Collaborative
Solutions Inc
An ANSYS Support Distributor

Date	January 11, 2000	Memo Number	STI25:000111
Subject	ANSYS Tips & Tricks: Structural SHELL Elements, Part 1		
Keywords	Structural: SHELL		

1. Introduction:

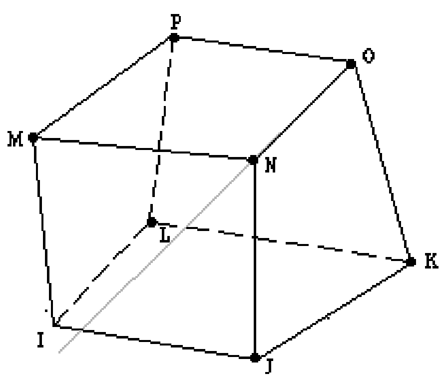
Because of the wide range of structural shell elements present in the ANSYS element library, it may be difficult for the user to keep track of the applicability of the various shell elements for specific cases. This memo is Part 1 in a series which hopes to address the differences between the shell elements and to provide the user with enough information to utilize the appropriate elements correctly.

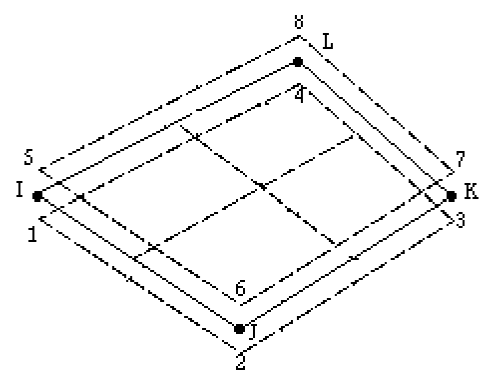
This first memo hopes to provide some simplified theoretical background[1] on general, lower-order 3D shell elements in ANSYS in linear and nonlinear applications. Axisymmetric shell elements[2], special-purpose shells[3], and composite shells[4] will not be addressed presently. Subsequent memos will address specific features as well as practical applications of shells.

2. Background Overview:

Shell elements are used to model structures where one dimension (the thickness) is much smaller than the other dimensions, where the dimensionality is based on that of the physical structure[5], not of the element size. As a result of this assumption, the stresses through the thickness of the shell are assumed to be negligible. Generally, shell elements follow shell and plate theory as covered by basic mechanics texts.

As a very simplified explanation, one can view a shell element as a "collapsed" solid element. Picture an eight-node solid hexahedral element as shown below. Collapse the sides of the solid element denoted by





edges IM, JN, KO, and LP. Each edge is now represented with one node (node I, J, K, or L, respectively) with the shell element – for example, edge JN of the solid is similar to edge 26 of the shell which is modeled with one node, node J. Because of this simplification of the representation of the edges, we have certain assumptions that need to be considered for shells.

3. Degrees of Freedom:

A solid element has three translational degrees of freedom, which are easy to understand physically (i.e., movement in x-, y-, or z-coordinates). A shell element, however, needs additional rotational degrees of freedom because of the fact that an edge is modeled with one node (while a shell is spatially 3D, it is still a geometrically 2D structure).

If one considers the edge IJ of the shell in the figure above, this edge is found to represent face 1256 of a 3D structure. When the bottom 12 of face 1256 moves relative to top 56, this needs to be represented as an out-of-plane rotation for edge IJ of the shell. In general, out-of-plane rotations for the nodes of the shell

[1] Additional considerations such as curvature and SHELL63 bending will be covered later
[2] SHELL51 (axisymmetric w/ torsion) and SHELL61 (axisymmetric-harmonic)
[3] SHELL28 (shear/twist panel), SHELL41 (membrane), and SHELL150 (p-element)
[4] SHELL91 (nonlinear composite) and SHELL99 (linear composite)
[5] These measures can be based on the distance between supports/constraints or the wavelength of the modeshapes of interest

2303 W. 190th St • Redondo Beach, CA 90278
310.896.1230 • FAX 310.896.1240 • http://www.csi-ansys.com

element can be defined by two orthogonal components. For example, in the case of node J, an out-of-plane rotation can be defined by a rotation about edge IJ and another rotation about an edge orthogonal to IJ but in the same plane (such as a rotation about edge JK if the shell was rectangular). This provides some insight into the two additional degrees of freedom which describe the out-of-plane rotations of the shell, labeled ROTX and ROTY.

