

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 and 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 default 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 default 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 on a in both the solid domains and at the top and bottom shell surfaces. When using the Shell dataset, the shell is represented by two surface plots, one on each side of the physical shell.

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 if 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

From the File menu, choose Open.

Browse to the model's Application Libraries folder and double-click the file bracket_static.mph.

COMPONENT I (COMPI)

Start by removing the results from the loaded model.

I In the Model Builder window, expand the Component I (compl) node.

RESULTS

Study I/Solution I (soll)

- I In the Model Builder window, expand the Results>Datasets node.
- 2 Right-click Results>Datasets>Study I/Solution I (soll) and choose Delete.
- 3 Right-click Results>Tables and choose Delete All.

Add a Shell interface to the model.

ADD PHYSICS

- I 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 I 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

- I 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.

Shell

- I In the **Definitions** toolbar, click **here 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.

SOLID MECHANICS (SOLID)

- I In the Model Builder window, under Component I (compl) 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 I, Fixed Constraint I

- I In the Model Builder window, under Component I (compl)>Solid Mechanics (solid), Ctrlclick to select Fixed Constraint I and Boundary Load I.
- 2 Right-click and choose Group.

Not Used; Applied in Shell Interface

- I 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)

- I In the Model Builder window, under Component I (compl) click Shell (shell).
- 2 In the Settings window for Shell, locate the Boundary Selection section.
- 3 From the Selection list, choose Shell.

Thickness and Offset I

- I In the Model Builder window, under Component I (compl)>Shell (shell) click Thickness and Offset I.
- 2 In the Settings window for Thickness and Offset, locate the Thickness and Offset section.
- **3** In the *d* text field, type 8[mm].
- 4 From the Offset definition list, choose Relative offset.
- **5** In the $z_{\text{reloffset}}$ text field, type -1.
- 6 Click the 🖂 Wireframe Rendering button in the Graphics toolbar.

Fixed Constraint I

- I 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 I

- I 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 I

- I In the Model Builder window, expand the Component I (compl)>Definitions>View I node, then click View I.
- 2 In the Settings window for View, locate the View section.
- **3** Select the **Show edge direction arrows** check box.

SHELL (SHELL)

Edge Load I

I In the Model Builder window, under Component I (compl)>Shell (shell) click Edge Load I.

- 2 In the Settings window for Edge Load, locate the Force section.
- **3** Specify the \mathbf{F}_A vector as

0	xl
<pre>load(P0*sign(X),Y-YC,Z)</pre>	yl
0	zl

Add the connection between the shells and the solids.

MULTIPHYSICS

Solid-Thin Structure Connection 1 (sshc1)

- I In the Physics toolbar, click A 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.

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)

- I In the Model Builder window, expand the Component I (compl)>Materials node.
- 2 Right-click Component I (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 I

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

RESULTS

Stress (solid)

The first 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.

Surface 1

- I In the Model Builder window, expand the Stress (solid) node.
- 2 Right-click Surface I and choose Copy.

Surface 2

- I In the Model Builder window, right-click Stress (shell) and choose Paste Surface.
- 2 In the Settings window for Surface, click to expand the Inherit Style section.
- 3 From the Plot list, choose Surface I.
- 4 Locate the Data section. From the Dataset list, choose Study I/Solution I (soll).

Surface 1

- I In the Model Builder window, click Surface I.
- 2 In the Settings window for Surface, click to expand the Range section.
- 3 Select the Manual color range check box.
- 4 In the Minimum text field, type 0.
- 5 In the Maximum text field, type 7e7.
- 6 In the Stress (shell) toolbar, click **I** Plot.

The second default plot shows the shell geometry, with the top surface colored in bluegreen and the bottom surface colored in white.

Shell Geometry (shell)

- I In the Model Builder window, click Shell Geometry (shell).
- 2 In the Shell Geometry (shell) toolbar, click 🗿 Plot.
- **3** Click the $4 \rightarrow$ **Zoom Extents** button in the **Graphics** toolbar.

The third default plot shows the shell thickness and the orientation of the local coordinate system.

Thickness and Orientation (shell)

- I In the Model Builder window, click Thickness and Orientation (shell).
- 2 In the Thickness and Orientation (shell) toolbar, click **O** Plot.

Create a plot of the maximum principal stresses.

Principal Strain

- I In the Model Builder window, right-click Stress (shell) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Principal Strain in the Label text field.

Surface 1

- I In the Model Builder window, expand the Principal Strain node, then click Surface I.
- 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.

Surface 2

- I In the Model Builder window, click Surface 2.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type solid.ep1.
- **4** Click the **Comextents** button in the **Graphics** toolbar.
- 5 In the Principal Strain toolbar, click 💽 Plot.