Created in COMSOL Multiphysics 6.1

# Bracket — Static 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.

This is the most fundamental model in the suite, showing a linear static analysis. It includes the definition of material properties and boundary conditions. After the solution is computed, you learn how to analyze results and check the reaction forces.

## Model Definition

The model used in this guide is a bracket made of steel. This type of bracket can be used to install an actuator that is mounted on a pin placed between the two holes in the bracket arms. The geometry is shown in Figure 1.

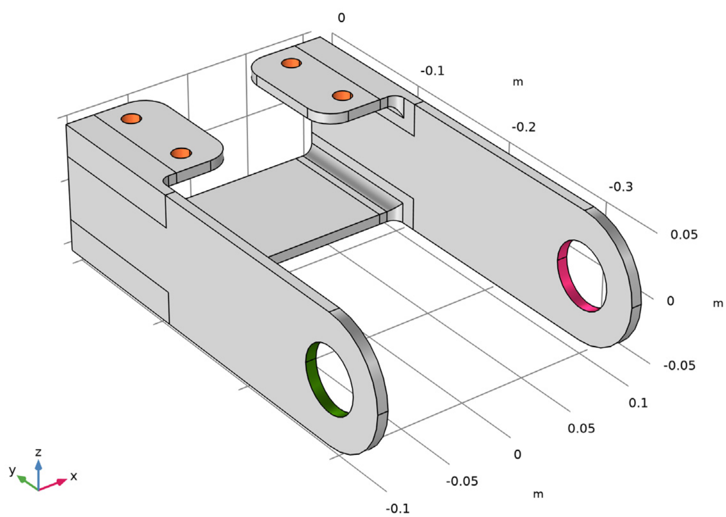



*Figure 1: Bracket geometry.*

In this analysis, the mounting bolts are assumed to be fixed and securely bonded to the bracket. One of the arms is loaded upward and the other downward. The loads are applied as a pressure on the inner surfaces of the holes, and their intensity is $P_0 \cos(\alpha)$, where $\alpha$ is the angle from the direction of the load resultants. Figure 2 below shows the loads applied to the bracket.

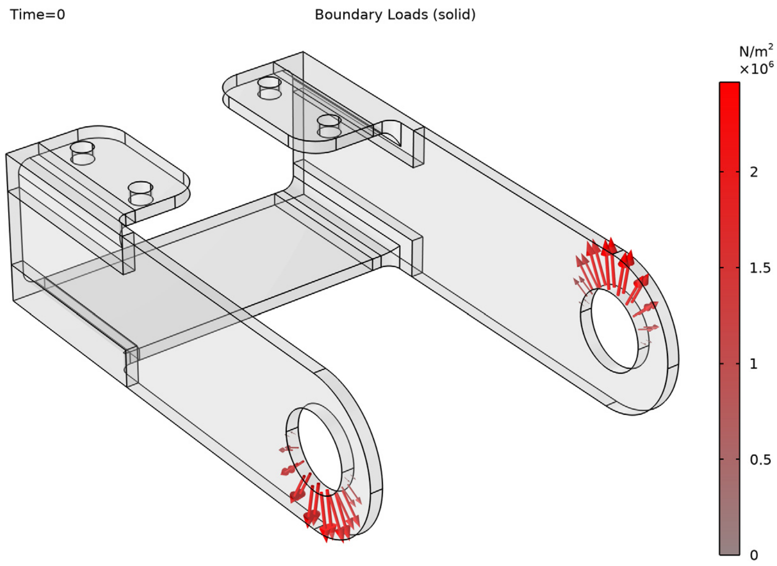



*Figure 2: Load distribution in the bracket arms.*

## Results

Figure 3 shows the von Mises stress distribution together with an exaggerated (automatically scaled) picture of the deformation. The high stress values are located in the vicinity of the mounting bolts and at the transition between the plates.

