

Shell FAQ



This page is devoted to **frequently asked questions** (FAQ) related to [shells](#).

On this page:

- [Formulation](#)
 - [What is the difference between thick and thin shells?](#)
 - [How are shell stresses calculated, and can these stresses be compared to the allowable stress of shell material?](#)
 - [How does the software apply stiffness modifiers to shell objects?](#)
- [Modeling](#)
 - [How is a simply supported slab modeled using shell objects?](#)
 - [What are the practical limits on the maximum thickness of shell objects relative to shell dimension?](#)
 - [How has my area object been meshed before any meshing is specified?](#)
 - [How are masonry shear walls modeled?](#)
 - [Why can I not draw an area object between four points?](#)
 - [How are shells merged with the sides of frame objects?](#)
- [Loading and output](#)
 - [What is the difference between the load assignments Uniform \(Shell\) and Uniform to Frame \(Shell\)?](#)
 - [How can I post-process the bridge-diaphragm area-object output to obtain meaningful design forces?](#)
- [Layered shell](#)
 - [For unsymmetrical layering, how are membrane and plate behaviors coupled?](#)
 - [How does material orientation relate to object orientation?](#)

Formulation

What is the difference between thick and thin shells?

Answer: For response, please see the [Thin vs. Thick shells](#) article.

How are shell stresses calculated, and can these stresses be compared to the allowable stress of shell material?

Answer: Yes, the [shell](#) stresses obtained using [SAP2000](#) may be compared directly to the allowable stress of the shell material, as this is the essence of Allowable Stress Design (ASD). Shell stresses may be reviewed in tabular format through Display > Show Tables > Analysis Results > Element Output > Area Output > Table: Element Stresses - Area Shells.

Stresses are calculated as the product of the stress-strain constitutive matrix and the strain vector obtained from [joint](#) displacements. The techniques and formulations involved are adopted from the following publications:

- Ibrahimbegovic, A., Wilson, E. (1991). [A Unified Formulation for Triangular and Quadrilateral Flat Shell Finite Elements with Six Nodal Degrees of Freedom](#). *Communications in Applied Numerical Methods*, 7(1), 1-9.
- Taylor, R., Simo, J. (1985). Bending and Membrane Elements for Analysis of Thick and Thin Shells. *Proceedings of the NUMETA 1985 Conference*, Swansea, Wales.

A comprehensive description can also be found in Dr. Wilson's textbook:

- Wilson, E. L. (2004). *Static and Dynamic Analysis of Structures* (4th ed.). Berkeley, CA: Computers and Structures, Inc.

How does the software apply stiffness modifiers to shell objects?

Answer: When [stiffness modifiers](#) are applied, the corresponding terms of the shell stiffness matrix are modified. This is comparable to modifying, for a given direction, the Young's Modulus or the shear moduli of the material. The structural stiffness matrix is then assembled, and global equilibrium equations are solved to generate deflections. These displacements and rotations then relate to strain fields which produce internal member forces and stresses.

Modeling

How is a simply supported slab modeled using shell objects?

Answer: For response, please see the [Modeling simply supported shells](#) tutorial.

What are the practical limits on the maximum thickness of shell objects relative to shell dimension?

Answer: During [shell](#) formulation, the ratio of plan dimension to thickness concerns the deformation span between inflection points, and not the actual plan dimension of the shell. Shell thickness may even be greater than the actual plan dimensions so long as the projection of curvature under plate-bending behavior meets the deformation-span to thickness ratios which follow:

Thick-plate effects become significant when the deformation-span to thickness ratio is between approximately 20:1 and 10:1. The formulation itself is adequate for ratio down to 5:1 or 4:1.

Verification example 2-012 is attached. This example models a 0.1in x 0.1in shell object with 0.5in thickness. The solution obtained using [SAP2000](#) matches that from the independent solution, validating the formulation and demonstrating the accuracy of shell behavior.

Please note that shell formulation is an approximate and special case of three-dimensional elasticity. Shell objects may be appropriate for some applications, while [solid](#) objects may be more suitable for others, such as with the assessment of localized behavior. Simple test problems may always be run to check the differences between [modeling](#) approaches.

How has my area object been meshed before any meshing is specified?

Extended Question: I have an area object with more than four vertices for which meshing is not specified. However, this object is meshed in the analysis model. How was this mesh created?

Answer: [CSI](#) Software uses a general [meshing](#) tool to mesh objects with more than four [nodes](#) when no auto-meshing has been assigned.

How are masonry shear walls modeled?

Answer: Masonry shear walls are modeled by defining the appropriate material properties or stiffness modifiers. A good resource for masonry properties is the [NCMA TEK 14-1A](#) publication.

Why can I not draw an area object between four points?

Answer: If the four points do not lie in a plane, an area object cannot be drawn because the profile is warped. Instead of using four-sided [shells](#), these areas may be drawn using triangles.

How are shells merged with the sides of frame objects?

Answer: Two modeling options are described as follows:

- Draw [special joints](#) along the boundary of the [frames](#), [constrain](#) them to the [joints](#) of the frame objects, then connect the [shells](#) to these special joints.
- Use [insertion points](#) to offset the frames from the shells.

Loading and output

What is the difference between the load assignments Uniform (Shell) and Uniform to Frame (Shell)?

