

# Optimization of an Imported Bracket Geometry — with Measuring Dimensions

## *Introduction*

---

In some application fields, there is a strong focus on weight reduction. For example, this is the case in the automotive industry, where every gram has a distinct price tag.

The bracket in this tutorial is used for mounting a heavy component on a vibrating foundation. It is thus important to keep the natural frequency well above the excitation frequency to avoid resonances. The bracket is also subjected to shock loads, which can be treated as a static acceleration load. This gives an optimization problem, where results from two different study types must be considered simultaneously. In this example, the weight of a mounting bracket is reduced, given an upper bound on the stresses and a lower bound on the first natural frequency, by changing the size and location of three holes on the bracket.

In this tutorial, you will import the bracket geometry from a CAD file and use the Offset Faces and Transform Faces operations to reparameterize the already existing holes in the bracket to move and resize them.

An integral part of setting up the optimization is defining the geometrical, or design, constraints, which in this case will ensure that the holes will not come close to each other or the edges of the bracket. You can achieve this by adding dimension features to the geometry sequence. These measure the modified geometry and create parameters with the values. Using the parameters, you can then formulate conditions in parameter check features to abort the building of the geometry in case a measured value is outside the acceptable interval. When a parameter check is triggered during the optimization, the solver will not solve the forward problem; instead, it will continue with the next iteration.

The tutorial [Optimization of an Imported Bracket Geometry — with Parameter Expressions](#) demonstrates an alternative way of setting up the design constraints for the optimization by defining parameter expressions that you can use directly in the constraints section of the optimization study settings.

---

**Note:** This application requires the Optimization Module, the Structural Mechanics Module, and the Design Module.

---

## *Model Definition*

---

The original bracket together with a sketched mounted component are shown in [Figure 1](#). The bracket is made of steel.

The component, which can be considered as rigid when compared with the bracket, has its center of gravity at the center of the circular cutout in the bracket. The mass is 4.4 kg, the moment of inertia around its longitudinal axis is  $7.1 \cdot 10^{-4}$  kg·m<sup>2</sup>, and the moment of inertia around the two transverse axes is  $9.3 \cdot 10^{-4}$  kg·m<sup>2</sup>.

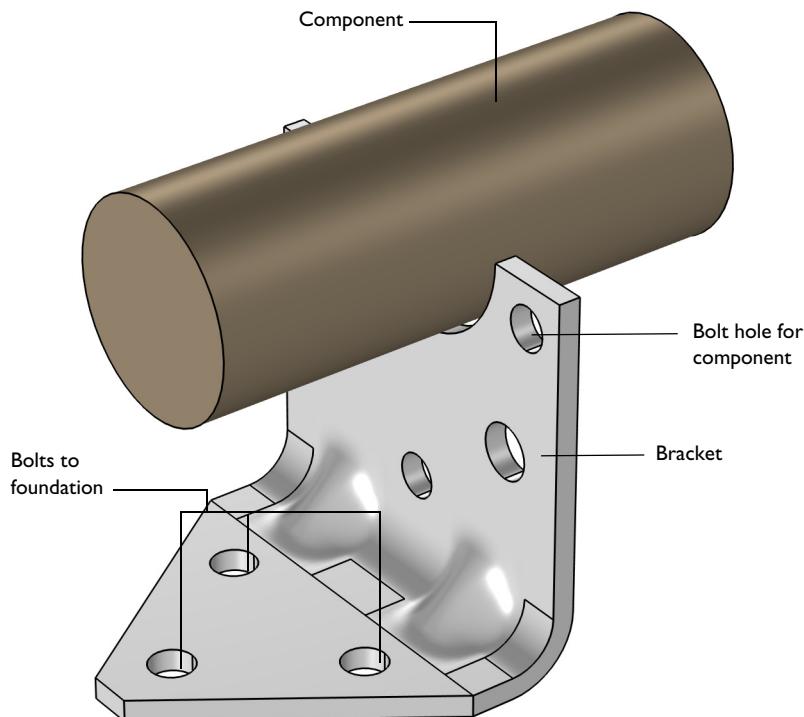


Figure 1: Bracket supporting a heavy component.

We can reduce the weight of the bracket by drilling holes in the bracket's vertical surface and optimizing the holes' size and placement. This is done by adding the Offset Faces and Transform Faces geometry operations to reparameterize the holes on the imported bracket geometry.

#### OPTIMIZATION PARAMETERS

Five geometrical parameters are used in the optimization. They are summarized in [Table 1](#) and shown in [Figure 2](#).

TABLE I: GEOMETRICAL PARAMETERS.