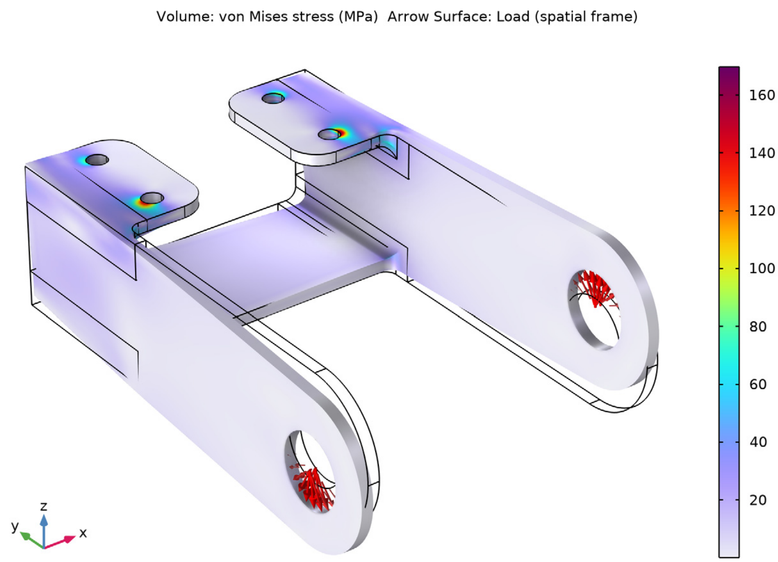



*Figure 3: Von Mises stress distribution in the bracket under a bending load.*

In Figure 4 you can see that the bracket base remains fixed while only the arms are deformed. The maximum total displacement is about 0.25 mm, which is in agreement with the assumption of small deformations.

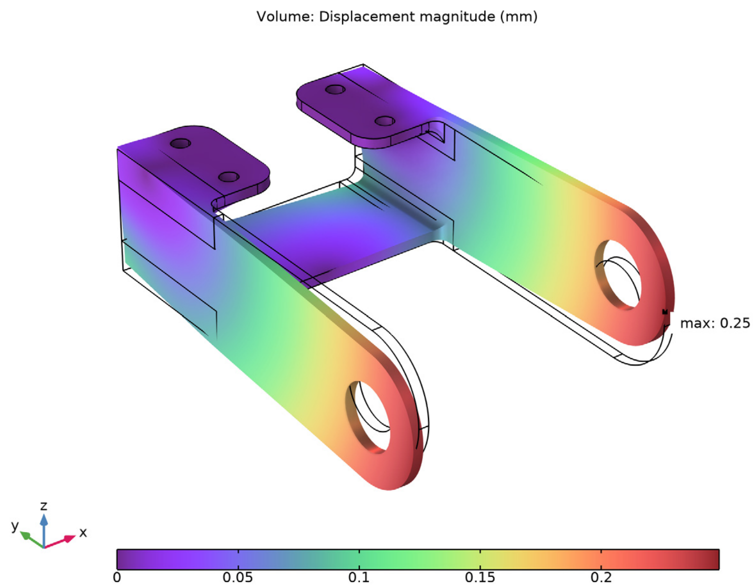



*Figure 4: Total displacement.*

Figure 5 shows the principal stresses in the bracket. The largest principal stress is shown with red arrows, the intermediate principal stress with green arrows, and the smallest principal stress with blue arrows. Since a state of plane stress prevails in large parts of the structure (the thin plates) one of the principal stresses is mostly zero.

