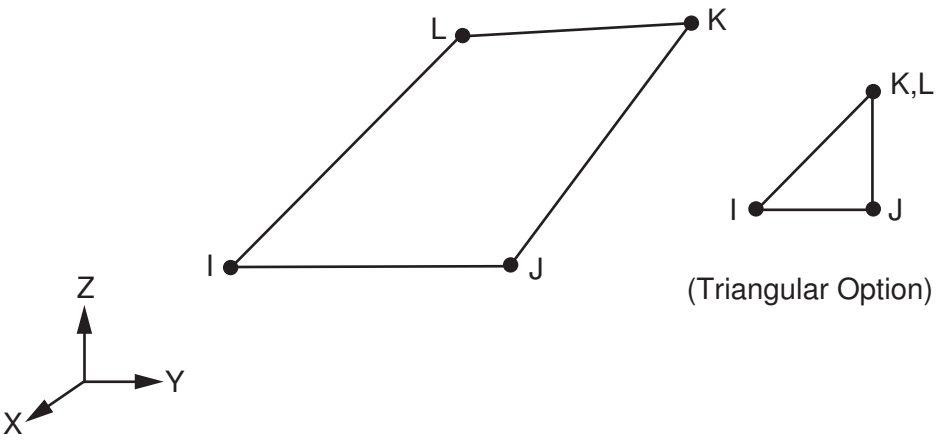
MP ME <> PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

SURF252 Element Description

SURF252 is used for radiation surface loads and can be used only with the radiosity solver method. It can be overlaid onto a face of any 3-D thermal solid or shell element that supports temperature DOF, except FLUID142 elements. This element is applicable to 3-D thermal analyses. Various other loads and surface effects may exist simultaneously (e.g., SURF152 and SURF154 and SURF252 may be applied on the same solid element faces to support convection heat flux and radiation heat flux loads).

This element can be created only by the **RSURF** command. The surface must also have the RDSF flag.

Figure 1 SURF252 Geometry

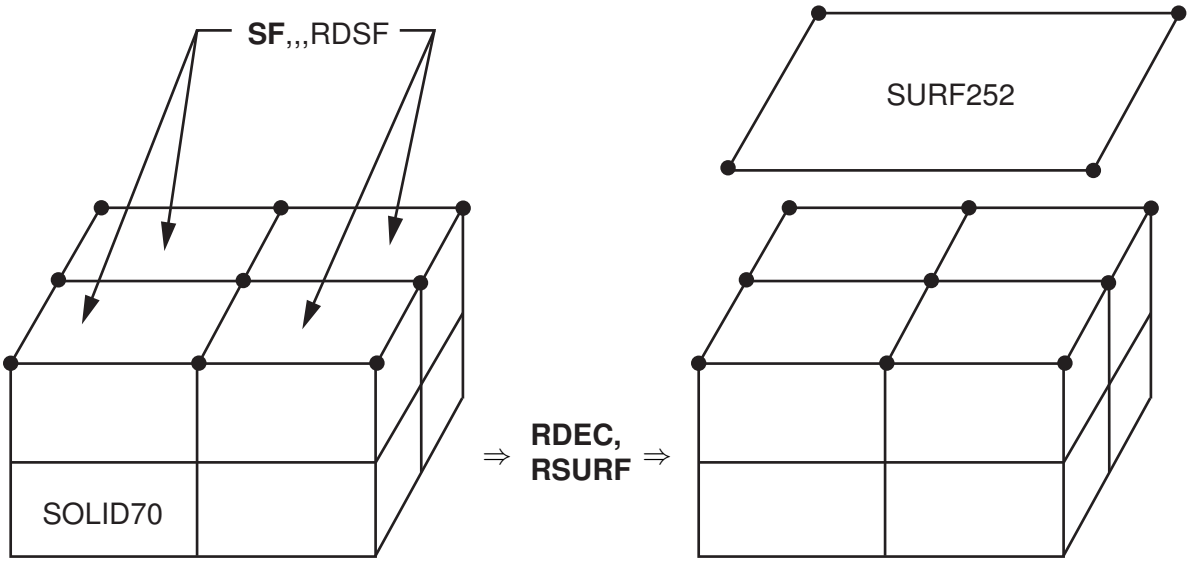


SURF252 Input Data

The geometry, node locations, and the coordinate system for this element are shown in *Figure 1, "SURF252 Geometry"*. The element is defined by three or four nodes, regardless of the underlying solid element.

You would typically generate SURF252 elements via the **RSURF** command, creating elements which are coincident with the solid element surface. However, if you are using decimation (**RDEC**), then the surface elements created will not coincide with the underlying solid element topology. See *Figure 2, "SURF252 Elements Without Coincident Nodes"*. Symmetrical SURF252 elements (produced when using the symmetry options [**RSYMM**]) will have no underlying solid elements. The **RSURF** command always produces extra nodes to define the SURF252 topology as shown in *Figure 2, "SURF252 Elements Without Coincident Nodes"*, regardless if **RDEC** is used.

Figure 2 SURF252 Elements Without Coincident Nodes



You cannot apply any loads on this element. During solution, the element extracts the temperature of the solid element and computes the radiation heat flux, which is transferred back as a surface load to the solid element.

The next table summarizes the element input. *Section 2.1: Element Input* gives a general description of element input.

SURF252 Input Summary

Element Name
SURF252

Nodes
I, J, K, L

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

SURF252 Output Data

*Table 1, "SURF252 Item and Sequence Numbers for the **ETABLE** and **ESOL** Commands"* lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic*

Analysis Guide and *Section 2.2.2.2: The Item and Sequence Number Table* in this manual for more information. The following notation is used in *Table 3, "SURF151 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "SURF252 Item and Sequence Numbers for the **ETABLE** and **ESOL** Commands"*