Parameter	Description	Lower limit (mm)	Upper limit (mm)
RC	Radius of the central hole	3	15
ZCO	Vertical distance from the bend to the edge of the central hole	1	23
RO	Radius of the outer hole	3	15
ZOO	Vertical distance from the bend to the edge of the outer hole	8	30
YOO	Horizontal distance from the edge of the bracket to outer hole	3	29

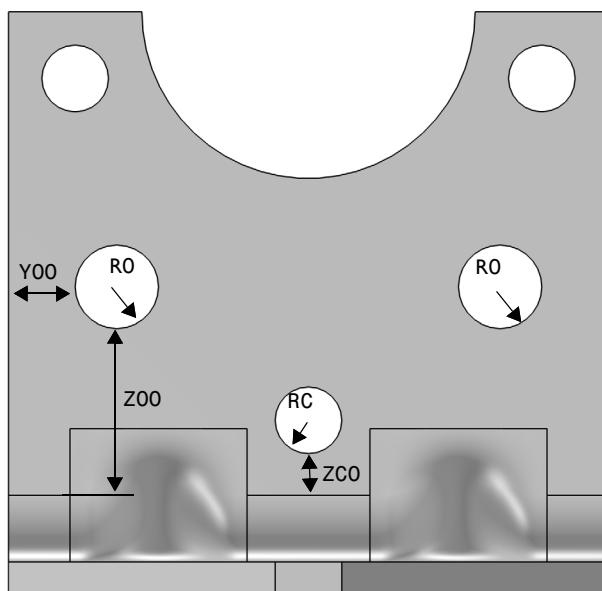


Figure 2: Geometrical parameters that are optimized in the study.

There are two possible paths to take when considering how to set up the control parameters for the optimization. Here we optimize the actual radii and positions of the

holes. An alternative would be to define the control parameters that are the displacements relative to the original geometry that is imported from the CAD file

You will measure all the original geometrical parameters that will be optimized in the study, by projecting the geometry onto a plane and using Dimensions and Constraints operations to create new parameters.

In the geometry sequence, one Transform Faces node is added to change both the position and the radius, defined by a scaling factor, of the central hole. By using the Transform Faces operation, we can move and change the size of the holes by applying a linear transformation (displacement, rotation, and isotropic scaling) to the faces of the holes. For the outer holes, we add an Offset Faces node to change the radius of the holes and we add two Transform Faces operations to displace the holes. Using the Offset Faces operation, we change the size of the holes on the bracket by offsetting the hole faces in the normal direction.

## CONSTRAINTS

Constraints can be divided into physical constraints and geometrical constraints.

The physical constraints are defined as follows:

- The lowest natural frequency must be at least 60 Hz.
- When exposed to a peak acceleration of  $4g$  in all three global directions simultaneously, the effective stress is not allowed to exceed 80 MPa anywhere. This criterion is nondifferentiable because the location of the peak stress can jump from one place to another. A gradient-free optimization algorithm must thus be used.

Geometrical constraints have to ensure that there must be at least 3 mm of material between two holes or between a hole and an edge. This criterion is enforced through limits on the control parameters and as constraints.

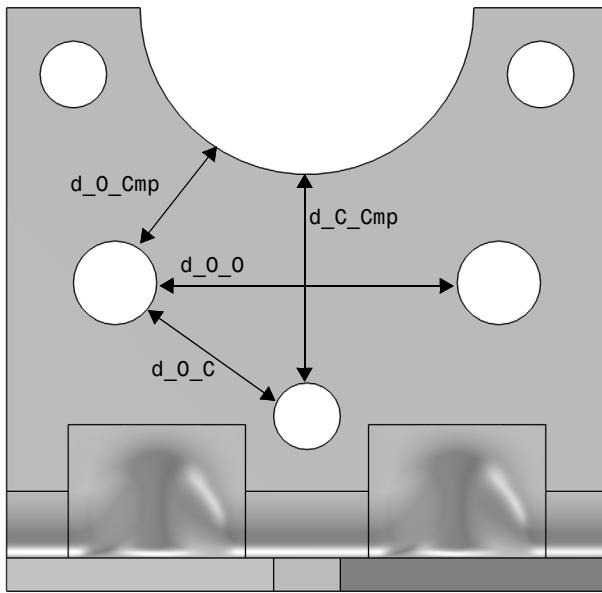
The COBYLA solver uses sampling in the control variable space to approximate the objective function, the constraints, and the control variable bounds. Individual samples may be computed outside the bounds and in violation of the constraints. Therefore, it is important to parameterize the geometry in such a way that it is robust with respect to (small) constraints and bound violations.

