Model created in COMSOL Multiphysics 6.4

Bracket — Shell Analysis

Introduction

The various examples based on a bracket geometry form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

In this example you study the stress in a bracket subjected to external loads. The thin parts with constant thickness are modeled using the Shell interface, and the transition regions where 3D effects are important are modeled using the Solid Mechanics interface. This example also shows how to connect shell elements with solid elements.

For thin geometries, it can be more efficient to use shell elements than solid elements, thus saving computational time and memory. The Shell interface in the Structural Mechanics Module can be used to model structures approximated by thin or thick shells. There is also a similar Plate interface for 2D problems. The thickness of the shell or plate is taken into account in the equations instead of being explicitly modeled in the geometry.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

The model is described in the section “The Fundamentals: A Static Linear Analysis” in the *Introduction to the Structural Mechanics Module*.

The parts of the geometry modeled with shells are highlighted in Figure 1 and the parts modeled with solids are highlighted in Figure 2.

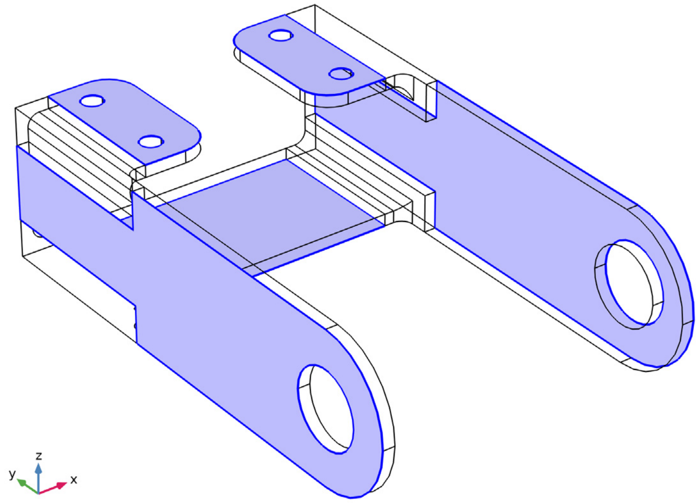

Figure 1: Shell domains in the bracket geometry.

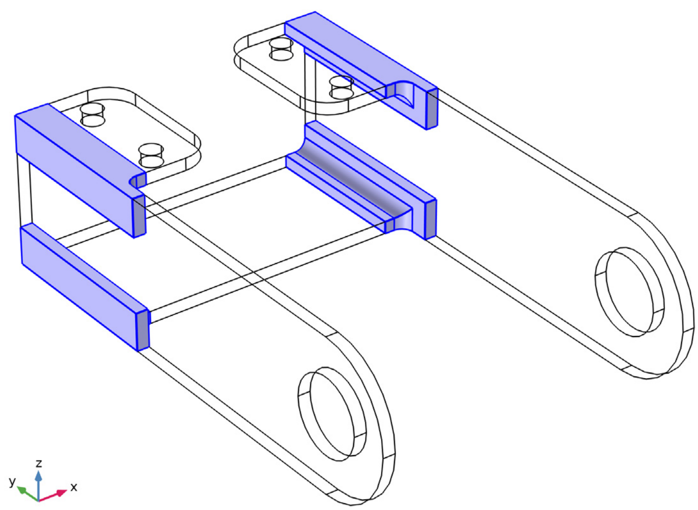

Figure 2: Solid domains in the bracket geometry.

The load is the same as in the solid model which forms the basis, but is now applied along the edges at the bracket holes.

Results and Discussion

The Shell interface generates a plot which indicates the physical location of top and bottom surfaces. Especially when working with offsets, as in the current example, this is an excellent tool for checking that the input data is correct. This plot is shown in Figure 3.