The associated nodal load for rotational DOF is a moment – moments are actually coupled forces that are needed along the face 1256 (one force on top 56 and opposite force on bottom 12). Because face 1256 is not explicitly modeled but is represented by edge IJ, a moment allows for the consideration of this coupled force to produce a given out-of-plane rotation.

There exist shell element formulations where each node has five degrees of freedom (UX, UY, UZ, ROTX, ROTY) as outlined in this section. Because of various reasons, including performance considerations, ANSYS utilizes shell elements with six degrees of freedom. The additional degree of freedom is an in-plane rotational degree of freedom ROTZ, also known as a "drilling DOF", for each node. Consider rotation of node J in a clockwise (or counter-clockwise) manner as defined by nodes I-J-K-L – this is what is meant by the drilling, or in-plane, rotational degree of freedom.

4. Drilling DOF:

For general shell elements with six degrees of freedom, the in-plane rotational degree of freedom actually has a 'fictitious' stiffness associated with it.[6] This in-plane rotational stiffness is needed for numerical reasons to prevent the stiffness matrix from becoming singular[7], not necessarily for physical representation since the in-plane behavior is defined by the shell's translational degrees of freedom.

There are different ways in which this drilling stiffness is calculated:

- A small value (such as 1.0e-5·EX) is used for the in-plane stiffness. This is similar to adding a "torsional spring" to control the in-plane rotational DOF (e.g., SHELL43/63/143 with KEYOPT(3)≠2)
- Including Allman-type rotational stiffness (e.g., SHELL43/63/143 with KEYOPT(3)=2)
- Use of a penalty method to relate in-plane rotational DOF with in-plane translational DOF (e.g., SHELL181)

The details of the methods to control the drilling freedom are not important enough to be covered in detail, although it suffices to say that because of this fictitious stiffness, there exist different ways to control this degree of freedom as listed above.

4.1 Considerations of the Drilling DOF:

There are two points regarding the in-plane rotational DOF which the user needs to be aware of: (1) because of numerical reasons, the first method of controlling the drilling freedom with a spring-type stiffness may not pass rigid-body mode tests for an assemblage of flat shells and (2) although both shells and beams have six degrees of freedom (UX/Y/Z and ROTX/Y/Z), beams should not be directly connected in the normal direction of shells.

For flat assemblage of shell elements which use a spring-type of stiffness to control the rigid-body modes, the in-plane stiffness may affect the sixth rigid-body mode. This manifests itself as a non-negligible value. For example, in the case of a simple rectangular plate meshed with SHELL63 elements, using the Block Lanczos method, the sixth mode is found to be ~1.0 Hz instead of 0.0 Hz. Changing KEYOPT(3)=2 to include Allman-type rotational stiffness, the sixth mode is then calculated to be 0.65E-03 which provides sufficient accuracy. Note that shell181 does not exhibit this behavior and calculates the sixth mode to be 0.18E-03.

This happens only with flat structures such as a simple plate or a box meshed with SHELL43/63/143 with KEYOPT(3)≠2. A curved surface (for example, one quadrant of a sphere) meshed with the same elements and keyoptions pass rigid-body tests. The reason for this is due to the fact that flat surfaces (an assemblage of coplanar shell elements) are very sensitive to in-plane rotation whereas curved surfaces are not (the in-plane rotation is taken care of by the out-of-plane DOF). Hence, any small stiffness value as used in option (1) to handle the drilling DOF will affect the sixth rigid-body mode whereas including

[6] For purposes of brevity and simplicity, the in-plane rotational stiffness can be assumed to be a "fictitious" stiffness. The in-plane rotation is not explicitly defined in the element shape function, so this accounts for the need for a "fictitious" stiffness.
[7] For an assemblage of coplanar flat shell elements, [K] becomes singular

Allman-type rotational DOF or using a penalty method to related in-plane translational DOF with rotational DOF circumvent this since no "articifical" stiffness is introduced.