Answer: The difference between these two options is as follows:

- **Uniform (Shell)** – uniform loading is applied directly to a [shell](#) object, then loading transfers to the structure through shell [joints](#) which coincide with structural members.
- **Uniform to Frame (Shell)** – uniform loading is applied directly to the [frame](#) objects specified along the edges of a shell. Load distribution may be one way or two way. Distribution of loading may be reviewed through Display > Show Load Assigns > Area.

How can I post-process the bridge-diaphragm area-object output to obtain meaningful design forces?

Answer: Diaphragm design forces may be obtained from the [joint](#) forces within the [shell](#) objects used to model the bridge diaphragms. Through post-processing, joint forces may be transformed into bridge response at the location of interest. Since this basically summarizes the process for generating section-cut forces, it may be more productive to obtain design forces through [section cuts](#).

Layered shell

For unsymmetrical layering, how are membrane and plate behaviors coupled?

Extended Question: The CSI Analysis Reference Manual states that "unless the layering is fully symmetrical in the thickness direction, membrane and plate behavior will be coupled." If the element is defined such that planes remain plane in the thickness direction, then even for a symmetric definition of membrane layers, would not the layers contribute to the out-of-plane bending stiffness and demonstrate stresses with plate behavior?

Answer: This is correct that the membrane layers contribute to bending stiffness. The statement here concerns whether or not transverse loading generates membrane forces and deformation, and likewise, whether or not in-plane loading generates bending behavior. As stated, in-plane deflection will not produce plate bending when a [layered shell](#) is symmetric through its thickness.

This coupling effect could be demonstrated by applying a uniform temperature load to the [shell](#) object. For symmetrical layering, the temperature load will cause only in-plane deformations. For nonsymmetrical layering, out-of-plane deformations will be generated. As illustrated in the following Wikipedia article, this is similar to [bimetallic strip](#) behavior.

How does material orientation relate to object orientation?

Extended Question: How is material orientation and S11, S22, and S12 orientation defined for shell objects? Are the 1, 2, and 3 directions the same as the local axes of the shell? A material angle of 90 degrees is used for longitudinal bar elements in the Nonlinear Shear Wall [Watch & Learn](#) video, then nonlinearity is defined in the S11 direction, which is the horizontal direction for the element. Is this correct that the software rotates the local axis, for that layer, by 90 degrees relative to the element local axis? Would it have the same effect to have the material at zero degrees, but define the nonlinearity in the S22 direction? On the other hand, if the angle simply refers to material orientation, should not the nonlinearity be defined in the S22 direction, which is the nonlinear direction?

Answer: As shown in Figure 1, the material orientation of a given layer is defined relative to the local coordinate system. Material angle is measured counterclockwise from local member orientation. Using a 90° material angle with nonlinearity along S11 has the same effect as using a 0° material angle with nonlinearity along S22.

For such uniaxial materials as rebar, where behavior is only defined in the material 1 direction, a 90° material angle with nonlinearity along S11 must be specified. For fiber-wrapped composites, it may be best to specify material behavior along another angle, perhaps at 45° from the member local axes.

Please note that while material properties are specified in material coordinates, loading and output forces and stresses will always be input and reported in terms of member local axes.

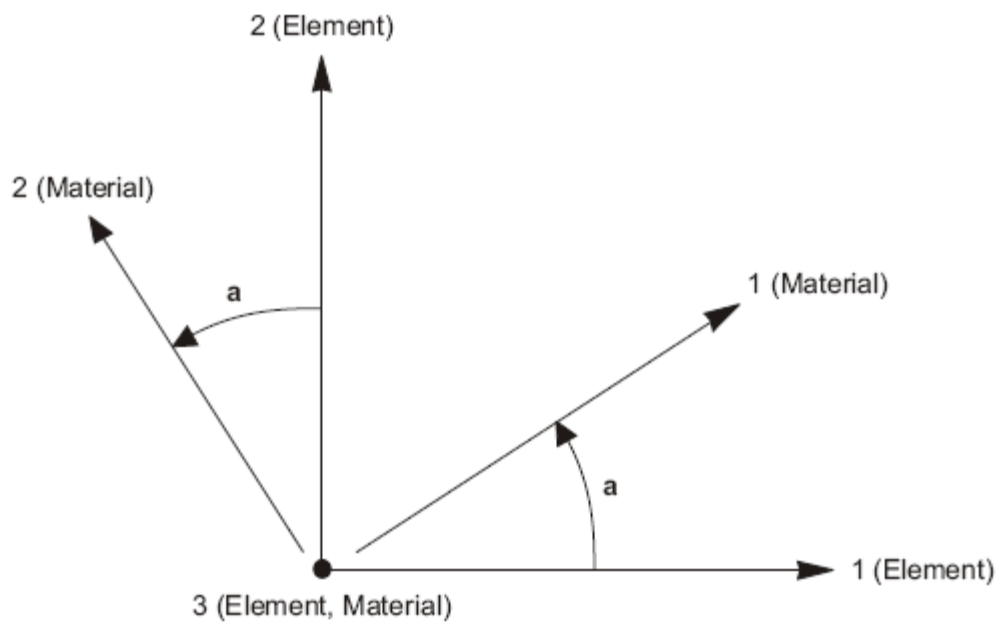


Figure 36
Shell Section Material Angle

Figure 1 - Shell-section material angle