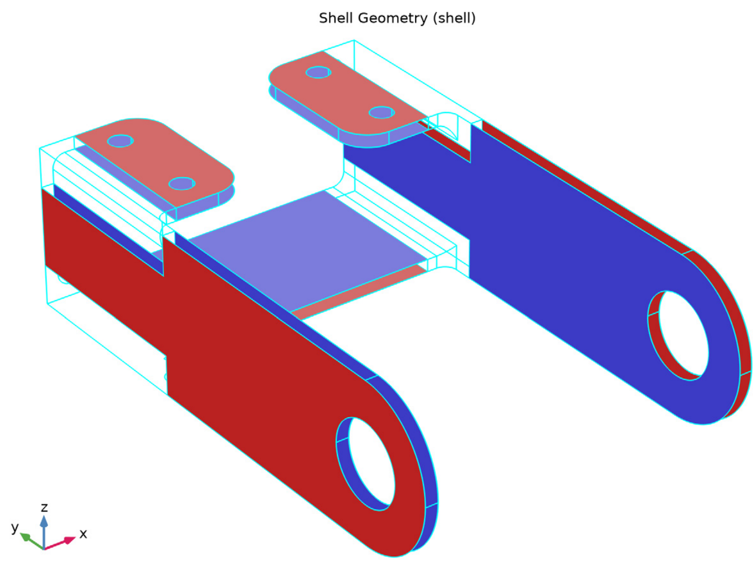

Figure 3: The shell geometry plot. Red indicates top surface and blue indicates bottom surface.

In another plot, also available under **Result Templates**, the thickness is indicated by color contours, and the directions of the shell local coordinate systems are shown. This plot is shown in Figure 4.

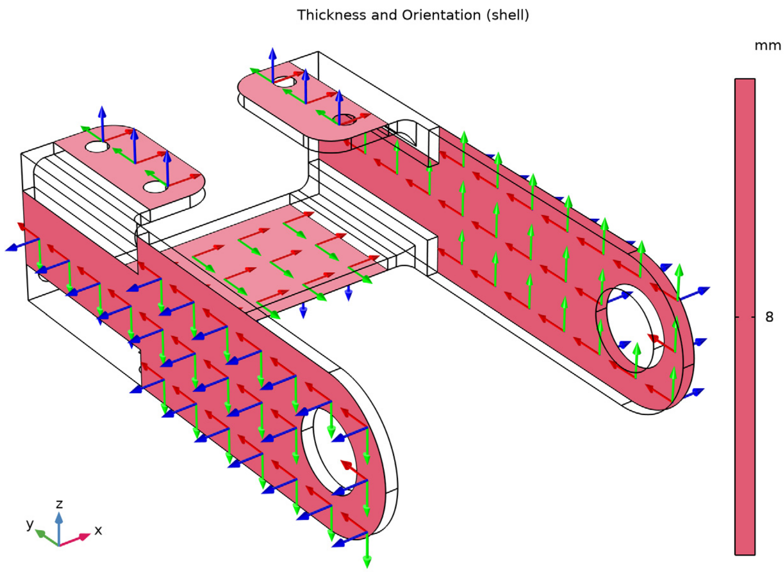

Figure 4: Plot of thickness and local directions. The Red-Green-Blue convention is used for ordering of the coordinate axes.

There are several types of automatic connections available in the Shell interface. When using such features, it is recommended that you inspect that the connections have been applied as intended. In Figure 5, such a plot is shown. It indicates the boundaries of the solid that are connected to shell elements.