Bounds and linear constraints are generally satisfied to high precision at the optimum point returned by the solver, but nonlinear constraints are often slightly violated. The reason is that the solver tends to converge from the outside of the feasible domain and terminates before the constraints are completely satisfied. Tightening the solver tolerances

will decrease the constraint violation but is often not worth the computational effort; it is better to specify constraints with a safety margin.

### GEOMETRICAL CONSTRAINTS

The geometrical constraints are shown in [Figure 3](#) and summarized in [Table 2](#).



*Figure 3: Geometrical constraints.*

As shown in [Figure 3](#), geometrical constraints are set up so that spaces between different geometrical entities are bounded in specific ranges. That is important because we do not want to have very fragile parts of the geometry that would break when loaded, and we want to preserve the original shape of the geometry. Here, we consider four different geometrical constraints, which are summarized in [Table 2](#).

TABLE 2: GEOMETRICAL CONSTRAINTS.

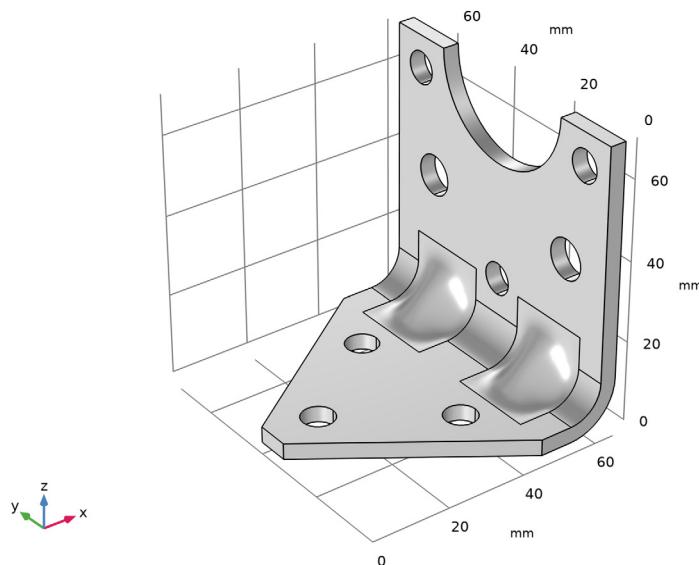
Parameter	Description	Limit (mm)
d_O_Cmp	Distance between the edge of outer hole and component	3
d_O_O	Distance between the edges of two outer holes	3
d_O_C	Distance between the edges of outer and central holes	3
d_C_Cmp	Distance between the edges of central hole and component	3

In this tutorial, the geometrical constraints are enforced by Parameter Check nodes in the geometry sequence. A Parameter Check node is useful to ensure that a parameter value stays within a certain range. The check gives an error if the condition that you provide is false. If so, the geometry with this set of parameters is skipped in the optimization study. In this case, the checked parameters are created by dimension nodes that measure the modified geometry.

### Results and Discussion

---

The initial geometry used in the optimization is shown in [Figure 4](#). Three rather small holes have been introduced.



*Figure 4: Initial geometry.*

The optimal values of the geometrical parameters are shown in [Table 3](#).

TABLE 3: OPTIMAL VALUES.

Parameter	Optimal value (mm)	Lower limit (mm)	Upper limit (mm)
RC	6.08	3	15
ZCO	4.99	1	23
RO	9.94	3	15

TABLE 3: OPTIMAL VALUES.

Parameter	Optimal value (mm)	Lower limit (mm)	Upper limit (mm)
Y00	3.08	3	29
Z00	19.72	8	30

The weight of the optimized bracket is about 187 g, a reduction of 17 g from the original 204.10 g. The stresses from the shock load on the optimized geometry are shown in Figure 5.

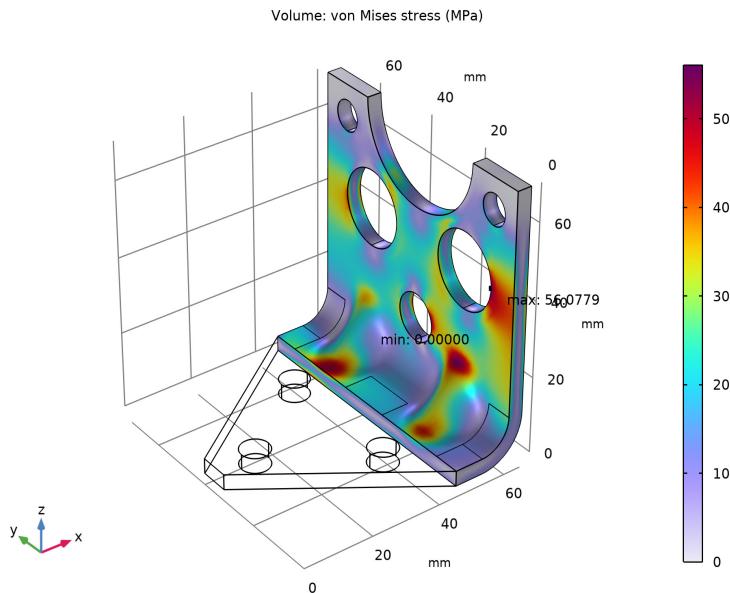


Figure 5: Stresses at peak load in the optimized design.