Item
: predetermined Item label for **ETABLE** command

E
: sequence number for single-valued or constant element data

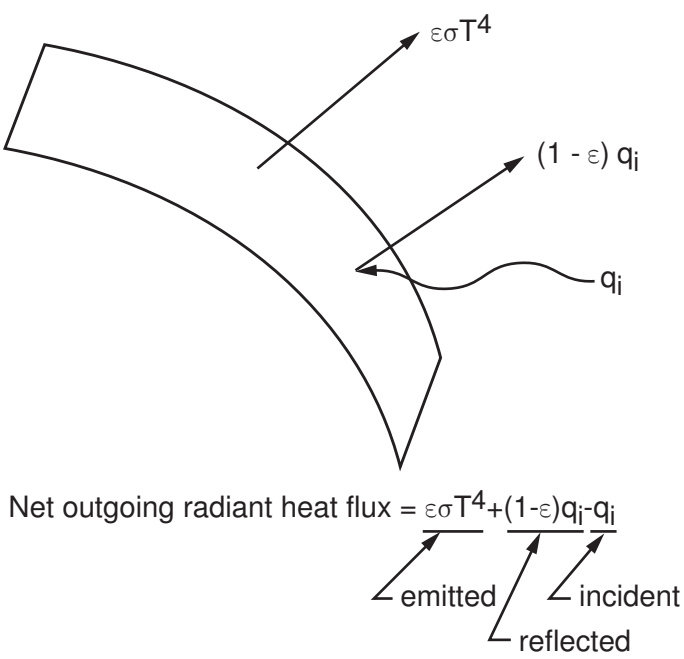
I,J
: sequence number for data at nodes I and J

Table 1 SURF252 Item and Sequence Numbers for the ETABLE and ESOL Commands

Name	Item	E
CENTROID X	NMISC	1
CENTROID Y	NMISC	2
CENTROID Z	NMISC	3
AREA	NMISC	4
TEMP	NMISC	5
EMISSIVITY	NMISC	6
Net radiation heat flux	NMISC	7
Emitted radiation heat flux	NMISC	8
Reflected radiation heat flux	NMISC	9
Incident radiant heat flux	NMISC	10
Enclosure No.	NMISC	18

The net radiation heat flux is the sum of the directly emitted radiation flux [$\varepsilon\sigma T^4$] plus the reflected radiation flux [$(1-\varepsilon)q_i$] minus the incoming radiation [q_i], as shown in *Figure 3, "Net Radiation Heat Flux"*.

Figure 3 Net Radiation Heat Flux



SURF252 Assumptions and Restrictions

- The element must not have a zero length.

SURF252 Product Restrictions

None

REINF265

3-D Smeared Reinforcing

MP ME ST PR PRN <> <> <> <> <> <> PP <>
Product Restrictions

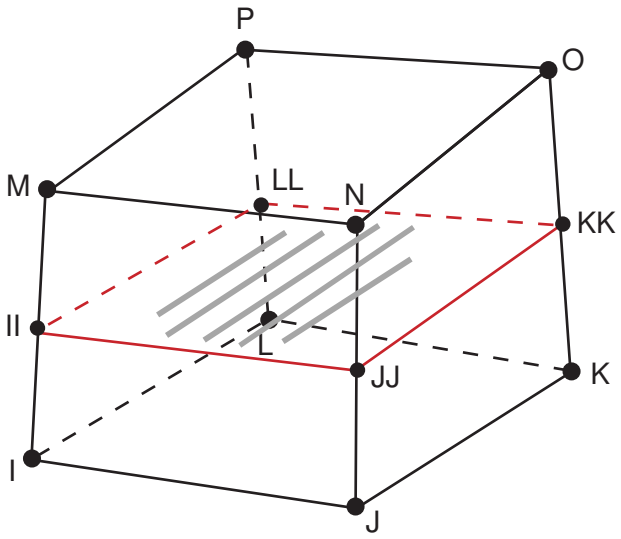
REINF265 Element Description

Use REINF265 with standard 3-D solid and shell elements (referred to here as the *base elements*) to provide extra reinforcing to those elements.

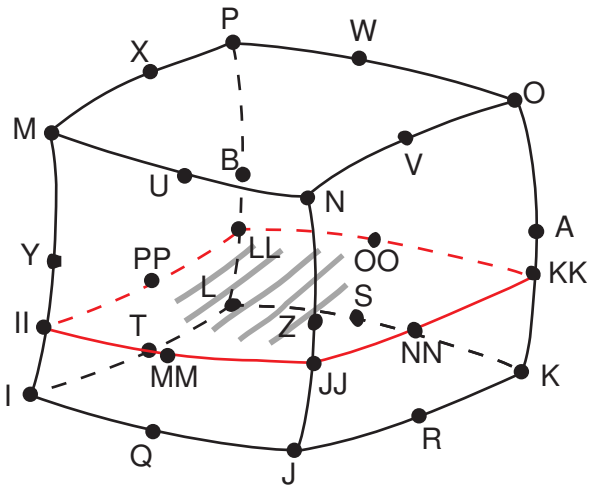
The element uses a smeared approach and is suitable for modeling evenly spaced reinforcing fibers that appear in layered form. Each reinforcing layer contains a cluster of fibers with unique orientation, material, and cross-section area, and is simplified as a homogenous membrane having unidirectional stiffness. You can specify multiple layers of reinforcings in one REINF265 element. The nodal locations, degrees of freedom, and connectivity of the REINF265 element are identical to those of the base element.

REINF265 has plasticity, stress stiffening, creep, large deflection, and large strain capabilities. See REINF265 in the *Theory Reference for ANSYS and ANSYS Workbench* for more details about this element.