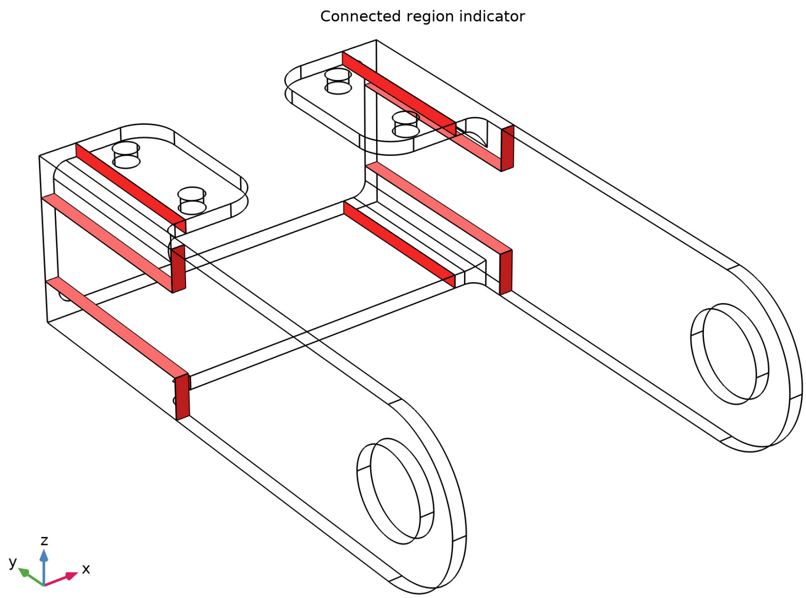

Figure 5: Generated connections between shell and solid elements.

Figure 6 shows the first principal strain in both the solid domains and in the shells. When using the Shell dataset, the shell is represented by a solid with thickness and offset taken from the Shell interface. The through-thickness representation of, for example, stresses is also full 3D. You can increase the resolution of the evaluation in the thickness direction, but using a high resolution may give slower plotting.

As can be seen, the continuity over the transition between the shell and the solid is very good.

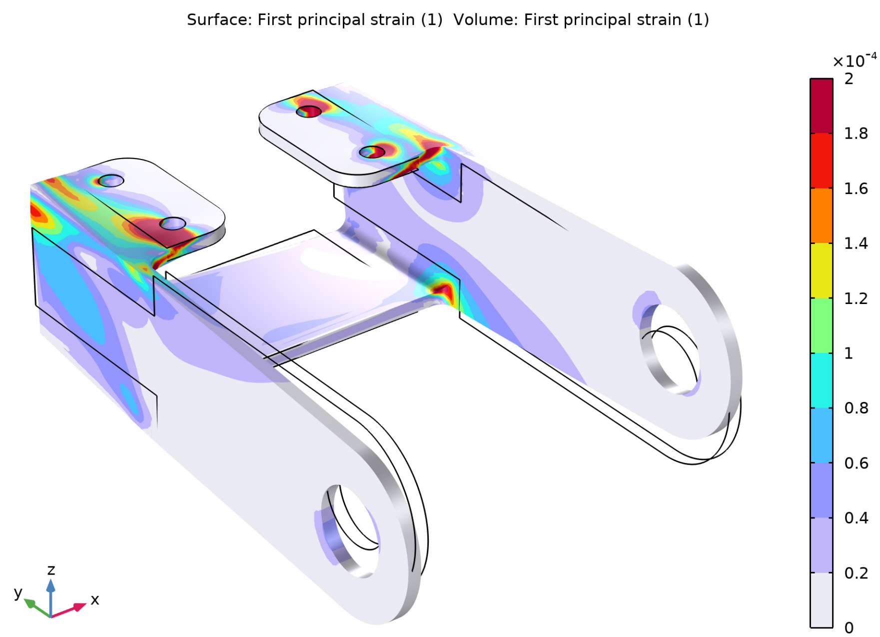

Figure 6: First principal strain distribution in the solid and at the top and bottom of the shell.

A special result type in the shell elements are the *section forces*. The section forces contain the stresses integrated through the thickness of the shell and represent three fundamental types of action:

- Membrane (`shell.Nl11`, `shell.Nl22`, and `shell.Nl12`)
- Bending (`shell.Ml11`, `shell.Ml22`, and `shell.Ml12`)
- Out-of-plane shear (`shell.Ql1` and `shell.Ql2`)

The section forces are always aligned with the shell local system.

As an example of section forces, they evaluated along a cut line through the left arm of the bracket, shown in Figure 7. In this cross section, it is reasonable to expect that the stress state is similar to that of a rectangular beam, subjected to a transverse force at the hole location.