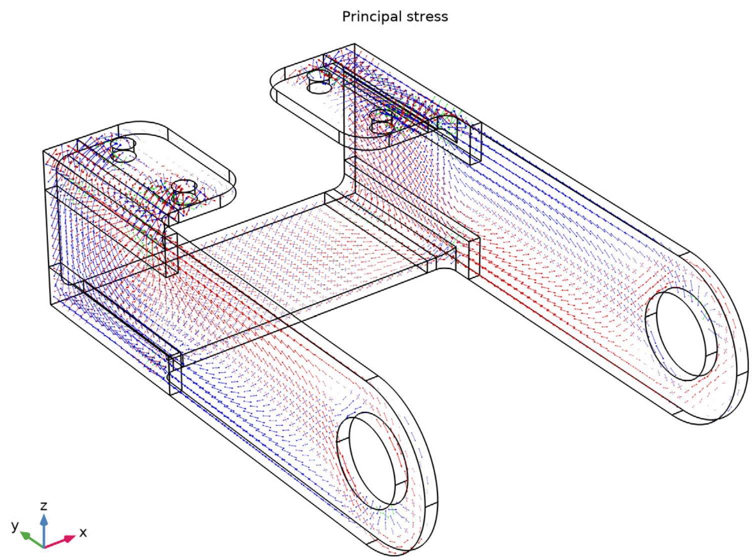



*Figure 5: Principal stress in the bracket left arm.*

In Table 1 you can see the reaction force in the $x$, $y$, and $z$ directions in each bolt. In all directions the sum is zero, which is a good check, since in this model there are no resultant forces. The slight asymmetry can be attributed to the mesh not being perfectly symmetric.

TABLE 1: REACTION FORCE IN BOLT.

| | **Reaction force, x direction (N)** | **Reaction force, y direction (N)** | **Reaction force, z direction (N)** |
|---|---|---|---|
| Bolt 1 | 451 | 307 | 2534 |
| Bolt 2 | -451 | 73 | -1702 |
| Bolt 3 | 452 | -307 | -2534 |
| Bolt 4 | -451 | -73 | 1702 |

**Application Library path:** `Structural_Mechanics_Module/Tutorials/bracket_static`

## *Modeling Instructions*

This example is the same as described in the *Introduction to the Structural Mechanics Module* document. If you are new to COMSOL Multiphysics, that may be a better starting point, since it contains a more detailed description.

From the **File** menu, choose **New**.

### NEW

In the **New** window, click **Model Wizard**.

### MODEL WIZARD

1 In the **Model Wizard** window, The first step to build a model is to open COMSOL and then specify the type of analysis you want to do - in this case, a stationary, solid mechanics analysis.

2 click **3D**.

3 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies>Stationary**.

7 Click **Done**.

### GLOBAL DEFINITIONS

It is good modeling practice to gather constants and parameters in one place so that you can change them easily. Using parameters will also improve the readability of your input data.

#### *Parameters 1*

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| P0 | 2.5[MPa] | 2.5E6 Pa | Peak load intensity |

### GEOMETRY 1

The next step is to create your geometry, which also can be imported from an external program. COMSOL Multiphysics supports a multitude of CAD programs and file formats. In this example, import a file in the COMSOL Multiphysics geometry file format (`.mphbin`).

### *Import 1 (imp1)*

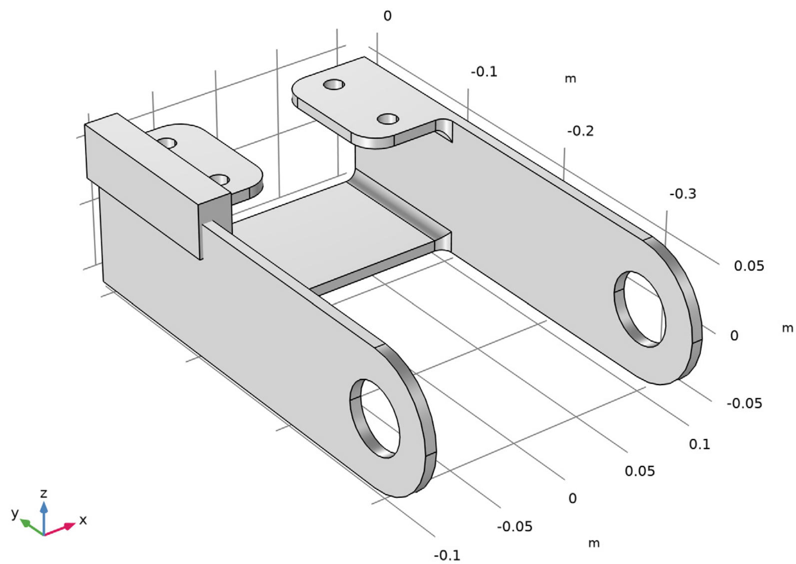
1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 From the **Source** list, choose **COMSOL Multiphysics file**.
4 Click **Browse**.
5 Browse to the model's Application Libraries folder and double-click the file `bracket.mphbin`.
6 Click **Import**.