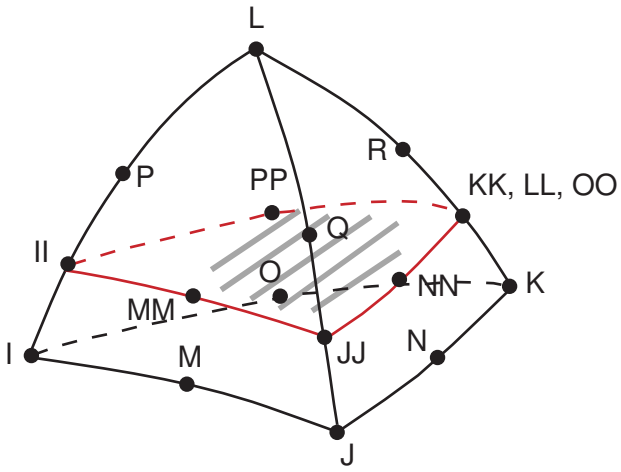
Figure 1 REINF265 Geometry



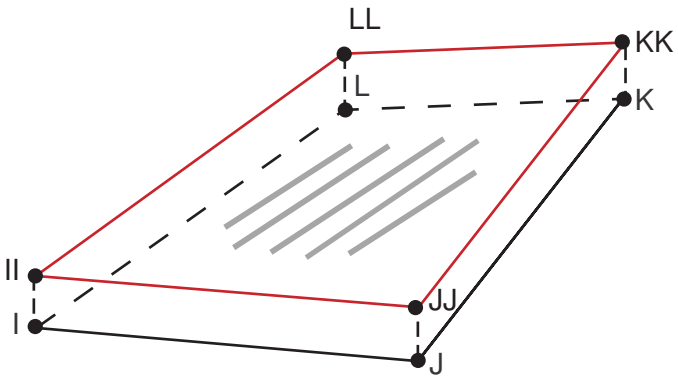
3-D 8-Node Solid or Solid Shell



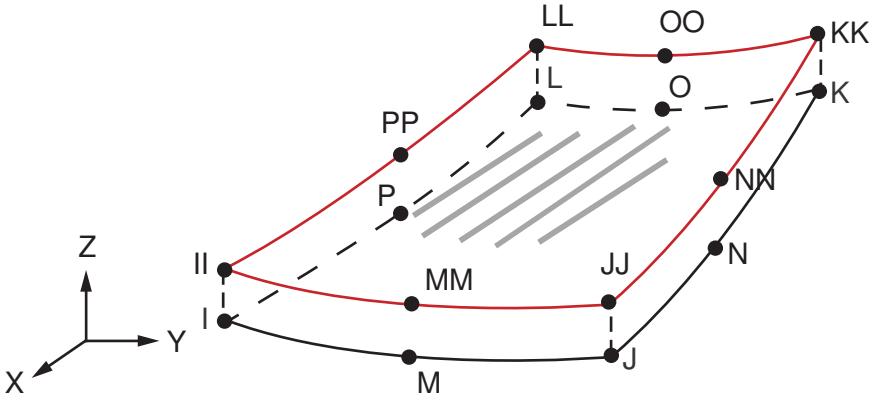
3-D 20-Node Solid



3-D 10-Node Tetrahedral Solid

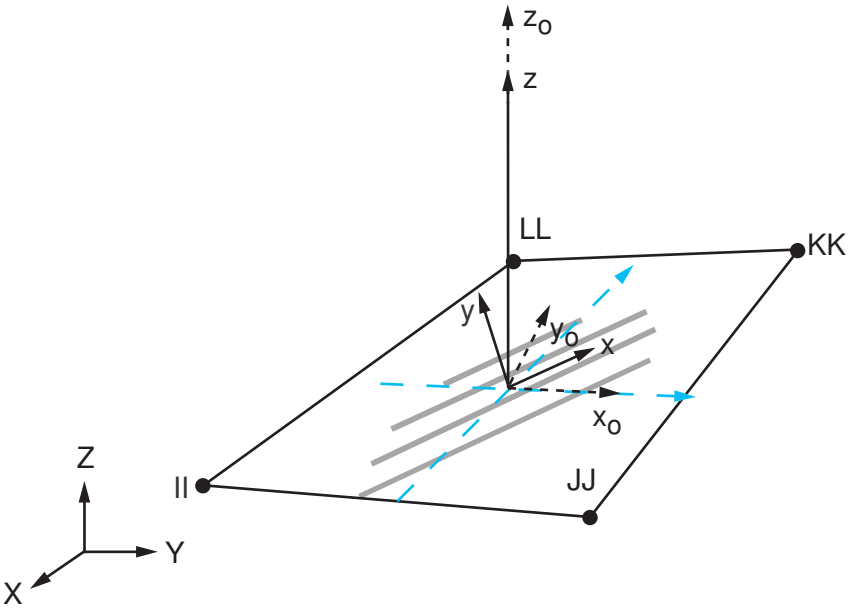


3-D 4-Node Shell



3-D 8-Node Shell

Figure 2 REINF265 Coordinate System



X = Layer x-axis if the local coordinate system reference number (**SECDATA**,,,*ESYS*) for the layer is specified
X_0 = Layer x-axis if the local coordinate system reference number is *not* specified

REINF265 Input Data

The geometry and nodal locations for this element are shown in *Figure 1, "REINF265 Geometry"*. The REINF265 element and its base element share the same nodes and element connectivity.

You can easily create REINF265 elements from the selected base elements via the **EREINF** command. Section commands (**SECTYPE** and **SECDATA**) define the material ID, cross-section area, spacing, location, and orientation of reinforcing fibers.

The equivalent thickness *h* of the smeared reinforcing layer is given by

$$h = A / S$$

where *A* is the cross-section area of a single fiber, and *S* is the distance between two adjacent fibers. You can define up to 250 reinforcing layers within one section.

The coordinate systems for one reinforcing layer are shown in *Figure 2, "REINF265 Coordinate System"*. Each reinforcing layer is indicated by its intersection points (II, JJ, KK, LL for linear base elements, and II, JJ, KK, LL, MM, NN, OO, PP for quadratic base elements) with the base elements. Fibers in this layer are always parallel to the first coordinate axis x. The x axis is default to the first parametric direction S1 at the center of the layer. The default axis is defined as

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{4}\right)\left[-\{x\}^{II} + \{x\}^{JJ} + \{x\}^{KK} - \{x\}^{LL}\right]$$

$\{x\}^{II}, \{x\}^{JJ}, \{x\}^{KK}, \{x\}^{LL}$ = global nodal coordinates

You can reorient the default layer coordinate system by projecting a local coordinate system (**LOCAL**) to the layer plane. One local coordinate system is allowed for each layer. The local coordinate system reference number is given via the **SECDATA** command.

You can further rotate the layer coordinate system by angle THETA (in degrees) for each layer. The value of THETA is also provided for each layer via **SECDATA**. See the **/PSYMB** command documentation for more information about the visualization of fiber orientations.

The REINF265 element does not accept element loading. Apply element loading only to the base element. The temperature of the REINF265 element is identical to the temperature of the base element.

A summary of the element input follows.