In order to know which components of the section force to study, the local directions can be examined in Figure 4. The first direction is the one along the arm, so `shell.Nl11` corresponds to the bending stresses in a beam and `shell.Nl12` corresponds to the shear stresses.

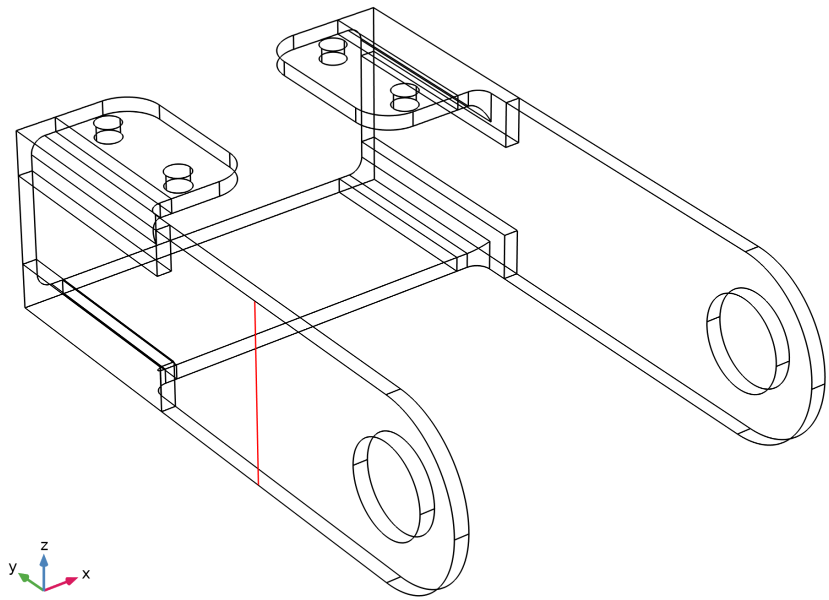

Figure 7: The cut line used for examining the section forces.

The relevant membrane forces along the cut are shown in Figure 8.

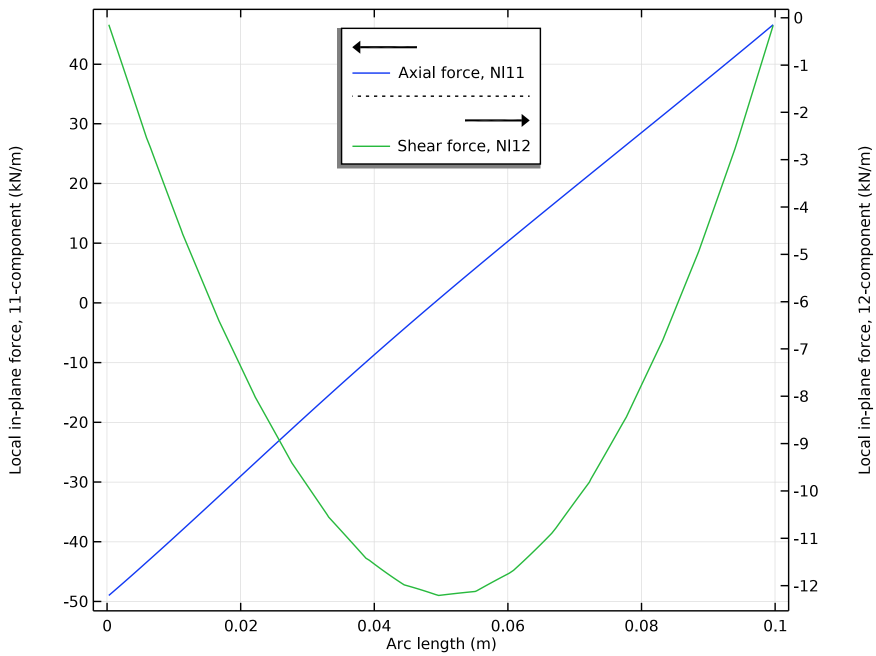

Figure 8: Membrane forces along the cut line.

