ABAQUS

Appendix 2

Shell Elements in ABAQUS/Explicit

Overview

- Conventional Shell Elements
- Continuum Shell Elements

Conventional Shell Elements

Conventional Shell Elements

– Triangular and quadrilateral conventional shell elements are available with linear interpolation and your choice of large-strain and small-strain formulations.

– A linear axisymmetric shell element is also available.

– For **most analyses** the standard large-strain shell elements are appropriate. These include:

- S4R
- S3R
- SAX1
- These elements are discussed briefly here and in more detail in the *Element Selection in ABAQUS/Standard* lecture notes.

– If, however, the analysis involves small membrane strains and arbitrarily large rotations, the small-strain shell elements (S4RS, S3RS, and S4RSW) are more computationally efficient.

Conventional Shell Elements

- **Reference surface offsets**
  - The reference surface of the shell is defined by the shell element's nodes and normal direction.
  - The reference surface is typically coincident with the shell's midsurface.
  - However, many situations arise in which it is more convenient to define the reference surface as offset from the shell's midsurface.

Conventional Shell Elements

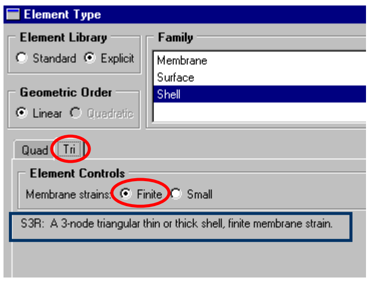
- For example, surfaces created in CAD packages usually represent either the top or the bottom surface of the shell body.
  - In this case it may be easier to define the reference surface to be coincident with the CAD surface and, therefore, offset from the shell's midsurface.
- Shell offsets can also be useful when modeling a shell with continuously varying thickness. In this case defining the nodes at the shell midsurface can be difficult.
- If one surface is smooth while the other is rough, as in some aircraft structures, it is easiest to use shell offsets to define the nodes at the smooth surface.

Conventional Shell Elements

- By default, shell offset and thickness are accounted for in contact constraints in ABAQUS/Explicit.
  - The effect of offset and thickness in contact can be suppressed.
- For stability purposes ABAQUS/Explicit automatically augments the rotary inertia used for shell elements. The additional inertia is on the order of the square of the offset, which may result in errors in the dynamics for large offsets.
- When large offsets (e.g., more than 10% of shell thickness) from the shell's midsurface are necessary, it may be better to use multi-point constraints or rigid body constraints instead.

Conventional Shell Elements

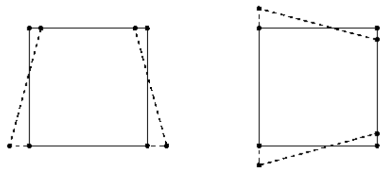
- **Large-strain shell elements in ABAQUS/Explicit**
  - The large-strain shell elements use a Mindlin-Reissner type of flexural theory that includes transverse shear.
  - **S3R**
    - There are no propagating hourglass modes.
    - Transverse shear constraints (2 per element) can cause mild shear locking.
    - Because of the facet approximation, it is not very accurate for curved shells.



Conventional Shell Elements

– **S4R**

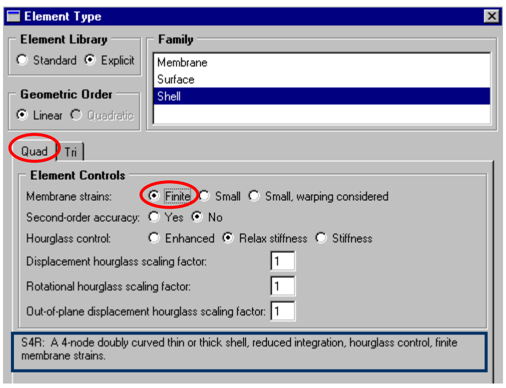
- Uniformly reduced integration to avoid shear and membrane locking.
- The element has several hourglass modes that may propagate over the mesh.