REINF265 Input Summary

Nodes

Same as those of the base element, as shown:

Base Element	REINF265 Nodes
3-D 8-Node Solid or Solid Shell	I,J,K,L,M,N,O,P
3-D 20-Node Solid	I,J,K,L,M,N,O,P,Q,R,S,T,U,V,W,X,Y,Z,A,B
3-D 10-Node Tetrahedral Solid	I,J,K,L,M,N,O,P,Q,R
3-D 4-Node Shell	I,J,K,L
3-D 8-Node Shell	I,J,K,L,M,N,O,P

Degrees of Freedom

Same as those of the base element, as shown:

Base Element	REINF265 DOFs
3-D 8-Node Solid or Solid Shell	UX, UY, UZ
3-D 20-Node Solid	UX, UY, UZ
3-D 10-Node Tetrahedral Solid	UX, UY, UZ
3-D 4-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 8-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

None

Body Loads

Temperatures --
Same as those of the base element

Special Features

Plasticity
Viscoelasticity
Viscoplasticity
Creep
Stress stiffening
Large deflection
Large strain
Initial stress import
Birth and death
Supports the following types of data tables associated with the **TB** command: BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, RATE, CREEP, PRONY, SHIFT, PLASTIC, and USER.

See the *Theory Reference for ANSYS and ANSYS Workbench* for details about the material models.

KEYOPT(10)
User-defined initial stress:

0 --
No user subroutine to provide initial stress (default)

1 --
Read initial stress data from user subroutine USTRESS

See the *Guide to ANSYS User Programmable Features* for user-written subroutines

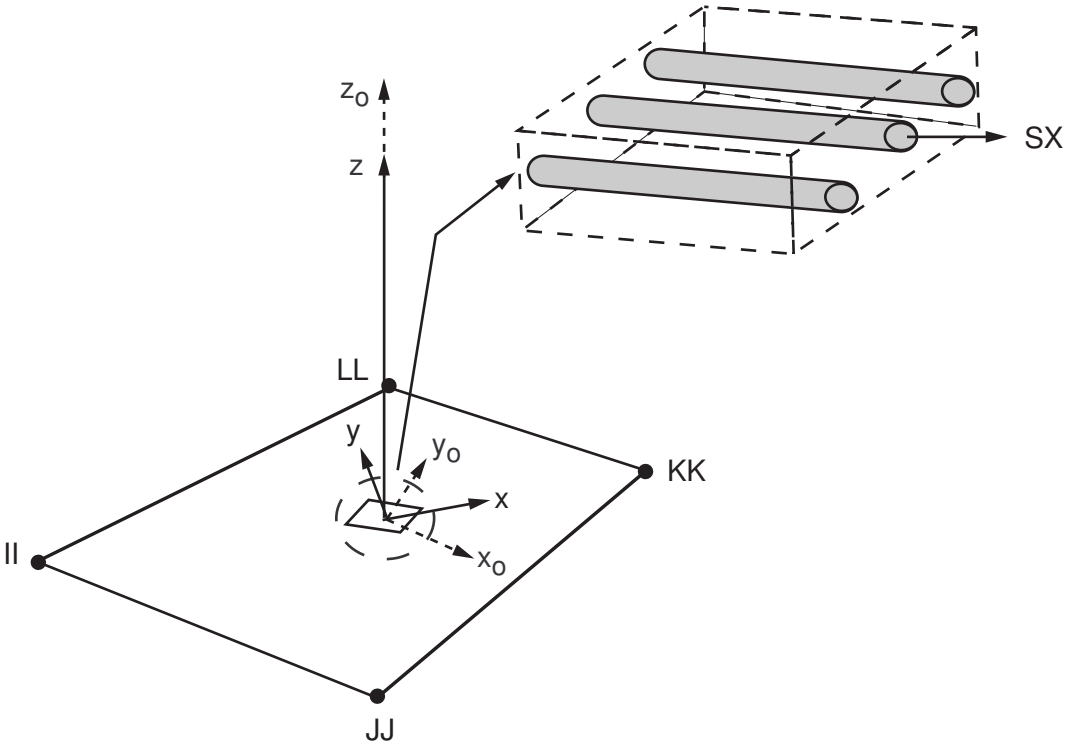
REINF265 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 1, "REINF265 Element Output Definitions"*.

The axial stress component is illustrated in *Figure 3, "REINF265 Stress Output"*.

Figure 3 REINF265 Stress Output



X = Layer x-axis if the local coordinate system reference number (**SECDATA**,,,,*ESYS*) for the layer is specified
X_0 = Layer x-axis if the local coordinate system reference number is *not* specified

Unlike layered solid or shell elements (such as SHELL181), the REINF265 element always outputs the element solution for *all* reinforcing layers. You can select solution items for a specific reinforcing layer via the **LAYER** command for listing and visualization. See the *Basic Analysis Guide* for ways to review results.

To inspect REINF265 element results, select only REINF265 element results *or* adjust translucency level of the base elements before executing any plotting command. REINF265 display options are also available directly via the ANSYS GUI (**Main Menu> Preprocessor> Sections> Reinforcing> Display Options**).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 1 REINF265 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes (as shown in *REINF265 Input Summary*)	-	Y
MAT	Material number	-	Y
AREA	Averaged cross-section area of reinforcing fibers	-	Y

Name	Definition	O	R
SPACING	Averaged distance between two adjacent fibers	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Center location	-	*3*
TEMP	T1, T2, T3, T4 for reinforcing layer 1; T5, T6, T7, T8 for reinforcing layer 2; ending with temperatures for the last reinforcing layer NL (4*NL maximum)	-	Y
S:X	Axial stresses	*2*	Y
EPEL:X	Axial elastic strains	*2*	Y
EPTH:X	Axial thermal strains	*2*	Y
EPPL:X	Axial plastic strains	*2*	*1*
EPCR:X	Axial creep strains	*2*	*1*
EPTO:X	Total axial mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	*1*
NL:CREQ	Accumulated equivalent creep strain	-	*1*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	*1*
NL:PLWK	Plastic work	-	*1*
N11	Averaged axial force	-	Y
LOCI:X, Y, Z	Integration point locations	-	*4*

1. Nonlinear solution output if the element has a nonlinear material.
2. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.

Table 2, "REINF265 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information. The following notation is used in *Table 2, "REINF265 Item and Sequence Numbers"*:

Name
: output quantity as defined in *Table 1, "REINF265 Element Output Definitions"*

Item
: predetermined Item label for **ETABLE**

E
: sequence number for single-valued or constant element data

Table 2 REINF265 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
N11	SMISC	(i - 1) * 3 + 1
AREA	SMISC	(i - 1) * 3 + 2
SPACING	SMISC	(i - 1) * 3 + 3

The i value (where $i = 1, 2, 3, ..., NL$) represents the reinforcing layer number of the element. NL is the maximum reinforcing layer number ($1 \leq NL \leq 250$).