For dynamics problems, a non-zero rigid-body mode is significant, so, *for flat shells only*, either (a) the keyoptions must be changed for SHELL43/63/143 to use an Allman-type rotational stiffness or (b) SHELL181 should be used instead. Higher-order elements such as SHELL93 also do not have this problem.

The second point mentioned above discusses connecting beams normal to shells. This is not recommended for two reasons. One reason is that the shell's in-plane rotational DOF does not have any significant torsional stiffness associated with it as explained above (it is included for numerical, not physical, reasons). Attaching a beam normal to the shell means that the torsional moment of the beam will be transferred to a very small torsional stiffness ("fictitious" stiffness) of the shell. Numerical difficulties will arise (small pivots), besides the fact that the results may not be correct.

A related reason is due to the fact that the degrees of freedom for the beams and shells are not exactly the same. A beam is geometrically 1D but spatially 3D. As a result, to physically model the simplified behavior of the cross-section, rotational degrees of freedom are introduced. The associated moments and forces are "concentrated" and represent that of the *entire* beam cross-section (forces and moments that act on the entire cross-section of the beam). On the other hand, the in-plane behavior of a shell element is fully defined by the 4 nodes – the simplification of geometry only occurs for the thickness, not the surface of the shell. Hence, the in-plane forces of the shell are not "concentrated" but are *local* forces – consequently, the portion of the shell these forces represent may not be the same as that of the beam.

As another way of looking at this, one can imagine that a node of a beam element represents the entire beam cross-section. A node on a shell represents ¼ of a rectangular shell, a localized area – the physical geometry of the two may not be the same, yet if a beam is directly connected normal to a shell surface, the assumption made is that the nodes characterize equivalent "sections".

A similar problem exists when connecting shells and solid elements (including solids with or without rotational DOF). Correct connection of beams normal to shell elements (or shells and solids) require care – for example, one method involves the use of constraint equations to "tie" the degrees of freedom together to ensure full moment continuity. A later memo will address different techniques for performing these connections in more detail.

5. Midsurface Representation:

The shell nodes define the midsurface of the structure as shown in the second figure of the shell element above. All shell elements are modeled with this assumption, with the exception of the composite SHELL91 and SHELL99 elements which can have its nodes represent the top, bottom, or midsurface of the structure (KEYOPT(11)).

The midsurface representation should be kept in mind when modeling shells with imported CAD geometry. Sometimes, a midsurface cannot be extracted from the CAD geometry, so the user either needs to determine if modeling the shells as the top or the bottom of the imported geometry is appropriate (i.e., meshing the top surfaces with the shell elements should imply that the shell offset is negligible for the results).

6. Thin and Thick Shells – Transverse Shear:

The main difference between "thin" and "thick" (or "moderately thick") shells is how one considers transverse shear.

One can picture a straight line that is normal to the undeformed midsurface such as edge 26 represented by node J in the second figure above. Any straight line, such as edge 26, normal to the undeformed midsurface is assumed to remain straight during deformation. This is because of the fact that this edge is modeled in a simplified manner by one node. However, while this "straight line" is assumed to remain straight, the treatment of the line relative to the deformed midsurface constitutes differences in Kirchhoff and Reissner/Mindlin theory.

In classical shell theory (Kirchhoff), any straight line (such as edge 26 above) that is normal to the midsurface before loading is assumed to remain straight and normal to the deformed midsurface. This essentially means that transverse shear deformation is neglected. This assumption is valid only for thin shells such as SHELL41 (membrane) and SHELL63. As a *very* general rule-of-thumb, if the thickness is less than 1/15-1/30 of the characteristic length of the surface of the shell[8], thin shell theory is suitable.

