

# SHELL181

## 4-Node Structural Shell

MP ME ST PR PRN DS DSS <> <> <> <> PP VT EME MFS  
[Product Restrictions](#)

## SHELL181 Element Description

SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a four-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 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).

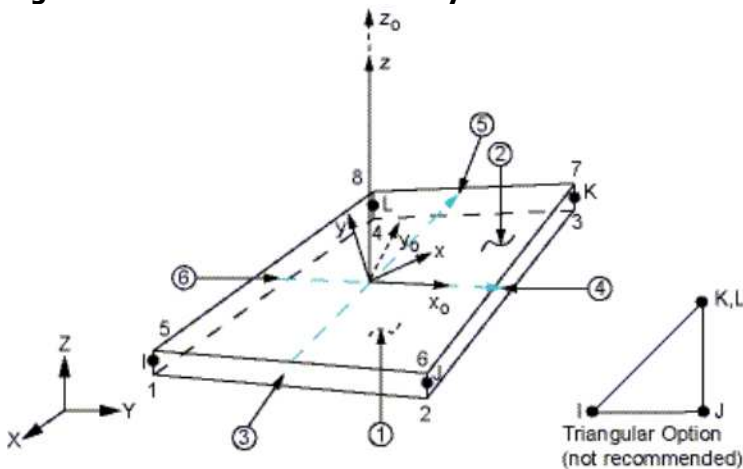
The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small.

See [SHELL181](#) in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details about this element.

## SHELL181 Input Data

The following figure shows the geometry, node locations, and the element coordinate system for this element. The element is defined by shell section information and by four nodes (I, J, K, and L).

**Figure 181.1 SHELL181 Geometry**



$x_0$  = Element x-axis if ESYS is not provided.

$x$  = Element x-axis if ESYS is provided.

## Single-Layer Definition

To define the thickness (and other information), use section definition, as follows:

[SECTYPE](#),,SHELL

[SECDATA](#),THICKNESS, ...

A single-layer shell section definition provides flexible options. For example, you can specify the number of integration points used and the material orientation.

## Multilayer Definition

The shell section commands allow for layered shell definition. Options are available for specifying the thickness, material orientation, and number of integration points through the thickness of the layers.

You can 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 equal distance between the two points. The default number of integration points for each layer is three; however, when a single layer is defined and plasticity is present, the number of integration points is changed to a minimum of five during solution.

The following additional capabilities are available when defining shell layers:

- SHELL181 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.
- You can use the [function tool](#) to define thickness as a function of global/local coordinates or node numbers ([SECFUNCTION](#)).
- You can specify offsets ([SECOFFSET](#)).
- A section can be partially defined using data from a [FiberSIM](#) .xml file ([SECTYPE](#),,SHELL,FIBERSIM).

## 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 and is shown as  $x_0$  in [Figure 181.1](#). In the most general case, the axis can be defined as:

$$S_1 = \frac{\partial \{x\}}{\partial s} / \left( \left| \frac{\partial \{x\}}{\partial s} \right| \right)$$

where:

$$\{x\} = \sum_{i=1}^8 h^i(s,r) \{x\}^i$$

$\{x\}^I, \{x\}^J, \{x\}^K, \{x\}^L$  = global nodal coordinates

For undistorted elements, the default orientation is the same as described in [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) for the layer via the [SECDATA](#) command. For an element, you can specify a single value of orientation in the plane of the element. Layer-wise orientation is supported.

You can also define element orientation via the [ESYS](#) command. See [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 face of 3-D elements, this option is similar to the [surface stress](#) option (described in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#)), 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-layer 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 program chooses an appropriate penalty stiffness by default. A drill stiffness factor can be specified via the [SECCONTROLS](#) command.

Element loads are described in [Node and Element Loads](#). Pressures may be input as surface loads on the element faces as shown by the circled numbers on [Figure 181.1](#). Positive pressures act into the element. Because shell edge pressure are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the shell thickness.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corner 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 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 181.2](#)), 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.