### *Block 1 (blk1)*

It is possible to create a free tetrahedral mesh covering the whole component. Such a strategy is however not efficient for the large flat regions. For this reason, you will partition the geometry, so that you can create a better mesh.

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Selections of Resulting Entities** section.
3 Find the **Cumulative selection** subsection. Click **New**.
4 In the **New Cumulative Selection** dialog box, type `Partition Block` in the **Name** text field.
5 Click **OK**.
6 In the **Settings** window for **Block**, locate the **Size and Shape** section.
7 In the **Width** text field, type `0.025`.
8 In the **Depth** text field, type `0.13`.
9 In the **Height** text field, type `0.04`.
10 Locate the **Position** section. In the **x** text field, type `-0.11`.
11 In the **y** text field, type `-0.12`.
12 In the **z** text field, type `0.025`.

**13** Click **Build Selected**.




*Mirror 1 (mir1)*

**1** In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

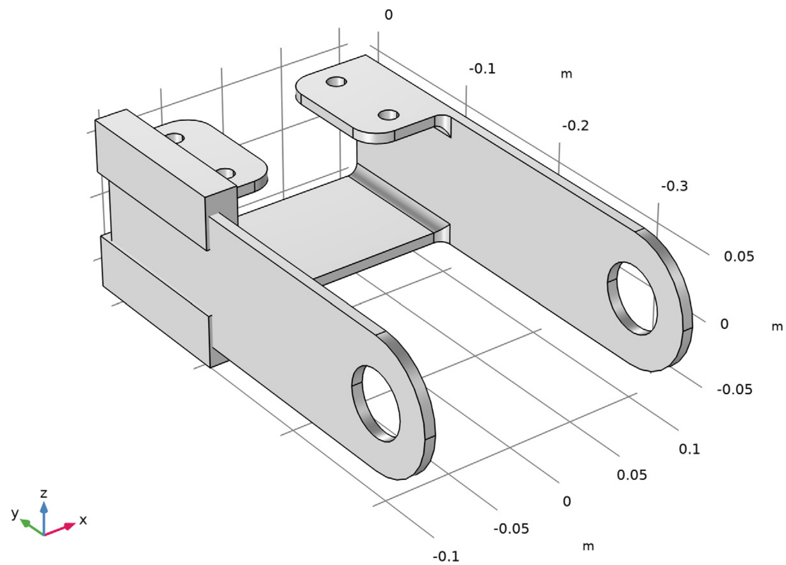
**2** In the **Settings** window for **Mirror**, locate the **Input** section.

**3** From the **Input objects** list, choose **Partition Block**.

**4** Select the **Keep input objects** check box.

**5** Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Partition Block**.

6 Click **Build Selected**.




*Mirror 2 (mir2)*

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Input** section.

3 From the **Input objects** list, choose **Partition Block**.

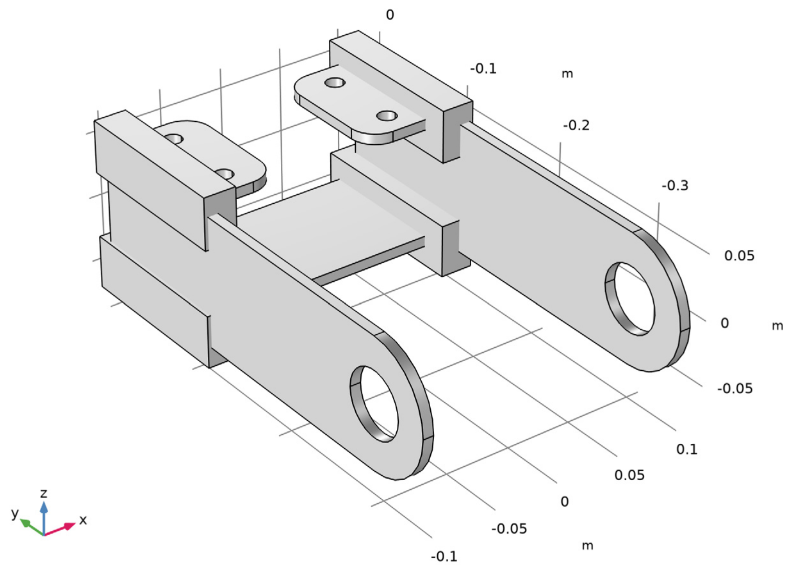
4 Select the **Keep input objects** check box.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type 1.