REINF265 Assumptions and Restrictions

- Zero-volume elements are invalid.
- This element can be used only with base element types SHELL181, SHELL281, SOLID185, SOLID186, SOLID187, and SOLSH190.
- A valid base element must be present for each REINF265 element.
- The reinforcing element is firmly attached to its base element. No relative movement between the reinforcing element and the base is allowed.
- Through-thickness reinforcing is not permitted in shells and layered solid elements.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). ANSYS ignores stress-stiffening effects in geometrically linear analyses (**NLGEOM**,OFF) when applied (via the **SSTIF**,ON command). You can activate *prestress* effects via the **PSTRES** command.
- The element supports a maximum of 250 reinforcing layers.

REINF265 Product Restrictions

None.

SHELL281

8-Node Finite Strain Shell

MP ME ST PR PRN DS DSS <> <> <> <> PP <>
Product Restrictions

SHELL281 Element Description

SHELL281 is suitable for analyzing thin to moderately-thick shell structures. It is an 8-node element with six degrees of freedom at each node: translations in the x, y, and z axes, and rotations about the x, y, and z-axes. (When using the membrane option, the element has translational degrees of freedom only.)

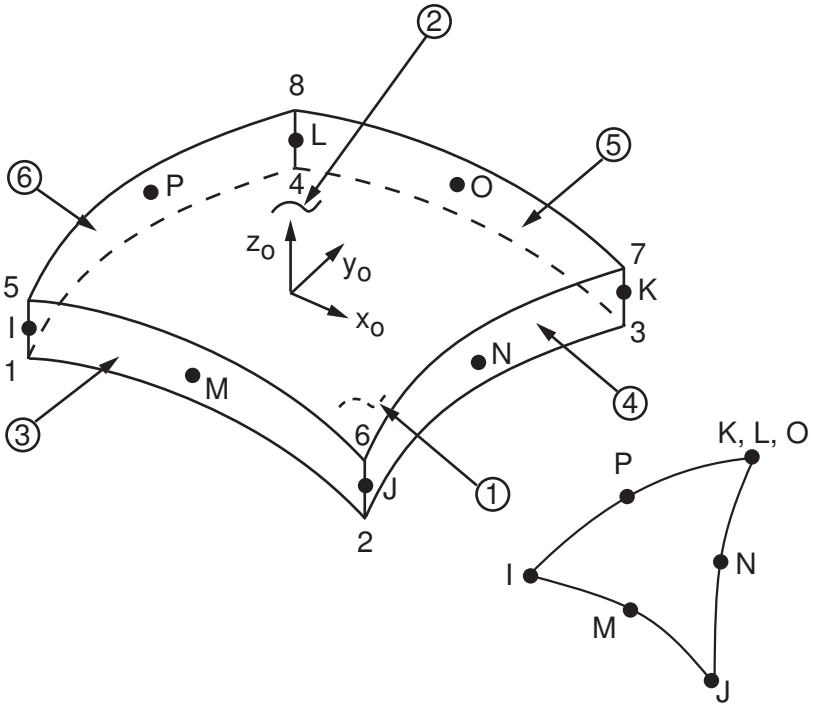
For better accuracy, ANSYS recommends quadrilateral shaped elements. Use degenerate triangular shapes sparingly.

SHELL281 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. The element accounts for follower (load stiffness) effects of distributed pressures.

SHELL281 may be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory (usually referred to as Mindlin-Reissner shell theory).

SHELL281 can be used instead of SHELL91, SHELL93, and SHELL99 for most problems.

Figure 1 SHELL281 Geometry



x_o = Element x-axis if ESYS is not provided.

x = Element x-axis if ESYS is provided.

SHELL281 Input Data

The geometry, node locations, and the element coordinate system for this element are shown in *Figure 1, "SHELL281 Geometry"*. The element is defined by eight nodes: I, J, K, L, M,N, O and P. Midside nodes may not be removed from this element. See *Section 2.2.2: Quadratic Elements (Midside Nodes)* in the *Modeling and Meshing Guide* for additional information about the use of midside nodes. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small. To define the thickness and other information, you can use either real constants or section definition (and a section can be partially defined using data from a FiberSIM `.xml` file). The option of using real constants is available only for single-layer shells. If a SHELL281 element references both real constant set data and a valid shell section type, real constant data is ignored.

SHELL281 also accepts the preintegrated shell section type (**SECTYPE**,,GENS). When the element is associated with the GENS section type, thickness or material definitions are not required. For more information, see *Section 17.3: Using Preintegrated General Shell Sections*.

Thickness Definition Using Real Constants

The thickness of the shell may be defined at each of its nodes. The thickness is assumed to vary smoothly over the area of the element. If the element has a constant thickness, only TK(I) needs to be input. If the thickness is not constant, all four thicknesses must be input.

Layered Section Definition Using Section Commands

Alternatively, the shell thickness and more general properties may be specified using section commands. SHELL281 may be associated with a shell section (**SECTYPE**). Shell section is a more general method to define shell construction than the real constants option. Shell section commands allow for layered composite shell definition, and provide the input options for specifying the thickness, material, orientation and number of integration points through the thickness of the layers. Note that a single layer shell is not precluded using shell section definition, but provides more flexible options such as the use of the ANSYS function builder to define thickness as a function of global coordinates and the number of integration points used.

You may designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer when using section input. When only one, the point is always located midway between the top and bottom surfaces. If three or more points, two points are located on the top and bottom surfaces respectively and the remaining points are distributed at equal distances between the two points. An exception occurs when designating five points, where the quarter point locations are moved five percent toward their nearest layer surface to agree with the locations selected with real constant input. The default number of integration points for each layer is 3. However, when a single layer is defined and plasticity is present, the number of integration points will be changed to a minimum of 5 during solution. Note that when Real Constants are used, ANSYS uses 5 points of integration and Sections will produce a comparable solution.

Other Input

The default orientation for this element has the S_1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element, which connects the midsides of edges LI and JK. In the most general case, the axis can be defined as:

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where:

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{4}\right)\left[-\{x\}^I + \{x\}^J + \{x\}^K - \{x\}^L\right]$$

$\{x\}^I, \{x\}^J, \{x\}^K, \{x\}^L$ = global nodal coordinates

For undistorted elements, the default orientation is the same as described in *Section 2.3: Coordinate Systems* (the first surface direction is aligned with the IJ side).