The values can be compared with analytical results according to beam theory. As the cut is located at a distance from the hole center of a = 100 mm, the bending moment is

$$M = Fa \tag{1}$$

The peak bending stress in a rectangular beam is

$$\sigma = \frac{6M}{th^2} \tag{2}$$

where t is the thickness and h is the height. The distribution along the cut should be linear.

The section force is the stress multiplied by the thickness, giving the peak value

$$N_{11} = \frac{6Fa}{h^2} = \frac{6 \cdot 800\ \mathrm{N} \cdot 100\ \mathrm{mm}}{(100\ \mathrm{mm})^2} = 48\ \mathrm{kN/m} \tag{3}$$

According to beam theory, the shear stress has a parabolic distribution with the peak value

$$\tau = \frac{3F}{2bt} \tag{4}$$

Thus, the peak value of the shear membrane force is

$$N_{12} = \frac{3F}{2b} = \frac{3 \cdot 800\ \mathrm{N}}{2 \cdot 100\ \mathrm{mm}} = 12\ \mathrm{kN/m} \tag{5}$$

As can be seen, the stress state in this part of the structure is very close to that of a beam. Checks like this can be valuable for model verification.

Notes About the COMSOL Implementation

You can specify an offset in the shell definition that the meshed surface is not the same as the midsurface of the real geometry. This is used here, since the external geometrical boundaries are immediately available.

The **Solid–Thin Structure Connection** multiphysics coupling is used for connecting shell edges to solid boundaries.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_shell`

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module** > **Tutorials** > **bracket_static** in the tree.

3 Click **Open**.

COMPONENT 1 (COMP1)

Start by removing the results from the loaded model.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

RESULTS

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** > **Study 1/Solution 1 (sol1)** and choose **Delete**.

3 Right-click **Results** > **Tables** and choose **Delete All**.

COMPONENT 1 (COMP1)

Add a Shell interface to the model.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics** > **Shell (shell)**.

4 Click the **Add to Component 1** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

DEFINITIONS

Add a number of selections.

Solid

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Solid` in the **Label** text field.

3 Select Domains 2, 3, 7, and 8 only.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Shell

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Shell` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 32, 37, 50, and 86 only.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solid**.

Disable not used boundary conditions from the Solid Mechanics interface. Both the loads and constraints will be applied to the shell part. This is just to clean up the Model

Builder tree. Since these nodes now have an empty selection, they would not influence the analysis if kept.

4 In the **Model Builder** window, expand the **Solid Mechanics (solid)** node.

Boundary Load 1, Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)**, Ctrl-click to select **Fixed Constraint 1** and **Boundary Load 1**.

2 Right-click and choose **Group**.

Not Used; Applied in Shell Interface

1 In the **Settings** window for **Group**, type `Not Used; Applied in Shell Interface` in the **Label** text field.

2 Right-click **Not Used; Applied in Shell Interface** and choose **Disable**.

Add settings to the Shell interface. Since the mesh is placed on the outer boundary, the location of the midsurface is described by an offset.

SHELL (SHELL)

1 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

3 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

4 From the **Selection** list, choose **Shell**.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d_0 text field, type `8[mm]`.

4 From the **Position** list, choose **Top surface on boundary**.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Bolt Hole Edges**.

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 In the **Settings** window for **Edge Load**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

4 Select Edges 4 and 188 only.

5 Locate the **Force** section. From the **Load type** list, choose **Force per reference area**.

6 Specify the $\mathbf{f}_A$ vector as

p0*cos(sys2.phi)	r

MULTIPHYSICS

Add the connection between the shells and the solids.

Solid–Thin Structure Connection 1 (sshc1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global** > **Solid–Thin Structure Connection**.