6 In the **z** text field, type 0.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Partition Block**.

**8** Click **Build Selected**.




*Partition Objects 1 (par1)*

**1** In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

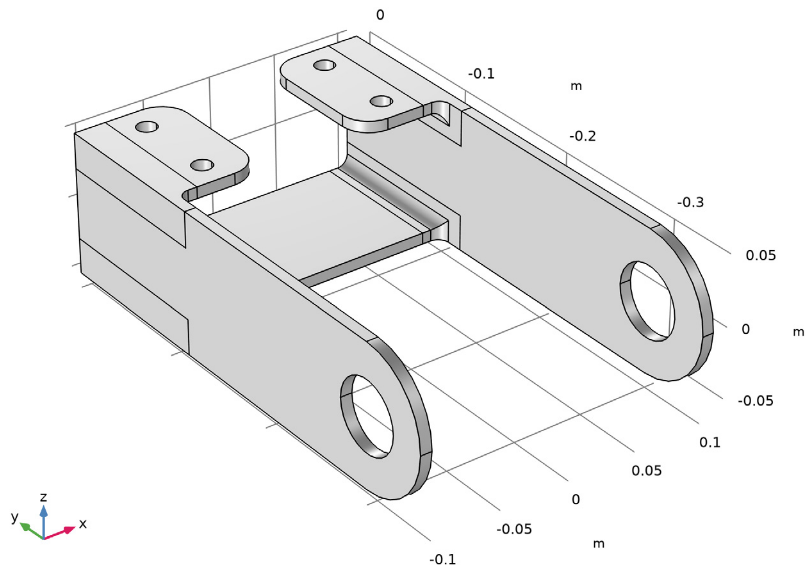
**2** Select the object **imp1** only.

**3** In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

**4** Find the **Tool objects** subsection. Click to select the **Activate Selection** toggle button.

**5** From the **Tool objects** list, choose **Partition Block**.

**6** Click **Build Selected**.




*Form Union (fin)*

**1** In the **Geometry** toolbar, click **Build All**.

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

### DEFINITIONS

Here you want to define an expression for the load applied to the load-carrying holes. Assume the load distribution to be defined by a trigonometric function.

*Analytic 1 (an1)*

**1** In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.

**2** In the **Settings** window for **Analytic**, type `load` in the **Function name** text field.

**3** Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

**4** In the **Arguments** text field, type `F, py, px`.

5 Locate the **Units** section. In the table, enter the following settings:

| Argument | Unit |
|---|---|
| F | Pa |
| py | m |
| px | m |

6 In the **Function** text field, type `Pa`.

*Bolt 1*

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

2 In the **Settings** window for **Explicit**, type `Bolt 1` in the **Label** text field.

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

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

5 Select Boundary 41 only.

6 Select the **Group by continuous tangent** check box.

7 Repeat the steps above to add three more explicit selections, with the following properties:

| Default node label | New node label | Select this boundary |
|---|---|---|
| Explicit 2 | Bolt 2 | 43 |
| Explicit 3 | Bolt 3 | 55 |
| Explicit 4 | Bolt 4 | 57 |

*Bolt Holes*

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

2 In the **Settings** window for **Union**, type `Bolt Holes` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the Color section. On Windows, select the eighth color in the first row of the palette (orange). On other platforms, choose Color 8 from the Color list.

5 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

6 In the **Add** dialog box, in the **Selections to add** list, choose **Bolt 1**, **Bolt 2**, **Bolt 3**, and **Bolt 4**.

7 Click **OK**.

Create selections for the two holes carrying the load.

*Left Pin Hole*

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Left Pin Hole` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 4 only.
5 Select the **Group by continuous tangent** check box.
6 Locate the Color section. On Windows, select the ninth color in the first row of the palette (green). On other platforms, choose Color 9 from the Color list.

*Right Pin Hole*

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Right Pin Hole` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 75 only.
5 Select the **Group by continuous tangent** check box.
6 Locate the Color section. On Windows, select the second color in the second row of the palette (a red color). On other platforms, choose Color 12 from the Color list.