The first surface direction S_1 can be rotated by angle THETA (in degrees) as a real constant for the element or for using the **SECDATA** command. For an element, you can specify a single value of orientation in the plane of the element. Layer-wise orientation is possible when section definition is used.

You can also define element orientation via the **ESYS** command. For more information, see *Section 2.3: Coordinate Systems*.

The element supports degeneration into a triangular form; however, use of the triangular form is *not* recommended, except when used as mesh filler elements or with the membrane option (KEYOPT(1) = 1). The triangle form is generally more robust when using the membrane option with large deflections.

To evaluate stresses and strains on exterior surfaces, use KEYOPT(1) = 2. When used as overlaid elements on the faces of 3-D elements, this option is similar to the surface stress option (described in the *Theory Reference for ANSYS and ANSYS Workbench*), but is more general and applicable to nonlinear analysis. The element used with this option does not provide any stiffness, mass, or load contributions. This option should only be used in single-layered shells. Irrespective of other settings, SHELL181 provides stress and strain output at the four in-plane integration points of the layer.

SHELL281 uses a penalty method to relate the independent rotational degrees of freedom about the normal (to the shell surface) with the in-plane components of displacements. The ANSYS program chooses an appropriate penalty stiffness by default; however, you can change the default value if necessary via the tenth real constant (drill stiffness factor). The value of this real constant is the scaling parameter for the default penalty stiffness. Using a higher value could contribute to a larger nonphysical energy content in the model; therefore, use caution when changing the default. When using the section definition with SHELL281, drill stiffness factor may be specified via the **SECCONTROLS** command.

Element loads are described in *Section 2.8: Node and Element Loads*. Pressures may be input as surface loads on the element faces as shown by the circled numbers in *Figure 1, "SHELL281 Geometry"*. Positive pressures act into the element. Edge pressures are input as force per unit length.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers (1-1024 maximum). The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and if exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

SHELL281 includes the effects of transverse shear deformation. The transverse shear stiffness of the element is a 2x2 matrix as shown below:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ \text{sym} & E_{22} \end{bmatrix}$$

In the above matrix, R7, R8, and R9 are real constants 7, 8, and 9. (See *Table 1, "SHELL281 Real Constants"*.) You can override the default transverse shear stiffness values by assigning different values to those real constants. This option is effective for analyzing sandwich shells. Alternatively, the **SECCONTROLS** command you to define transverse shear stiffness values.

For a single-layer shell with isotropic material, default transverse shear stiffnesses are:

$$E = \begin{bmatrix} kGh & \\ & kGh \end{bmatrix}$$

In the above matrix, k = 5/6, G = shear modulus, and h = thickness of the shell.

SHELL281 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties. Only isotropic, anisotropic, and orthotropic linear elastic properties can be input for elasticity. The von Mises isotropic hardening plasticity models can be invoked with BISO (bilinear isotropic hardening), MISO (multilinear isotropic hardening), and NLISO (nonlinear isotropic hardening) options. The kinematic hardening plasticity models can be invoked with BKIN (bilinear kinematic hardening), MKIN and KINH (multilinear kinematic hardening), and CHABOCHE (nonlinear kinematic hardening). Invoking plasticity assumes that the elastic properties are isotropic (that is, if orthotropic elasticity is used with plasticity, ANSYS assumes the isotropic elastic modulus = EX and Poisson's ratio = NUXY).

Hyperelastic material properties (2, 3, 5, or 9 parameter Mooney-Rivlin material model, Neo-Hookean model, Polynomial form model, Arruda-Boyce model, and user-defined model) can be used with this element. Poisson's ratio is used to specify the compressibility of the material. If less than 0, Poisson's ratio is set to 0; if greater than or equal to 0.5, Poisson's ratio is set to 0.5 (fully incompressible).

Both isotropic and orthotropic thermal expansion coefficients can be input using **MP**,ALPX. When used with hyperelasticity, isotropic expansion is assumed.

Issue the **BETAD** command to specify the global value of damping. If **MP**,DAMP is defined for the material number of the element (assigned with the **MAT** command), it is used for the element instead of the value from the **BETAD** command. Similarly, use the **TREF** command to specify the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

KEYOPT(8) = 2 is used to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. Use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

KEYOPT(9) = 1 is used to read initial thickness data from a user subroutine.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*. Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

A summary of the element input is given in *SHELL281 Input Summary*. A general description of element input is given in *Section 2.1: Element Input*.

SHELL281 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0

UX, UY, UZ if KEYOPT(1) = 1

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, ADMSUA
E11, E22, E12, DRILL
See *Table 1, "SHELL281 Real Constants"* for more information.
If a SHELL281 element references a valid shell section type, any real constant data specified is ignored.

Material Properties

EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ

Specify DAMP only once for the element (use **MAT** command to assign material property set). REFT may be provided once for the element, or may be assigned on a per layer basis. See the discussion in *SHELL281 Input Summary* for more details.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +N direction),
face 2 (I-J-K-L) (top, in -N direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

For KEYOPT(1) = 0 (Bending and membrane stiffness):

T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum). Hence, for one-layer elements, eight temperatures are used.

For KEYOPT(1) = 1 (Membrane stiffness only):

T1, T2, T3, T4 for layer 1, T5, T6, T7, T8 for layer 2, similarly for all layers (4*NL maximum). Hence, for one-layer elements, four temperatures are used.

Special Features

Plasticity (PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL)
Hyperelasticity (AHYPER, HYPER)
Viscoelasticity (PRONY, SHIFT)
Viscoplasticity/Creep (CREEP, RATE)
Elasticity (ANEL)

Other material (USER, SDAMP)
Stress stiffening
Large deflection
Large strain
Initial stress import
Nonlinear stabilization
Automatic selection of element technology
Birth and death
Section definition for layered shells and preintegrated shell sections for input of homogenous section stiffnesses

Note

Items in parentheses refer to data tables associated with the **TB** command. See the *Theory Reference for ANSYS and ANSYS Workbench* for details about the material models.

Note

See *Section 2.18: Automatic Selection of Element Technologies* and **ETCONTROL** for more information on selection of element technologies.

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default)

1 --
Membrane stiffness only

2 --
Stress/strain evaluation only

KEYOPT(8)
Specify layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (multi-layer elements) (default)

1 --
Store data for TOP and BOTTOM, for all layers (multi-layer elements)

Note

The volume of data may be considerable.