On the other hand, shear-flexible theory (Mindlin) assumes that any straight line that is normal to the midsurface prior to deformation is assumed to remain straight but does not necessarily remain normal to the midsurface after loading. As a result, this theory allows transverse shear deformation, where transverse shear strains are constant through the thickness of the shell. This is a first-order approximation which is appropriate for "thick" (or "moderately thick") shells. SHELL43, 143, 181, 91, 93, and 99 utilize shear-flexible theory. It may be instructive to note that the transverse shear strains for the lower-order shell elements (43, 143, 181) are modified to avoid shear locking, as consistent with the Bathe-Dvorkin formulation (also known as the MITC4 shell elements).[9] As a *very* general guideline, if the thickness of the shell is between 1/10 and 1/20 of the characteristic length of the surface of the shell, thick shells are suitable.

7. Stiffness Matrix and Solvers – Membrane vs. Bending Behavior:

For shell elements, the sparse or frontal solvers are often recommended over the iterative solvers such as PCG solvers. To better understand why this is so, a brief overview of the element stiffness matrices for shells is helpful.

A shell element represents a structure which is thin in one direction compared with its other dimensions. As a result, the stiffness in bending is significantly smaller than its in-plane (membrane) stiffness. This is analogous to a beam which can carry significant loads axially (in tension) but much smaller loads in bending.

From a simplified, numerical perspective, a flat shell element can be thought of as a superposition of a plane stress element and a plate bending element. A plane stress element is simple enough to understand since one often deals with these elements in ANSYS (e.g., PLANE42). They have in-plane translational degrees of freedom and the through-thickness stress is assumed to be zero. A plate bending element (not explicitly available in ANSYS but a SHELL63 element can mimic this) generally has three degrees of freedom – two out-of-plane rotations and translation normal to the shell. A plate bending element, as its name suggests, can only model bending.[10] A flat shell element is therefore a combination of both a plane stress and plate bending element, giving it two rotational DOF and three translational DOF – as mentioned in Section 4, the in-plane rotation needs additional considerations, but this contributes to the sixth DOF for shells.

In reality, shell curvature, out-of-plane loads, and the type of boundary conditions lead to both bending and membrane forces (moments). Because the membrane stiffness is much larger than the bending stiffness, the matrix may become poorly-conditioned. Iterative solvers such as the PCG solver are not as effective for ill-conditioned matrices, so the sparse and frontal solvers are recommended as they are not sensitive to poorly-conditioned matrices.

Note that for in-plane behavior, the lower-order shells support "incompatible modes" (a.k.a. "extra displacement shapes"), which is generally the default behavior. SHELL181, however, uses uniform reduced integration as the default behavior but can also support full integration with incompatible modes. Full integration without incompatible modes should never be used as it poorly predicts in-plane behavior (locking becomes an issue).

8. Small Strain vs. Finite Strain:

All of the shell elements support large rotation (large deflection). However, SHELL63 does not support material nonlinearities or finite strain. SHELL43 and SHELL181 are used for finite strain behavior (as well as the higher-order SHELL93). The user should keep this in mind for nonlinear applications.

SHELL181 does account for thickness change in nonlinear applications (whereas SHELL93 does not). Note that this thickness change is due to the "stretching" of the shell. To preserve volume in nonlinear applications (in plasticity or hyperelasticity, where $\nu \approx 0.5$), any change in length of the shell will result in a change in thickness of the shell. This thickness change is *not* accounted for in a contact analysis where the

[8] The characteristic length can be thought of as distance between supports on the surface of the shell structure.
[9] Please refer to Bathe, Ch. 5.4.2, for a summary
[10] Please refer to Bathe, Ch. 5.4.2 or Cook, Ch. 11 for details on plate elements

shell is being compressed between its top and bottom surfaces. To model this type of thickness change, SOLID elements must be used. To reiterate, the change in thickness of shell elements such as SHELL181 is due to the "stretching" of the shell only.

Another benefit of SHELL181 in general nonlinear problems is its use of reduced integration. This provides for faster solution times, although one needs a finer mesh and may have to verify that hourglassing is not a problem.

To summarize: for general large rotation nonlinear analyses, any shell will suffice. For other nonlinear applications where finite strain becomes important, the author recommends SHELL181. This is especially true in ANSYS 5.6 since SHELL181 supports most material nonlinearities.