- Converges to shear flexible theory for thick shells and classical theory for thin shells.
- **S4R is a robust, general-purpose element that is suitable for a wide range of applications.**

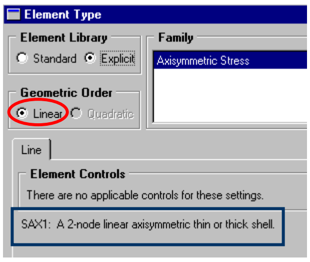
Conventional Shell Elements

– **S4R** (cont'd)



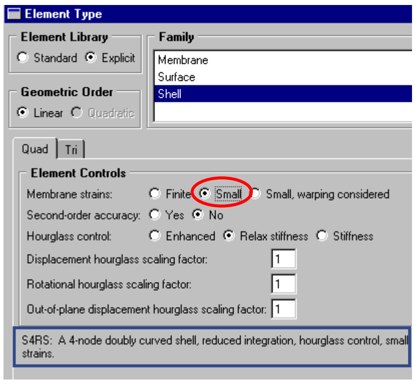
Conventional Shell Elements

- **Axisymmetric shell elements with axisymmetric response**
  - These elements can be used when the geometry and the loading of the structure is axisymmetric.
  - One element type available:
    - SAX1 (linear interpolation)
  - This element possess three degrees of freedom per node (u_r, u_z, ϕ_θ) and account for finite membrane strains.



Conventional Shell Elements

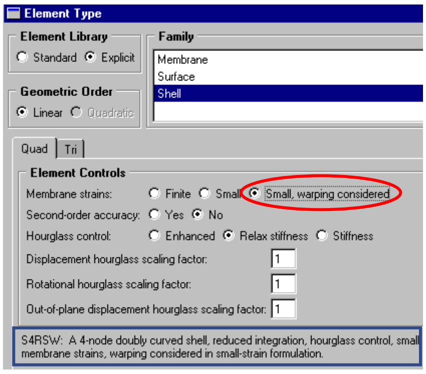
- **Small-strain shell elements in ABAQUS/Explicit**
  - The small-strain shell elements use a Mindlin-Reissner type of flexural theory that includes transverse shear.
  - **S4RS**
    - The S4RS quadrilateral shell element with reduced integration for small-strain problems is based on the formulation given by Belytschko, Lin, and Tsay (1984).
    - This is a very efficient shell element, and it is the default element in other explicit codes.



Conventional Shell Elements

– **S4RSW**

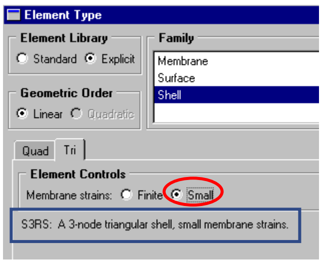
- The S4RSW quadrilateral shell element with reduced integration for small-strain problems and warped configurations is based on the formulations given by Belytschko, Wong, and Chiang (1992).
- In this element additional terms have been added to account for warped configurations so that the originally twisted, cantilever beam problem can be solved correctly (about 20% additional cost).



Conventional Shell Elements

– **S3RS**

- This triangular shell element is based on the formulation of S4RS.
- It is not subject to the zero energy modes inherent in the quadrilateral element formulation; therefore, hourglass control is not used.
- Hence, significant CPU time can be saved compared to using degenerate S4RS elements for problems incorporating many triangular shell elements.



Conventional Shell Elements

- The small strain shell elements are appropriate for modeling structures undergoing small membrane strains and finite rotations.
  - The dynamic impact of many structures, where large-scale buckling can occur but the membrane stretching and compression remains small, falls within this class of problem.
  - Typical metal forming problems will require a finite-strain shell formulation.
  - No warning is given if these elements are used inappropriately (i.e., when the membrane strains become large) because no criteria to judge their performance are generally applicable.
  - If there are significant differences between logarithmic strains and nominal strains, it probably is a large-strain problem.
  - Try a prototype analysis with both element types, and judge which element type to use in future analyses.

Conventional Shell Elements

- The small strain shell elements can reduce the computational cost of ABAQUS/Explicit analyses significantly because fewer element calculations are needed.
- The Head Impact problem was supplied by TRW and consists of 799 triangular shell elements and 12589 quadrilateral shell elements.
- All problems were run on an SGI R8000.

Problem	S4R	S4RS	S4RSW
Cylindrical panel	347	222 (63%)	252 (72%)
Pipe whip	200	132 (66%)	151 (75%)
Head impact	37,297	28,431 (76%)	31,425 (84%)

Comparing CPU times for finite- and small-strain shells

Conventional Shell Elements

– Transverse shear stiffness for small strain shell elements:

  • When the ∗SHELL SECTION option is used, these elements use the constitutive equations directly to calculate the transverse shear stresses.

  • When the ∗SHELL GENERAL SECTION option is used, the transverse shear stiffness for the small-strain shell elements is the same as that for the finite-strain shell elements.