2 --
Store data for TOP, BOTTOM, and MID for all layers; applies to single- and multi-layer elements

KEYOPT(9)
User thickness option:

0 --
No user subroutine to provide initial thickness (default)

1 --
Read initial thickness data from user subroutine UTHICK

Note

See the *Guide to ANSYS User Programmable Features* for information about user-written subroutines

KEYOPT(10)

User-defined initial stress:

0 --

No user subroutine to provide initial stress (default)

1 --

Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for information about user-written subroutines

Table 1 SHELL281 Real Constants

No.	Name	Description
1	TK(I)	Thickness at node I
2	TK(J)	Thickness at node J
3	TK(K)	Thickness at node K
4	TK(L)	Thickness at node L
5	THETA	Angle of first surface direction, in degrees
6	ADMSUA	Added mass per unit area
7	E_{11}	Transverse shear stiffness[*2*]
8	E_{22}	Transverse shear stiffness[*2*]
9	E_{12}	Transverse shear stiffness[*2*]
10	Drill Stiffness Factor	In-plane rotation stiffness[*1*,*2*]
11	Membrane HG Factor	Membrane hourglass control factor[*1*,*2*]
12	Bending HG Factor	Bending hourglass control factor[*1*,*2*]

1. Valid values for these real constants are any positive number. However, we recommend using values between 1 and 10. If you specify 0.0, the value defaults to 1.0.
2. ANSYS provides default values.

*See **SECCONTROLS** command if section definition is used.

SHELL281 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in *Table 2, "SHELL281 Element Output Definitions"*

Several items are illustrated in *Figure 2, "SHELL281 Stress Output"*.

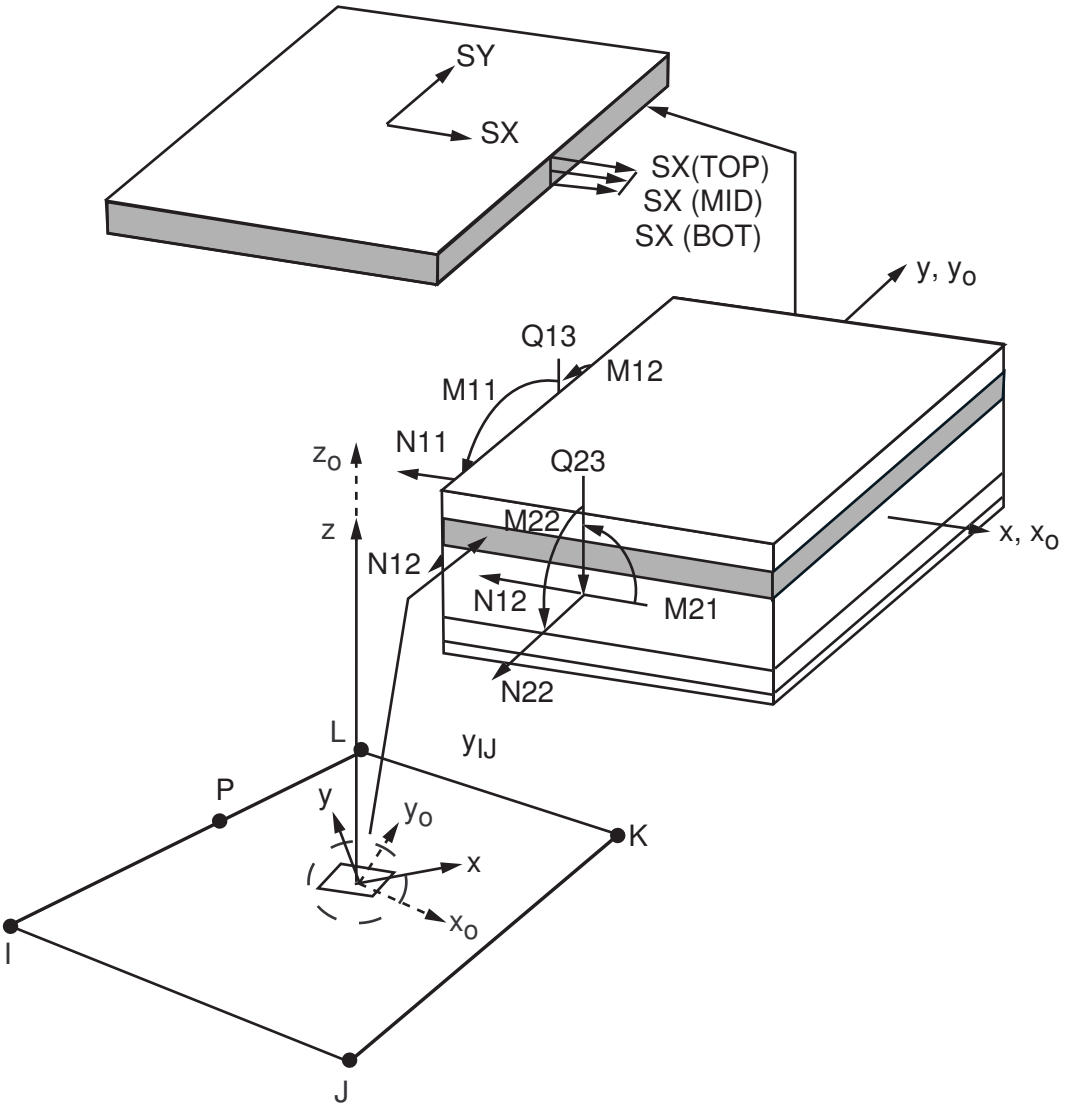
KEYOPT(8) controls the amount of data output to the results file for processing with the **LAYER** command. Interlaminar shear stress is available as SYZ and SXZ evaluated at the layer interfaces. KEYOPT(8) must be set to either

1 or 2 to output these stresses in the POST1 postprocessor. A general description of solution output is given in *Section 2.2: Solution Output*. See the *Basic Analysis Guide* for ways to review results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear forces Q13, Q23 are available only in resultant form: that is, use SMISC,7 (or 8). Likewise, the transverse shear strains, γ_{13} and γ_{23}, are constant through the thickness and are only available as SMISC items (SMISC,15 and SMISC,16, respectively).

SHELL281 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, ANSYS suggests issuing the **OUTRES** command to ensure that the required results are stored in the database.

