

[Elements Reference](#) | [Part I, Element Library](#) |

SHELL99 Linear Layered Structural Shell

MP ME ST PR PRN <> <> <> <> <> <> PP <>

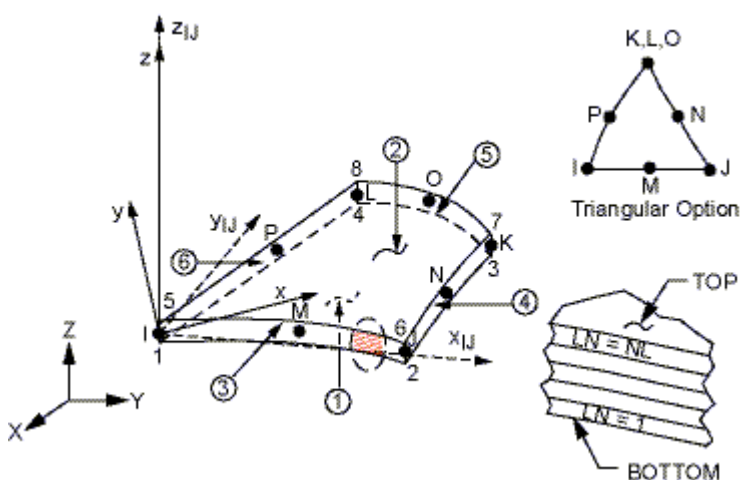
[Product Restrictions](#)

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 99.1 SHELL99 Geometry



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

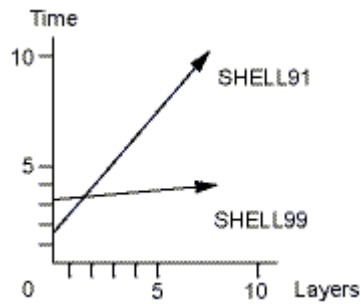
LN = Layer Number

NL = Total Number of Layers

SHELL99 Input Data

The geometry, node locations, and the coordinate system for this element are shown in [Figure 99.1: "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:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - \begin{Bmatrix} MT \\ BT \end{Bmatrix}$$

where these terms are defined in the [Theory Reference for ANSYS and ANSYS Workbench](#). The submatrix [A] is input by real constants as:

$$[A]_{6 \times 6} = \begin{bmatrix} A_1 & A_2 & A_3 & A_4 & A_5 & A_6 \\ A_2 & A_7 & A_8 & A_9 & A_{10} & A_{11} \\ A_3 & A_8 & A_{12} & A_{13} & A_{14} & A_{15} \\ A_4 & A_9 & A_{13} & A_{16} & A_{17} & A_{18} \\ A_5 & A_{10} & A_{14} & A_{17} & A_{19} & A_{20} \\ A_6 & A_{11} & A_{15} & A_{18} & A_{20} & A_{21} \end{bmatrix} \quad \text{or} \quad [A]_{3 \times 3} = \begin{bmatrix} A_1 & A_2 & A_3 \\ A_2 & A_4 & A_5 \\ A_3 & A_5 & A_6 \end{bmatrix}$$

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:

$$\begin{Bmatrix} N \\ M \\ L \end{Bmatrix} = \begin{bmatrix} A & B & E \\ B & D & E \\ E & E & F \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \\ \omega \end{Bmatrix} - \begin{Bmatrix} MT \\ BT \\ QT \end{Bmatrix} \quad (99-1)$$

If KEYOPT(2) = 4, the transverse shear terms are, for example, $A_6 * \text{TRSHEAR}$ 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 [Coordinate Systems](#). The local coordinate system for each layer is defined as shown in [Figure 99.2: "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 [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(1) 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 91.4: "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 91.5: "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 [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 99.1: "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 99.1: "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 [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 99.1: "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 99.1 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
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 [Failure Criteria](#).

SHELL99 Output Data

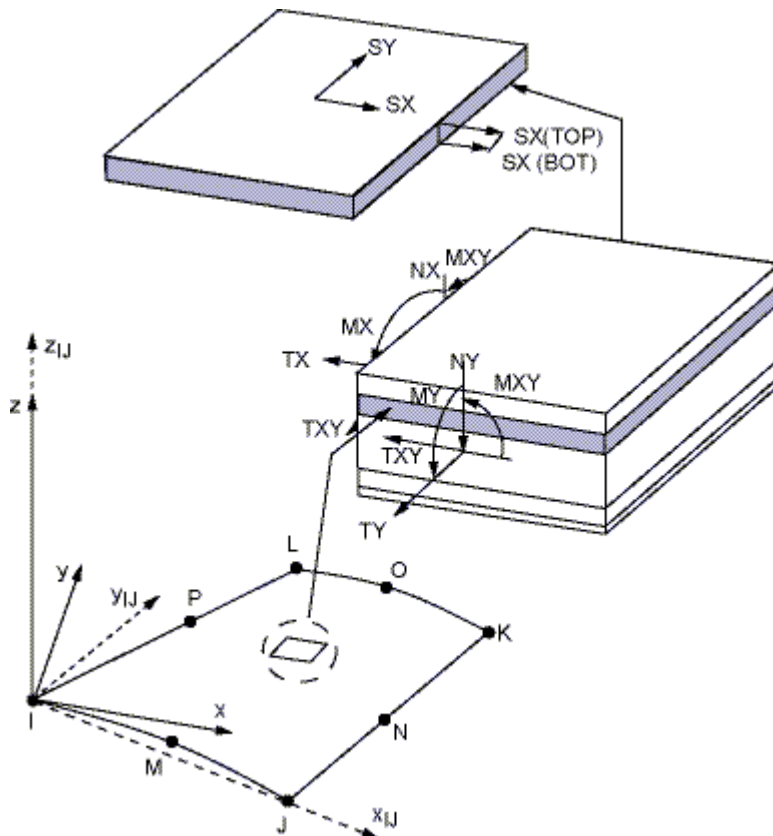
The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in [Table 99.2: "SHELL99 Element Output Definitions"](#).

Several items are illustrated in [Figure 99.2: "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 99.2: "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 [Solution Output](#). See the [Basic Analysis Guide](#) for ways to view results.

Figure 99.2 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 99.2 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
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
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	10	-

system			
--------	--	--	--

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) ≠ 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) ≠ 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 99.3: "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 [The Item and Sequence Number Table](#) in this manual for more information. The following notation is used in [Table 99.3: "SHELL99 Item and Sequence Numbers"](#):

Name

output quantity as defined in the [Table 99.2: "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 99.3 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	<u>ETABLE</u> and <u>ESOL</u> 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			
EPELFXZ	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 99.3: "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 ($\text{KEYOPT}(11) \neq 0$), 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 ($\text{KEYOPT}(2) = 2$ 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 91.5: "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.

Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