2 In the **Settings** window for **Solid–Thin Structure Connection**, locate the **Connection Settings** section.

3 Select the **Manual control of selections** checkbox.

4 Select Boundaries 9, 11, 13, 14, 30, 34, 58, 62, 80, 81, 83, and 84 only.

5 From the **Method** list, choose **Flexible**.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

MATERIALS

The current material is attached only to the domains. A separate material definition is needed for the boundaries where the Shell interface is active.

Structural steel 1 (mat2)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** node.

2 Right-click **Component 1 (comp1)** > **Materials** > **Structural steel (mat1)** and choose **Duplicate**.

3 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Shell**.

STUDY 1

Run the analysis.

In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot for the Shell interface shows the von Mises stress using a **Shell** dataset. Create a combined stress plot with the results from both interfaces by copying the default stress plot for the solid.

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node.

2 Right-click **Volume 1** and choose **Copy**.

Stress, Solid + Shell

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, Solid + Shell` in the **Label** text field.

Volume 1

1 Right-click **Stress, Solid + Shell** and choose **Paste Volume**.

2 In the **Settings** window for **Volume**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Surface 1**.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

Surface 1

1 In the **Model Builder** window, click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 In the **Maximum** text field, type `70`.

5 In the **Stress, Solid + Shell** toolbar, click **Plot**.

Add plots from **Result Templates**, showing the shell geometry, the thickness and local shell system orientation, and the solid-shell connections. The first plot shows the top surface in red and the bottom surface in blue.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Shell Geometry (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Thickness and Orientation (shell)**.

6 Click the **Add Result Template** button in the window toolbar.

7 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid–Thin Structure Connection 1** > **Connected Region Indicator (sshc1)**.

8 Click the **Add Result Template** button in the window toolbar.

9 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Shell Geometry (shell)

1 In the **Shell Geometry (shell)** toolbar, click **Plot**.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

Thickness and Orientation (shell)

1 In the **Model Builder** window, click **Thickness and Orientation (shell)**.

2 In the **Thickness and Orientation (shell)** toolbar, click **Plot**.

Connected Region Indicator (sshc1)

1 In the **Model Builder** window, click **Connected Region Indicator (sshc1)**.

2 In the **Connected Region Indicator (sshc1)** toolbar, click **Plot**.

Create a plot of the maximum principal stresses.

Principal Strain

1 In the **Model Builder** window, right-click **Stress, Solid + Shell** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Principal Strain` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Principal Strain** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `shell.ep1`.

4 Locate the **Range** section. In the **Maximum** text field, type `2.0E-4`.

5 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

Volume 1

1 In the **Model Builder** window, click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.ep1`.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

5 In the **Principal Strain** toolbar, click **Plot**.

Create a cut line, and plot section forces along it.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `-0.215/2`.

4 In row **Point 2**, set **X** to `-0.215/2`.

5 In row **Point 1**, set **Y** to `-0.2`.

6 In row **Point 2**, set **Y** to `-0.2`.

7 In row **Point 1**, set **Z** to `-0.1`.

8 In row **Point 2**, set **Z** to `0.1`.

9 From the **Snapping** list, choose **Snap to closest boundary**.

10 Click **Plot**.

Section Forces at Cut

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Section Forces at Cut` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 3D 1**.

Line Graph 1

1 Right-click **Section Forces at Cut** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Shell** > **Section forces** > **Local in-plane force - N/m** > **shell.Nl11 - Local in-plane force, 11-component**.

3 Locate the **y-Axis Data** section. From the **Unit** list, choose **kN/m**.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Axial force, Nl11`

7 In the **Section Forces at Cut** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `shell.Nl12`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Shear force, Nl12`

5 In the **Section Forces at Cut** toolbar, click **Plot**.

Section Forces at Cut

1 In the **Model Builder** window, click **Section Forces at Cut**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

5 In the table, select the **Plot on secondary y-axis** checkbox for **Line Graph 2**.

6 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

7 In the **Section Forces at Cut** toolbar, click **Plot**.