Figure 2 SHELL281 Stress Output



x_o = Element x-axis if ESYS is not provided.
x = Element x-axis if ESYS is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 2 SHELL281 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	*4*
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J,I; P4 at K,J; P5 at L,K; P6 at I,L	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	*1*
S:X, Y, Z, XY, YZ, XZ	Stresses	*3*	*1*
S:INT	Stress intensity	-	*1*
S:EQV	Equivalent stress	-	*1*
EPEL:X, Y, Z, XY	Elastic strains	*3*	*1*
EPEL:EQV	Equivalent elastic strains [*7*]	*3*	*1*
EPTH:X, Y, Z, XY	Thermal strains	*3*	*1*
EPTH:EQV	Equivalent thermal strains [*7*]	*3*	*1*
EPPL:X, Y, Z, XY	Average plastic strains	*3*	*2*
EPPL:EQV	Equivalent plastic strains [*7*]	*3*	*2*
EPCR:X, Y, Z, XY	Average creep strains	*3*	*2*
EPCR:EQV	Equivalent creep strains [*7*]	*3*	*2*
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	*2*
NL:CREQ	Accumulated equivalent creep strain	-	*2*
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	*2*
NL:PLWK	Plastic work	-	*2*
NL:HPRES	Hydrostatic pressure	-	*2*
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	*2*
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	*8*
Q13, Q23	Transverse shear forces (per unit length)	-	*8*
$\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{12}$	Membrane strains	-	Y
k_{11}, k_{22}, k_{12}	Curvatures	-	*8*

Name	Definition	O	R
γ_{13}, γ_{23}	Transverse shear strains	-	*8*
LOCI:X, Y, Z	Integration point locations	-	*5*
SVAR:1, 2, ... , N	State variables	-	*6*
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Interlaminar shear stress vector sum	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all five section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the USERMAT subroutine and **TB**,STATE are used.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
8. Not available if the membrane element option is used (KEYOPT(1) = 1).

Table 3, "SHELL281 Item and Sequence Numbers" lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and *Section 2.2.2.2: The Item and Sequence Number Table* in this document for more information. The following notation is used in *Table 3, "SHELL281 Item and Sequence Numbers"*:

Name
: output quantity as defined in the *Table 2, "SHELL181 Element Output Definitions"*

Item
: predetermined *Item* label for **ETABLE**

E
: sequence number for single-valued or constant element data

I,J,K,L
: sequence number for data at nodes I, J, K, L

Table 3 SHELL281 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
N11	SMISC	1	-	-	-	-
N22	SMISC	2	-	-	-	-
N12	SMISC	3	-	-	-	-
M11	SMISC	4	-	-	-	-
M22	SMISC	5	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
M12	SMISC	6	-	-	-	-
Q13	SMISC	7	-	-	-	-
Q23	SMISC	8	-	-	-	-
ε_{11}	SMISC	9	-	-	-	-
ε_{22}	SMISC	10	-	-	-	-
ε_{12}	SMISC	11	-	-	-	-
k_{11}	SMISC	12	-	-	-	-
k_{22}	SMISC	13	-	-	-	-
k_{12}	SMISC	14	-	-	-	-
γ_{13}	SMISC	15	-	-	-	-
γ_{23}	SMISC	16	-	-	-	-
THICK	SMISC	17	-	-	-	-
P1	SMISC	-	18	19	20	21
P2	SMISC	-	22	23	24	25
P3	SMISC	-	27	26	-	-
P4	SMISC	-	-	29	28	-
P5	SMISC	-	-	-	31	30
P6	SMISC	-	32	-	-	33

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SHELL281 Assumptions and Restrictions

- Zero area elements are not allowed. (This condition occurs most often whenever the elements are not numbered properly.)
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed (but zero thickness *layers* are allowed).
- In a nonlinear analysis, the solution is terminated if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).
- ANSYS, Inc.recommends against using SHELL281 in triangular form.
- This element works best with a full Newton-Raphson solution scheme (**NROPT**,FULL,ON). For nonlinear problems dominated by large rotations and loading, ANSYS recommends against using the **PRED**,ON command.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers may not change between load steps.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assump-

tions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.

- Transverse shear stiffness of the shell section is estimated by a energy equivalence procedure (of the generalized section forces & strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- A maximum of 250 layers is supported.
- The layer orientation angle has no effect if the material of the layer is hyperelastic.
- If a shell section has only one layer and the number of section integration points is equal to one, or if KEYOPT(1) = 1, then the shell does not have any bending stiffness. In such a case, solver difficulties and convergence problems may occur.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF) when specified via **SSTIF**,ON. Prestress effects can be activated via a **PSTRES** command.
- The through-thickness stress, SZ, is always zero.
- When the element is associated with preintegrated shell sections (**SECTYPE**,,GENS), additional restrictions apply. For more information, see *Section 17.3.2: Considerations for Employing Preintegrated Shell Sections*.
- This element does not support the **RIMPORT** command.

SHELL281 Product Restrictions

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Bibliography

[1.] *Nuclear Systems Material Handbook*. Vol. 1: Design Data, Part 1: Structural Materials, Group 1: High Alloy Steels . U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.

[2.] *Nuclear Systems Material Handbook*. Vol. 1: Design Data, Part 1: Structural Materials, Group 2: Low Alloy Steels, Section 2-2 1/4 CR - 1 Mo. . U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.

[3.] F. Barlat and J. Lian. *"Plastic Behavior and Stretchability of Sheet Metals*. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions". *Int. Journal of Plasticity, 5*. pg. 51-66.

[4.] F. Barlat, D. J. Lege, and J. C. Brem. *"A Six-Component Yield Function for Anistropic Materials". Int. Journal of Plasticity, 7*. pg. 693-712.

[5.] R. Hill. *"A Theory of the Yielding and Plastic Flow of Anisotropic Metals". Proceedings of the Royal Society of London, Series A., Vol. 193*. 1948.

[6.] F. K. Chang and K. Y. Chang. *"A Progressive Damage Model for Laminated Composites Containing Stress Concentration". Journal of Composite Materials, 21*. pg. 834-855. 1987a.

[7.] R. G. Dean. *Evaluation and Development of Water Wave Theories for Engineering Application*. Volume 2, Tabulation of Dimensionless Stream Function Theory Variables, Special Report No. 1, . U. S. Army Corps of Engineers, Coastal Engineering Research Center. Fort Belvoir, VA. November 1974.

[8.] Michael E. McCormick. *Ocean Engineering Wave Mechanices*. Wiley & Sons. New York. 1973.

Index

H

I

L

M

N

P

R

S

T

V