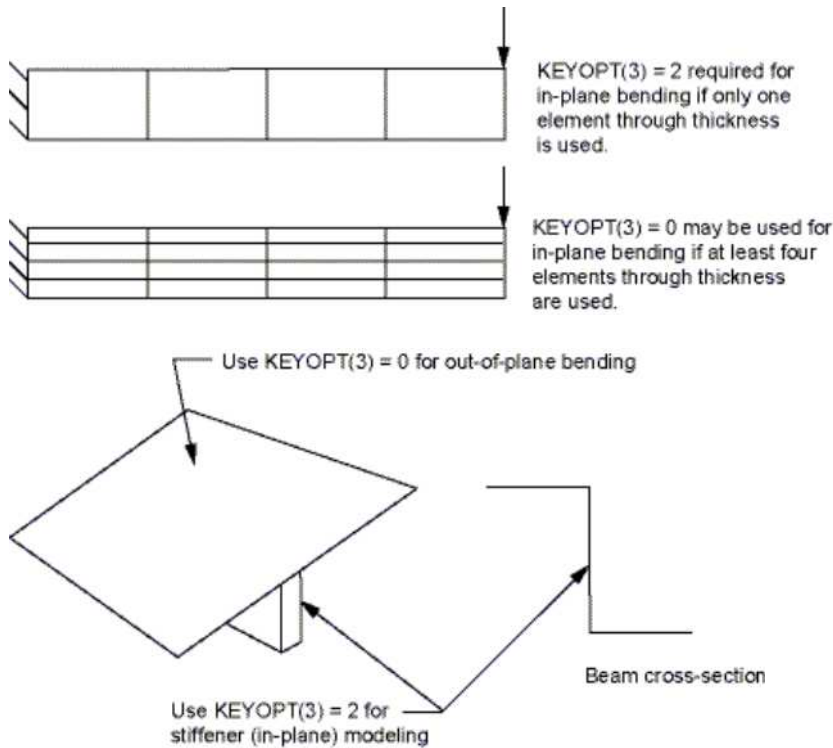
Bilinear elements, when fully integrated, are too stiff in in-plane bending. SHELL181 uses the method of incompatible modes to enhance the accuracy in bending-dominated problems. This approach is also called "extra shapes" or "bubble modes approach". SHELL181 uses the formulation that ensures satisfaction 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 181.2](#)

**Figure 181.2 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. To specify the hourglass stiffness scaling factors, use the [SECCONROLS](#) 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:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ \text{sym} & E_{22} \end{bmatrix}$$

To define transverse shear stiffness values, use the [SECCONROLS](#) command.

For a single-layer shell with isotropic material, default transverse shear stiffnesses are:

$$E = \begin{bmatrix} kGh & 0 \\ 0 & kGh \end{bmatrix}$$

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 stores 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 reads 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 "[Initial State](#)" in the *Basic Analysis Guide*.

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 [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

### 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, BB, CDM)

Viscoelasticity (PRONY, SHIFT)

Viscoplasticity/Creep (CREEP, RATE)

Elasticity (ELASTIC, ANEL)

Other material (USER, SDAMP)

Stress stiffening

Large deflection

Large strain

Initial state

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

Linear perturbation

Items in parentheses refer to data tables associated with the **TB** command. See "[Structures with Material Nonlinearities](#)" in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for details of the material models.

See [Automatic Selection of Element Technologies](#) and **ETCONTROL** for more information about selecting 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)
- 2 -- Store data for TOP, BOTTOM, and MID for all layers; applies to single- and multi-layer elements

---

**Note:** Volume of data may be excessive.

#### 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

## 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 181.1: SHELL181 Element Output Definitions](#)

Several items are illustrated in [Figure 181.3](#).

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 [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,  $\gamma_{13}$  and  $\gamma_{23}$ , are constant through the thickness and are only available as SMISC items (SMISC,15 and SMISC,16, respectively).

