Created in COMSOL Multiphysics 6.1

# 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.

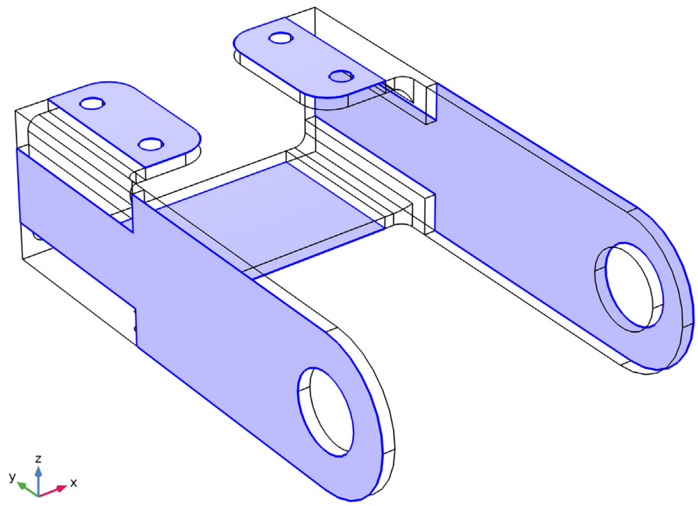


*Figure 1: Shell domains in the bracket geometry.*

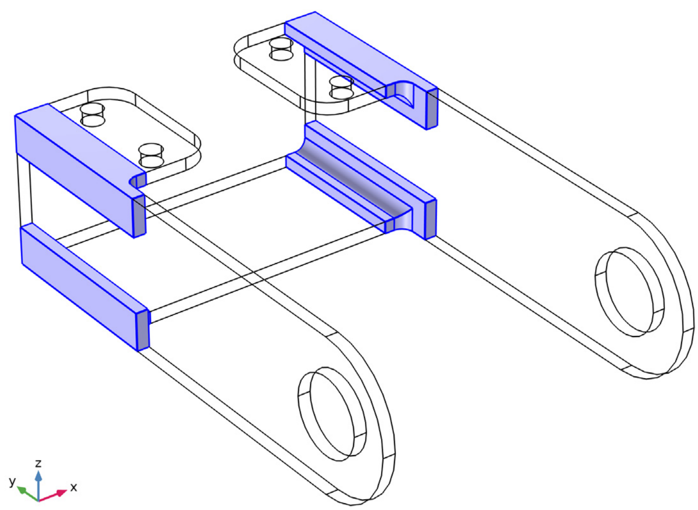


*Figure 2: Solid domains in the bracket geometry.*

The load is applied along the edge at the bracket holes.

## *Results and Discussion*

The Shell interface generates a predefined 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.

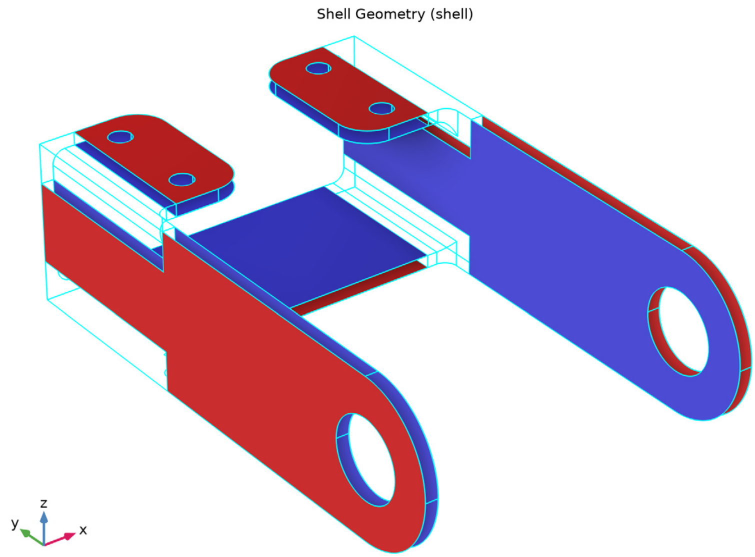


*Figure 3: The shell geometry plot. Red indicates top surface and blue indicates bottom surface.*

In another predefined plot, 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.