*Pin Holes*

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Pin Holes` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose **Left Pin Hole** and **Right Pin Hole**.
6 Click **OK**.

Add a selection to be used during mesh generation.

*Bolt Hole Edges*

1 In the **Definitions** toolbar, click **Adjacent**.
2 In the **Settings** window for **Adjacent**, type `Bolt Hole Edges` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.
5 Locate the **Input Entities** section. Under **Input selections**, click + **Add**.

6 In the **Add** dialog box, select **Bolt Holes** in the **Input selections** list.

7 Click **OK**.

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

### MATERIALS

COMSOL Multiphysics is equipped with built-in material properties for a number of common materials. Here, choose structural steel. The material is automatically assigned to all domains.

### ADD MATERIAL

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

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

3 In the tree, select **Built-in>Structural steel**.

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

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

### SOLID MECHANICS (SOLID)

By default, the Solid Mechanics interface assumes that the participating material models are linear elastic, which is appropriate for this example. All that is left to do is to define the constraints and loads.

#### *Fixed Constraint 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

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

3 From the **Selection** list, choose **Bolt Holes**.

### GEOMETRY 1

Find out the coordinates of the hole center.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Measure**.

2 In the **Measure** window for **Measure**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 On the object **fin**, select Points 2 and 5 only.

## SOLID MECHANICS (SOLID)

Apply a boundary load to the bracket holes. The predefined boundary system is used for orienting the load in the normal direction. Note how boolean expressions like `Z>0` can be used to limit the part of the hole where the load is active. Also, the sign of the `X`-coordinate is used to flip where the load is applied.

### *Boundary Load 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pin Holes**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

| | |
|---|---|
| `0` | t1 |
| `0` | t2 |
| `load(-P0,Y-(-0.3[m]),Z)*(sign(X)*Z>0)` | n |

In order to make the expression more readable, you can convert the hole center coordinate into a parameter. It is possible to add new parameters on the fly from any input field that supports parameters. You can right-click in an empty text field to do that, but you can also convert an existing value into a parameter.

6 Mark the string `(-0.3[m])`, right-click, and select **Create Parameter**.

7 In the **Create Parameter** dialog, enter `YC` in the **Name** text field.

8 In the **Description** text field, enter `Y-coordinate of the hole center`, and click **OK**.

The value in the expression for the load is now automatically substituted by the new parameter name.

The parameter was added to the list in the **Parameters 1** node, and can be modified there if necessary. You can also select any parameter in a text field, right-click it, and immediately modify its value or description.

## MESH 1

Start by creating an edge mesh around the bolt holes to make sure that they are properly resolved.

### *Edge 1*

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

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

### *Distribution 1*

1 Right-click **Edge 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `8`.

Create a mesh which is swept through the thin flat parts, and then use a free tetrahedral mesh in the parts with a more complex geometry. Note that the transition between the two types of elements is automatic.

### *Swept 1*

1 In the **Mesh** toolbar, click **Swept**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 4–6, and 9 only.

5 Click to expand the **Source Faces** section. Select Boundaries 1, 33, 37, 50, and 72 only.

### *Size 1*

1 Right-click **Swept 1** and choose **Size**.

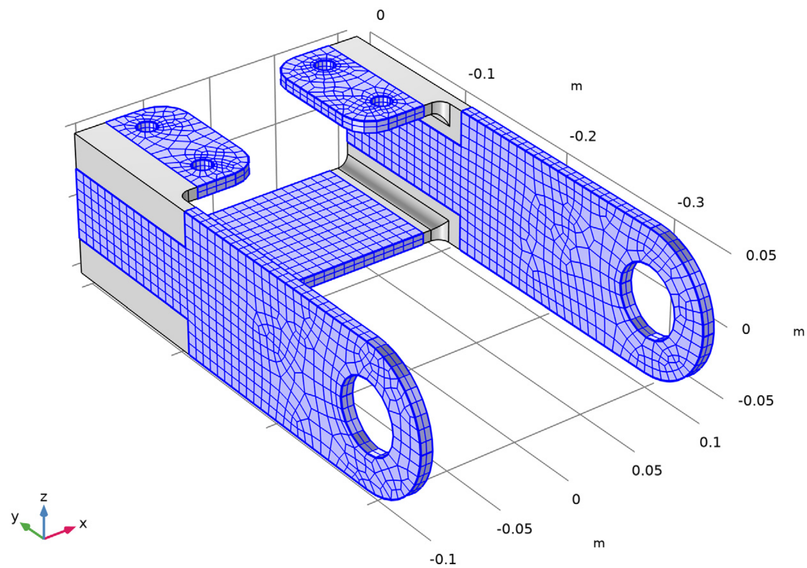
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `8[mm]`.