9. Plasticity and Hyperelasticity:

In ANSYS 5.5, SHELL181 supports isotropic hardening while other shells such as SHELL43/143/93 support kinematic hardening. The choice of shells was dictated by the hardening law in 5.5. However, in ANSYS 5.6, SHELL181 supports not only isotropic and kinematic hardening but the new Chaboche model and Voce hardening laws as well.

SHELL181 is also the only shell element to support Mooney-Rivlin hyperelastic models. In future revisions (e.g., 5.7), the new Arruda-Boyce hyperelastic model is planned to be incorporated into SHELL181.

10. Conclusion:

This memo provided a simplified, brief introduction to shell elements (both in general and as implemented in ANSYS), focusing on the lower-order shell elements (SHELL43/63/143/181). There are various issues related to the drilling freedom, transverse shear effects, solver considerations, and nonlinear capabilities which were outlined. Later memos will hope to address these and other points in more detail to provide a more practical view of the use of shell elements in ANSYS, thereby helping the user to implement these elements in the most effective manner.

Because there are so many options and considerations, the author recommends to users to start off simple – model their structure with SHELL63 elements, for example. Once the behavior of the system is better understood, transverse shear deformation or nonlinearities can included, as deemed necessary.

The ANSYS 18x family of elements are being developed as the "next generation" of structural elements which will provide the user with all the capabilities in a smaller element library. ANSYS 5.6 has provided much of this in the form of enhancements to SHELL181. As a result, the user may want to consider using SHELL181 instead of SHELL43 or SHELL143 for better performance and more features in nonlinear applications.

11. References:

1) Kohnke, P. ed., "ANSYS Theory Reference Release 5.6", 11th ed., ANSYS, Inc. 1999
2) Bathe, K.J., "Finite Element Procedures", Prentice-Hall, Inc. 1996
3) Cook, R.D., Malkus, D.S., Plesha, M.E., "Concepts and Applications of Finite Element Analysis", 3rd ed., John Wiley and Sons, Inc. 1989
4) Private conversation, 1/6/00, G. Bhashyam, ANSYS, Inc.

Sheldon Imaoka
Collaborative Solutions, Inc. (LA Office)
Engineering Consultant

ANSYS Tips and Tricks

"ANSYS Tips & Tricks" is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe can send an email to operator@csi-ansys.com with their name and company information with the subject heading "ANSYS Tips and Tricks subscription". Older issues may be posted on CSI's website at http://www.csi-ansys.com.

Corrections, comments, and suggestions are welcome and can be sent to operator@csi-ansys.com [they will be distributed to the appropriate person(s)]. While CSI engineers base their "Tips & Tricks" on technical support calls and user questions, ideas on future topics are appreciated. Users who wish to submit their own "ANSYS Tips and Tricks" are encouraged to do so by emailing the above address for more information.

XANSYS Mailing List

The xansys mailing list is a forum for questions and discussions of the use of ANSYS. As of 12/99, there are more than 900 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the "digest" option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

```
This list is for *ALL* users of the ANSYS finite element analysis program from
around the world. The list allows rapid communication among users concerning
program bugs/ideas/modeling techniques. This list is NOT affiliated with ANSYS,
Inc. even though several members of the ANSYS, Inc. staff are subscribers and
regular contributors.

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com
To unsubscribe send blank email to xansys-unsubscribe@onelist.com
Archived on http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html
ANOTHER archive on http://www.eScribe.com/software/xansys/
(A poor archive is also at http://www.onelist.com/archives.cgi/xansys)
```

CSI ANSYS Technical Support

Collaborative Solutions, Inc. is committed to providing the best customer support in our industry. Three people will be devoted to technical support from 8:00am to 5:00pm PST every working day. CSI customers with active TECS (maintenance) agreements may contact CSI by any of the following ways:

```
Phone:  760-431-4815 (ask for ANSYS technical support)
Fax:  760-431-4824
Web:  http://www.csi-ansys.com
E-mail:  firstname.lastname@csi-ansys.com
Anonymous ftp site: ftp://ftp.csi-ansys.com

CSI Engineers:
Karen Dhuyvetter
Greg Miller
Sean Harvey
Alfred Saad
Bill Bulat
Sheldon Imaoka
David Haberman
```

All comments and suggestions are welcome.