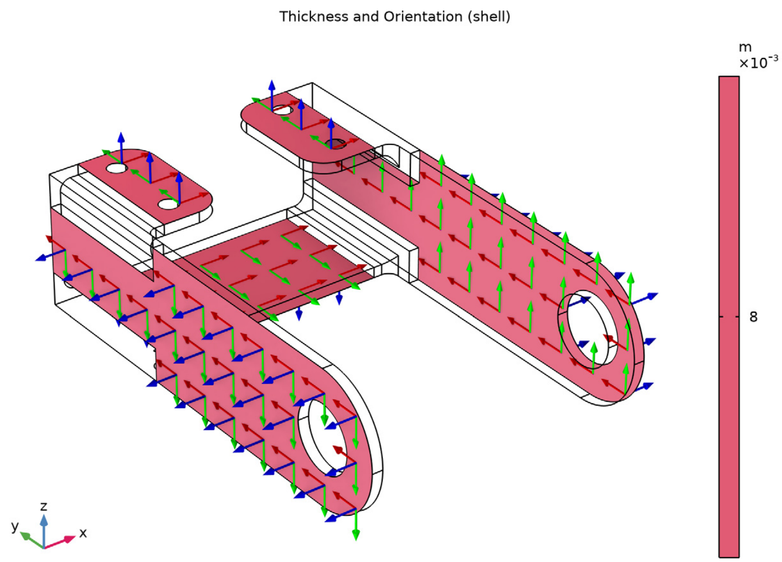


*Figure 4: Plot of thickness and local directions. The Red-Green-Blue convention is used for ordering of the coordinate axes.*

Figure 5 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.

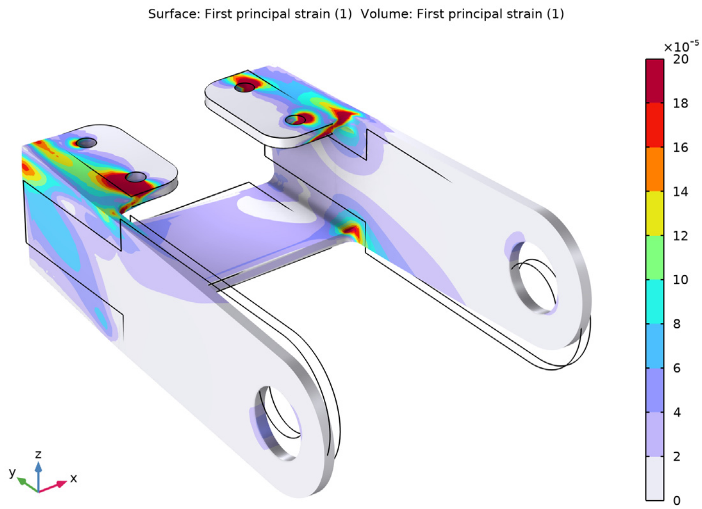


*Figure 5: First principal strain distribution in the solid and at the top and bottom of the shell.*

## 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** 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**.

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 **Add to Component 1** in the window toolbar.

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

Add a number of selections.

## DEFINITIONS

*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 **Local edge system**.

**4** Select Edges 4 and 188 only.

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

In order to use the local edge system, you need to know the orientation of the edges and the positive shell normal orientation. The orientation of the shell normal can be displayed by enabling physics symbols, and then moving to the **Linear Elastic Material** node. The line orientation arrows are controlled from the **View** node.

**6** In the **Model Builder** window, click **Shell (shell)**.

**7** In the **Settings** window for **Shell**, locate the **Physics Symbols** section.

**8** Select the **Enable physics symbols** check box.

## DEFINITIONS

### *View 1*

**1** In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **View 1**.

**2** In the **Settings** window for **View**, locate the **View** section.

**3** Select the **Show edge direction arrows** check box.

## SHELL (SHELL)

### *Edge Load 1*

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

**2** In the **Settings** window for **Edge Load**, locate the **Force** section.

**3** Specify the $\mathbf{F}_A$ vector as

| | |
|---|---|
| `0` | xl |
| `load(P0*sign(X),Y-YC,Z)` | yl |
| `0` | zl |

Add the connection between the shells and the solids.

## MULTIPHYSICS

### *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** check box.
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.

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

## MATERIALS

### *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**.

Run the analysis.

## STUDY 1

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

## RESULTS

### *Stress (solid)*

The default plot for the Shell interface shows the von Mises stress on top and bottom surfaces. 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)**.

5 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

### *Surface 1*

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

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

3 From the **Unit** list, choose **MPa**.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

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

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

Add predefined plots showing the shell geometry as well as the thickness and local shell system orientation. The first plot shows the top surface in red and the bottom surface in blue.

7 In the **Home** toolbar, click **Add Predefined Plot**.

## ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

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

3 Click **Add Plot** in the window toolbar.

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

5 Click **Add Plot** in the window toolbar.

6 In the **Home** toolbar, click **Add Predefined Plot**.

7 Click **Add Predefined Plot**.

## RESULTS

### *Shell Geometry (shell)*

1 In the **Model Builder** window, under **Results** click **Shell Geometry (shell)**.
2 In the **Shell Geometry (shell)** toolbar, click **Plot**.
3 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**.

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 **Minimum** text field, type `0`.
5 In the **Maximum** text field, type `2.0E-4`.
6 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**.