**6** Click **Build Selected**.




### Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

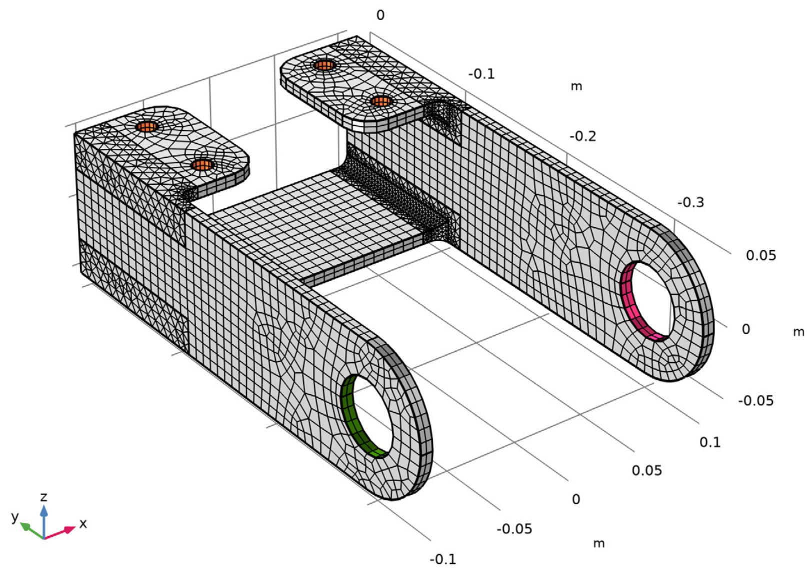
### Size 1

**1** Right-click **Free Tetrahedral 1** and choose **Size**.

**2** In the **Settings** window for **Size**, locate the **Element Size** section.

**3** From the **Predefined** list, choose **Finer**.

4 Click **Build All**.




The steps below show how to visualize the load distribution in the current geometry before computing the solution.

## STUDY 1

1 In the **Study** toolbar, click **Get Initial Value**.

Note that the Study node automatically defines a solver sequence for the simulation based on the selected physics (Solid Mechanics) and study type (Stationary).

The generation of initial values also creates any default plots. For this study type, it is a stress plot. You can now add an arrow plot of the loads from a list of predefined plots.

2 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)>Solid Mechanics>Applied Loads (solid)> Boundary Loads (solid)**.

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

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

## RESULTS

*Boundary Loads (solid)*

1 In the **Model Builder** window, under **Results** click **Boundary Loads (solid)**.
2 In the **Boundary Loads (solid)** toolbar, click **Plot**.

Now that you have verified the load distribution, solve the model.

## STUDY 1

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

The default plot shows the von Mises stress distribution, together with an exaggerated (automatically scaled) picture of the deformation.

## RESULTS

*Volume 1*

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Ambient Occlusion**.

Combine the stress plot with the load arrows.

*Boundary Load 1*

1 In the **Model Builder** window, expand the **Results>Boundary Loads (solid)** node.
2 Right-click **Boundary Load 1** and choose **Copy**.

*Boundary Load 1*

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Paste Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
3 From the **Arrow base** list, choose **Head**.
4 Select the **Scale factor** check box. In the associated text field, type `1E-8`.
5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.
6 Clear the **Arrow scale factor** check box.
7 Clear the **Color** check box.
8 Clear the **Color and data range** check box.

*Color Expression*

1 In the **Model Builder** window, expand the **Boundary Load 1** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

4 Click the **Show Grid** button in the **Graphics** toolbar.

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

Add a plot showing the displacement of the bracket. This is another one of the predefined plots.