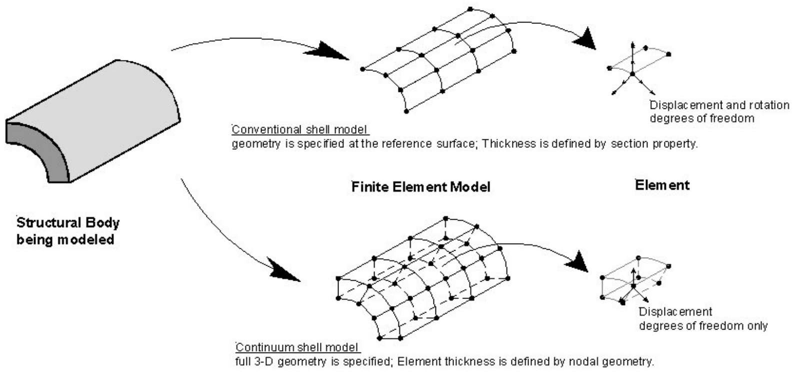
– Hourglass control for small strain shell elements:

  • An additional third parameter controls the hourglass stiffness associated with the out-of-plane displacement degree of freedom.

Continuum Shell Elements

Continuum Shell Elements

- **Overview**
  - Continuum shell elements are three-dimensional stress/displacement elements for use in modeling structures that are generally slender, with a shell-like response but continuum element topology.



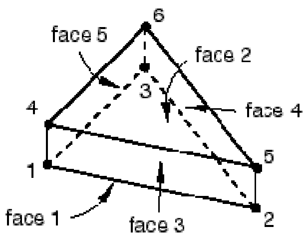
Continuum Shell Elements

- **The elements allow for:**
  - Thick and thin shell applications.
  - Linear and nonlinear behavior (both large deformation and elastic-plastic material response).
  - Thickness tapering.
    - The elements derive from 3-D meshed geometry.
  - A high aspect ratio between in-plane dimensions and the thickness.
  - More accurate contact modeling than conventional shells.
    - They take into account two-sided contact and thickness changes.
  - Stacking.
    - They capture more accurately the through-thickness response for composite laminate structures.

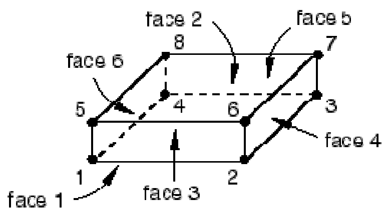
Continuum Shell Elements

- **Element topology**
  - Two elements topologies are available in ABAQUS

SC6R	6-node triangular in-plane continuum shell wedge, general-purpose, finite membrane strains
SC8R	8-node hexahedron, general-purpose, finite membrane strains



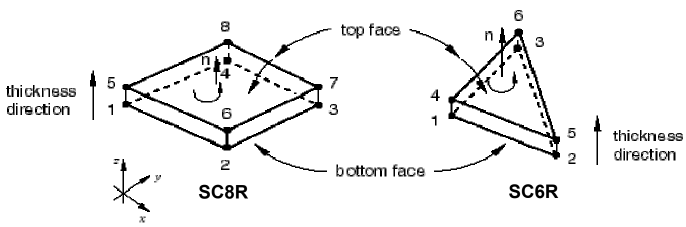
6-node continuum-shell



8-node continuum-shell

Continuum Shell Elements

- **Default thickness direction**
  - The kinematic response in the thickness direction is different from that in the in-plane directions for the continuum shell.
  - The thickness direction can be ambiguous for the SC8R element.
    - Any of the 6-faces could be the bottom face.
  - The default behavior uses the nodal connectivity:



Continuum Shell Elements

- **Alternative methods for defining the thickness direction**
  - Define the thickness direction based on the element isoperimetric directions.

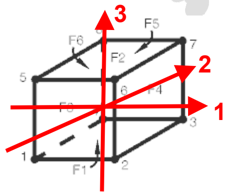
    `*SHELL SECTION, STACK DIRECTION=n`

    `*SHELL GENERAL SECTION, STACK DIRECTION=n`

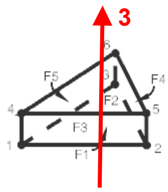
  - Define the thickness direction based on material orientations