ANSYS computes moments (M11, M22, M12) with respect to the shell reference plane. By default, ANSYS adopts the shell midplane as the reference plane. To offset the reference plane to any other specified location, issue the **SECOFFSET** command. When there is a nonzero offset (L) from the reference plane to the midplane, moments with respect to the midplane ( $\overline{M11}, \overline{M22}, \overline{M12}$ ) can be recovered from stress resultants with respect to the reference plane as follows:

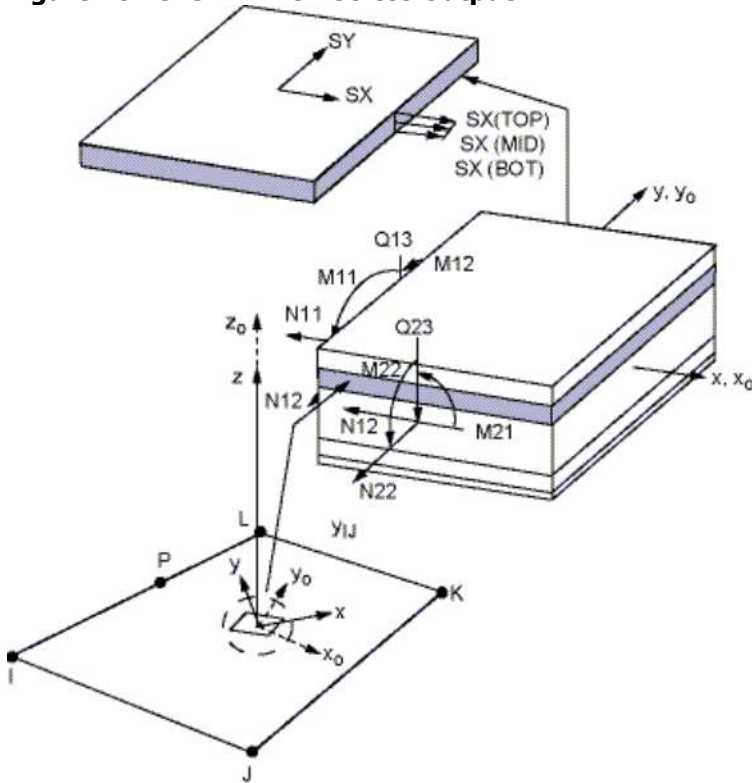
$$\overline{M11} = M11 - L \times N11$$

$$\overline{M22} = M22 - L \times N22$$

$$\overline{M12} = M12 - L \times N12$$

SHELL181 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, ANSYS suggests using the [OUTRES](#) command to ensure that the required results are stored in the database.

**Figure 181.3 SHELL181 Stress Output**



$x_0$  = 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 that 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 "-" indicates that the item is *not* available.

**Table 181.1 SHELL181 Element Output Definitions**

Name	Definition	O	R
EL	Element number and name	Y	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	-	<a href="#">4</a>

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	-	<a href="#">1</a>
S:X, Y, Z, XY, YZ, XZ	Stresses	<a href="#">3</a>	<a href="#">1</a>
S:1, 2, 3	Principal stresses	-	<a href="#">1</a>
S:INT	Stress intensity	-	<a href="#">1</a>
S:EQV	Equivalent stress	-	<a href="#">1</a>
EPEL:X, Y, Z, XY	Elastic strains	<a href="#">3</a>	<a href="#">1</a>
EPEL:EQV	Equivalent elastic strains <a href="#">[Z]</a>	-	<a href="#">1</a>
EPTH:X, Y, Z, XY	Thermal strains	<a href="#">3</a>	<a href="#">1</a>
EPTH:EQV	Equivalent thermal strains <a href="#">[Z]</a>	-	<a href="#">1</a>
EPPL:X, Y, Z, XY	Average plastic strains	<a href="#">3</a>	<a href="#">2</a>
EPPL:EQV	Equivalent plastic strains <a href="#">[Z]</a>	-	<a href="#">2</a>
EPCR:X, Y, Z, XY	Average creep strains	<a href="#">3</a>	<a href="#">2</a>
EPCR:EQV	Equivalent creep strains <a href="#">[Z]</a>	-	<a href="#">2</a>
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	<a href="#">3</a>	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	-	-
NL:EPEQ	Accumulated equivalent plastic strain	-	<a href="#">2</a>
NL:CREQ	Accumulated equivalent creep strain	-	<a href="#">2</a>
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	<a href="#">2</a>
NL:PLWK	Plastic work	-	<a href="#">2</a>
NL:HPRES	Hydrostatic pressure	-	<a href="#">2</a>
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	<a href="#">2</a>
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	<a href="#">8</a>
Q13, Q23	Transverse shear forces (per unit length)	-	<a href="#">8</a>
$\epsilon_{11}, \epsilon_{22}, \epsilon_{12}$	Membrane strains	-	Y
$k_{11}, k_{22}, k_{12}$	Curvatures	-	<a href="#">8</a>
$\gamma_{13}, \gamma_{23}$	Transverse shear strains	-	<a href="#">8</a>
LOCI:X, Y, Z	Integration point locations	-	<a href="#">5</a>
SVAR:1, 2, ... , N	State variables	-	<a href="#">6</a>
ILSXZ	SXZ interlaminar shear stress	-	Y

ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Magnitude of the interlaminar shear stress vector	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22, 12	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13, 23	Averaged transverse shear stresses	-	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 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 181.2: 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 [The Item and Sequence Number Table](#) in this manual for more information. The following notation is used in [Table 181.2: SHELL181 Item and Sequence Numbers](#):

**Name**

output quantity as defined in the [Table 181.1: 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 181.2 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	-	-	-	-

$\epsilon_{11}$	SMISC	9	-	-	-	-
$\epsilon_{22}$	SMISC	10	-	-	-	-
$\epsilon_{12}$	SMISC	11	-	-	-	-
$k_{11}$	SMISC	12	-	-	-	-
$k_{22}$	SMISC	13	-	-	-	-
$k_{12}$	SMISC	14	-	-	-	-
$\gamma_{13}$	SMISC	15	-	-	-	-
$\gamma_{23}$	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
Sm: 11	SMISC	34	-	-	-	-
Sm: 22	SMISC	35	-	-	-	-
Sm: 12	SMISC	36	-	-	-	-
Sb: 11	SMISC	37	-	-	-	-
Sb: 22	SMISC	38	-	-	-	-
Sb: 12	SMISC	39	-	-	-	-
Sp: 11 (at shell bottom)	SMISC	40	-	-	-	-
Sp: 22 (at shell bottom)	SMISC	41	-	-	-	-
Sp: 12 (at shell bottom)	SMISC	42	-	-	-	-
Sp: 11 (at shell top)	SMISC	43	-	-	-	-
Sp: 22 (at shell top)	SMISC	44	-	-	-	-
Sp: 12 (at shell top)	SMISC	45	-	-	-	-
St: 13	SMISC	46	-	-	-	-
St: 23	SMISC	47	-	-	-	-

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

- ANSYS recommends against using this element in triangular form, except as a filler element. Avoid triangular form especially in areas with high stress gradients.
- Zero-area elements are not allowed. (Zero-area elements occur most often whenever the elements are numbered improperly.)
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed (but zero thickness layers are allowed).
- If multiple load steps are used, the number of layers may not change between load steps.
- When the element is associated with [preintegrated shell sections](#) (**SECTYPE**,,GENS), additional restrictions apply. For more information, see [Considerations for Using Preintegrated Shell Sections](#).
- If reduced integration is used (KEYOPT(3) = 0) SHELL181 ignores rotary inertia effects when an unbalanced laminate construction is used, and all inertial effects are assumed to be in the nodal plane (that is, an unbalanced laminate construction and offsets have no effect on the mass properties of the element).
- For most composite analyses, ANSYS recommends setting KEYOPT(3) = 2 (necessary to capture the stress gradients).
- 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.
- 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.
- 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.
- The layer orientation angle has no effect if the material of the layer is hyperelastic.
- Before using this element in a simulation containing curved thick shell structures with unbalanced laminate construction or shell offsets, validate the usage via full 3-D modeling with a solid element in a simpler representative model. This element may underestimate the curved thick shell stiffness, particularly when the offset is large and the structure is under torsional load.
- The through-thickness stress, SZ, is always zero.
- This element works best with the full Newton-Raphson solution scheme (**NROPT**,FULL,ON).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- In a nonlinear analysis, the solution process terminates if the thickness at any integration point that was define with a nonzero thickness vanishes (within a small numerical tolerance).
- 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 has no bending stiffness, a condition that can result in solver and convergence problems.

## SHELL181 Product Restrictions

### ANSYS Professional.

- The only special features allowed are stress stiffening, large deflections, and plasticity (BISO, BKIN).