6 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)>Solid Mechanics>Displacement (solid)**.

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

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

## RESULTS

*Total Displacement*

1 In the **Model Builder** window, under **Results** click **Displacement (solid)**.

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

*Volume 1*

1 In the **Model Builder** window, expand the **Total Displacement** node, then click **Volume 1**.

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

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

*Marker 1*

1 Right-click **Volume 1** and choose **Marker**.

2 In the **Settings** window for **Marker**, locate the **Display** section.

3 From the **Display** list, choose **Max**.

4 Locate the **Coloring and Style** section. From the **Background color** list, choose **From theme**.

5 Locate the **Text Format** section. In the **Display precision** text field, type `2`.

*Total Displacement*

1 In the **Model Builder** window, under **Results** click **Total Displacement**.
2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.
3 From the **Position** list, choose **Bottom**.
4 In the **Total Displacement** toolbar, click **Plot**.

Create another plot to display the principal stresses.

*Principal Stress*

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Principal Stress` in the **Label** text field.

*Principal Stress Volume 1*

1 In the **Principal Stress** toolbar, click **More Plots** and choose **Principal Stress Volume**.
2 In the **Settings** window for **Principal Stress Volume**, locate the **Positioning** section.
3 Find the **X grid points** subsection. In the **Points** text field, type `30`.
4 Find the **Y grid points** subsection. In the **Points** text field, type `60`.
5 Find the **Z grid points** subsection. In the **Points** text field, type `15`.
6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.
7 In the **Principal Stress** toolbar, click **Plot**.

A final check is to compute the total reaction force along the $x$, $y$, and $z$ directions. Use a surface integration over the constrained boundaries.

*Evaluation Group: Reactions*

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Reactions` in the **Label** text field.

*Bolt 1*

1 Right-click **Evaluation Group: Reactions** and choose **Integration>Surface Integration**.
2 In the **Settings** window for **Surface Integration**, type `Bolt 1` in the **Label** text field.
3 Locate the **Selection** section. From the **Selection** list, choose **Bolt 1**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Reactions>Reaction force (spatial frame) - N>All expressions in this group**.

**5** Locate the **Expressions** section. In the table, enter the following settings:

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `solid.RFx` | `N` | `Bolt 1, X` |
| `solid.RFy` | `N` | `Bolt 1, Y` |
| `solid.RFz` | `N` | `Bolt 1, Z` |

*Bolt 2*

**1** Right-click **Bolt 1** and choose **Duplicate**.

**2** In the **Settings** window for **Surface Integration**, type `Bolt 2` in the **Label** text field.

**3** Locate the **Selection** section. From the **Selection** list, choose **Bolt 2**.

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

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `solid.RFx` | `N` | `Bolt 2, X` |
| `solid.RFy` | `N` | `Bolt 2, Y` |
| `solid.RFz` | `N` | `Bolt 2, Z` |

*Bolt 3*

**1** Right-click **Bolt 2** and choose **Duplicate**.

**2** In the **Settings** window for **Surface Integration**, type `Bolt 3` in the **Label** text field.

**3** Locate the **Selection** section. From the **Selection** list, choose **Bolt 3**.

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

| **Expression** | **Unit** | **Description** |
|---|---|---|
| `solid.RFx` | `N` | `Bolt 3, X` |
| `solid.RFy` | `N` | `Bolt 3, Y` |
| `solid.RFz` | `N` | `Bolt 3, Z` |

*Bolt 4*

**1** Right-click **Bolt 3** and choose **Duplicate**.

**2** In the **Settings** window for **Surface Integration**, type `Bolt 4` in the **Label** text field.

**3** Locate the **Selection** section. From the **Selection** list, choose **Bolt 4**.

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

| Expression | Unit | Description |
|---|---|---|
| `solid.RFx` | `N` | `Bolt 4, X` |
| `solid.RFy` | `N` | `Bolt 4, Y` |
| `solid.RFz` | `N` | `Bolt 4, Z` |

**5** In the **Evaluation Group: Reactions** toolbar, click = **Evaluate**.