    `*SHELL SECTION, STACK DIRECTION=ORIENTATION, ORIENTATION=name`

    `*SHELL GENERAL SECTION, STACK DIRECTION=ORIENTATION, ORIENTATION=name`



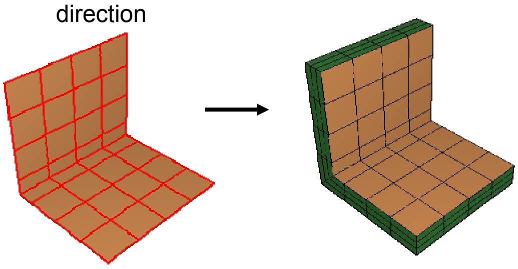
SC8R

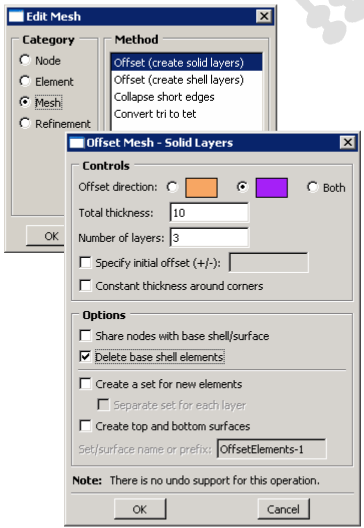


SC6R

Continuum Shell Elements

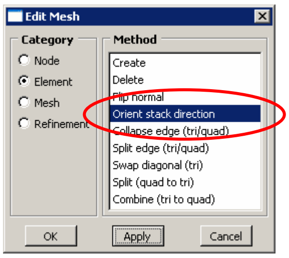
- **Modeling in ABAQUS/CAE**
  - Offset a shell mesh to generate layers of solid elements.
  - The starting point is a shell orphan mesh.
  - Shell mesh is “thickened” by offsetting nodes normal to the boundary and building elements that propagate out in the normal direction

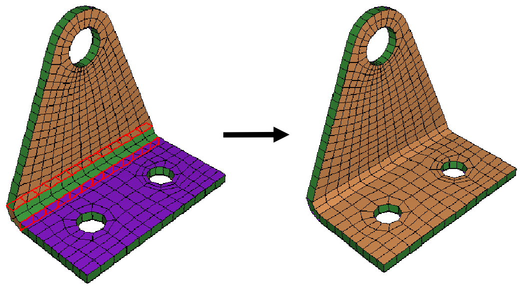




Continuum Shell Elements

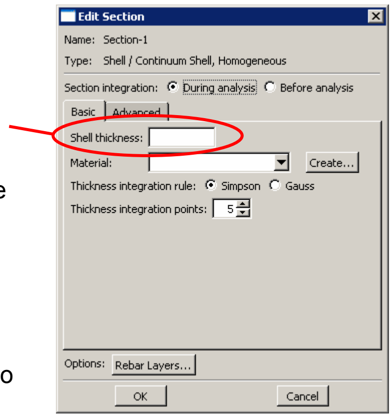
- **Controlling the stack orientation in ABAQUS/CAE**
  - Selected elements are oriented with respect to a reference top face.
  - Node labels, element labels, and node coordinates are not altered.
  - Surfaces are managed during the transformation.
  - The tool is only available for orphan meshes.
    - Orphan mesh parts can be created from meshed native geometry.





Continuum Shell Elements

- **Shell thickness**
  - The thickness of the element is **always** taken from the nodal coordinates.
  - A thickness value **must** be given on the data line of the *SHELL SECTION or *SHELL GENERAL SECTION option.
    - However, it is only used to calculate some initial properties, such as hourglass stiffness, which are then appropriately scaled to the element thickness.
  - The thickness may be continuously varying (tapered shells) without having to specify *NODAL THICKNESS.



Continuum Shell Elements

- **Change in thickness**
  - The change in thickness is calculated from the nodal displacements, an effective thickness modulus, and an effective section Poisson's ratio.
  - ABAQUS computes an effective thickness strain based on the total thickness change and a strain obtained by enforcing the plane stress condition for a section Poisson's ratio.

$$\varepsilon_{\text{eff}} = \varepsilon_{33} - \varepsilon_{\text{ps}}$$

where

ε_{33} is the thickness strain computed from nodal displacements,

ε_{ps} is the strain obtained by enforcing the plane stress condition for a section Poisson's ratio, and

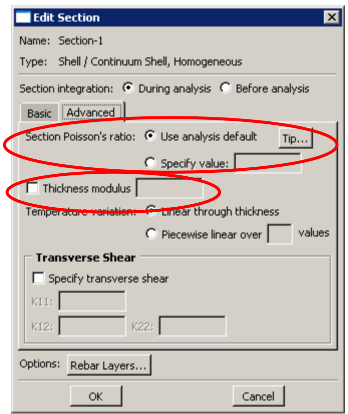
ε_{eff} is the effective thickness strain assumed to be elastic and based on the thickness modulus.

Continuum Shell Elements

- **By default**, the thickness modulus and section Poisson's Ratio are based on the initial material properties.
- Alternatively, you may define an effective thickness modulus and effective section Poisson's Ratio as part of the shell section definition.
  - Input File Usage:

```
*SHELL SECTION, POISSON=v,
  THICKNESS MODULUS=e

*SHELL GENERAL SECTION,
  POISSON=v, THICKNESS MODULUS=e
```



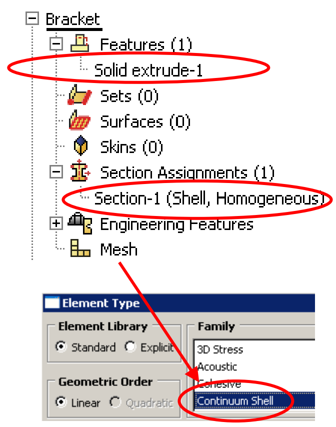
Continuum Shell Elements

- **Surfaces, contact, and coupling**
  - Surfaces are defined on the element in the same way that they are defined on continuum solid elements.
    - All surface-based loads are activated (top, bottom, and edges).
  - Contact takes place on the **actual** shell surface, not the reference surface.
  - Double-sided contact is permitted.
  - Coupling is fully supported. Matching meshes are not required for:
    - Continuum shell to continuum shell elements (*TIE).
    - Continuum shell to continuum solid elements (*TIE).
      - Can be directly connected for matching meshes, however.
    - Conventional shell to continuum shell coupling (*SHELL TO SOLID COUPLING).

Continuum Shell Elements

- **The user interface looks like the interface for continuum solid elements (where appropriate) or conventional shell elements (where appropriate).**

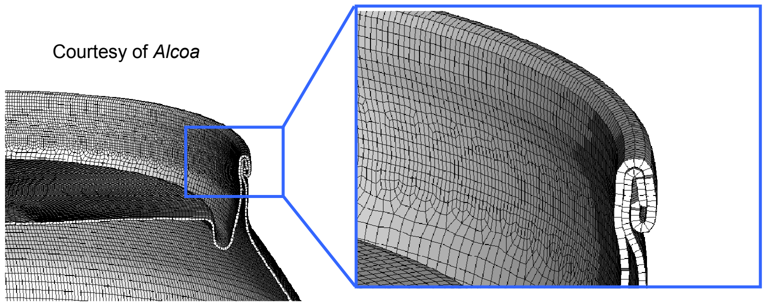
```
*element, type=SC6R, elset=triangles
*element, type=SC8R, elset=quads
*shell section, elset=triangles,
  material=steel, poisson=v,
  thickness modulus=e
*shell section, elset=quads, composite,
  orientation=orient,
  stack direction = {1|2|3|orientation}
thickness, # sect pts, material, orientation
*material, name=steel
*elastic
*plastic
  …
```



Continuum Shell Elements

- **Example: Can forming problem**
  - Here we are modeling the process that forms the lip/seam between the top of the can and the sidewall.
    - Difficulties are encountered using conventional shell elements.
    - The problem is readily solved with continuum shell elements.

Courtesy of *Alcoa*



Continuum Shell Elements

- **Limitations**
  - Continuum shell elements **cannot be used with the hyperelastic or hyperfoam** material models.
  - Continuum shell elements **cannot be used with *SHELL GENERAL SECTION** where the section stiffness is provided directly on the data lines.
  - Although continuum shells provide robust and accurate solutions to most shell applications, conventional shells are a better choice for **very thin shell applications**.
  - In ABAQUS/Explicit the element stable time increment can be controlled by the continuum shell element thickness.
    - This may significantly increase the number of increments taken to complete the analysis when compared to the same problem modeled with conventional shell elements.
    - The small stable time increment size may be mitigated by specifying a lower stiffness in the thickness direction when appropriate.

ABAQUS

Continuum Shell Elements

- **ABAQUS/CAE visualization support currently limited to:**
  - Plotting deformed and undeformed model shapes
  - Plotting material orientations and thickness directions
  - Contouring material quantities by specifying a particular section, similar to conventional shells.
    - Material quantities for a specific section are contoured over the entire element.
    - This may be misleading since in most case the stress at the top and bottom surface may differ when bending occurs!