The optimal solution gives three fairly large holes.

There are several possible arrangements of the holes that give the same weight reduction within a small tolerance. It is therefore possible that the design variables are not always the same at convergence.

### Notes About the COMSOL Implementation

The bracket geometry is imported from a CAD file saved in STEP format.

The component mounted on the bracket is not modeled in detail. It is replaced by a Rigid Connector having the equivalent inertial properties.

---

**Application Library path:** Design\_Module/Tutorial\_Examples/  
bracket\_import\_optimization\_dimensions

---

## *Modeling Instructions*

---

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

### **NEW**

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

### **MODEL WIZARD**

- 1 In the **Model Wizard** window, click  **3D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Eigenfrequency**.
- 6 Click  **Done**.

### **GLOBAL DEFINITIONS**

#### *Parameters I*

In the parameters table, you will load the parameters needed to define the physics and initial values of optimization parameters used in the optimization study.

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `bracket_import_optimization_dimensions_parameters.txt`.
- 5 In the **Home** toolbar, click  **Parameter Case**.

Creating a parameter case for the original design parameters will come in handy when comparing the optimized and unmodified bracket geometry.

- 6 In the **Settings** window for **Case**, type **Original Values** in the **Label** text field.

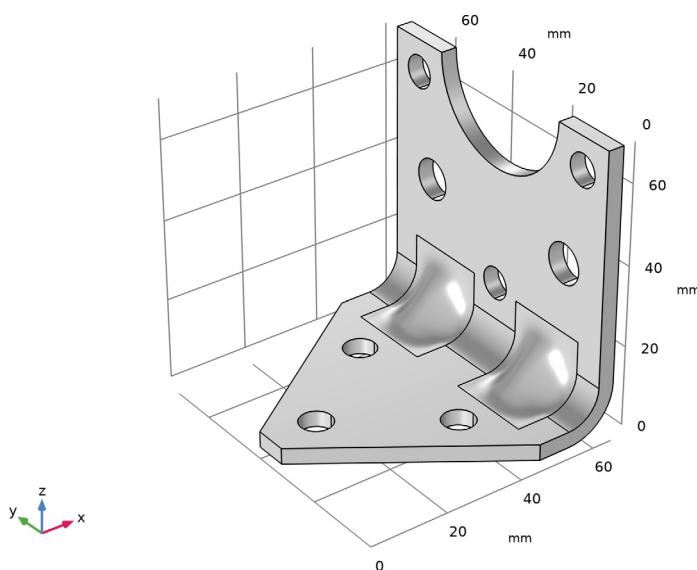
## GEOMETRY I

The bracket geometry for this tutorial has been saved in the STEP format. Make sure that the CAD Import Module geometry kernel is used.

- 1 In the **Model Builder** window, under **Component 1** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
- 3 From the **Geometry representation** list, choose **CAD kernel**.

### *Import 1*

- 1 In the **Home** toolbar, click  **Import**.
  - 2 In the **Settings** window for **Import**, locate the **Import** section.
  - 3 Click  **Browse**.
  - 4 Browse to the model's Application Libraries folder and double-click the file `bracket_import_optimization_geom.step`.
  - 5 From the **Length unit** list, choose **From CAD document**.
- With this option, the length unit of the geometry is automatically set to the unit found in the imported file, in this case, mm. Otherwise, you can manually specify the length unit in the **Settings** window for the **Geometry** node.
- 6 Click  **Import**.



### *Original Geometry*

In the next section, you will add measuring dimensions to get all the necessary measurements of the imported geometry. You will first project the geometry onto a plane and measure the radius of the holes, distances between the holes and the edges of the geometry, which will define your optimization parameters, as well as all other dimensions of the geometry that will be useful for future analysis.

- 1 In the **Geometry** toolbar, click  **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, type **Original Geometry** in the **Label** text field.
- 3 Locate the **Plane Definition** section. From the **Plane** list, choose **yz-plane**.

### *Original Geometry>Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

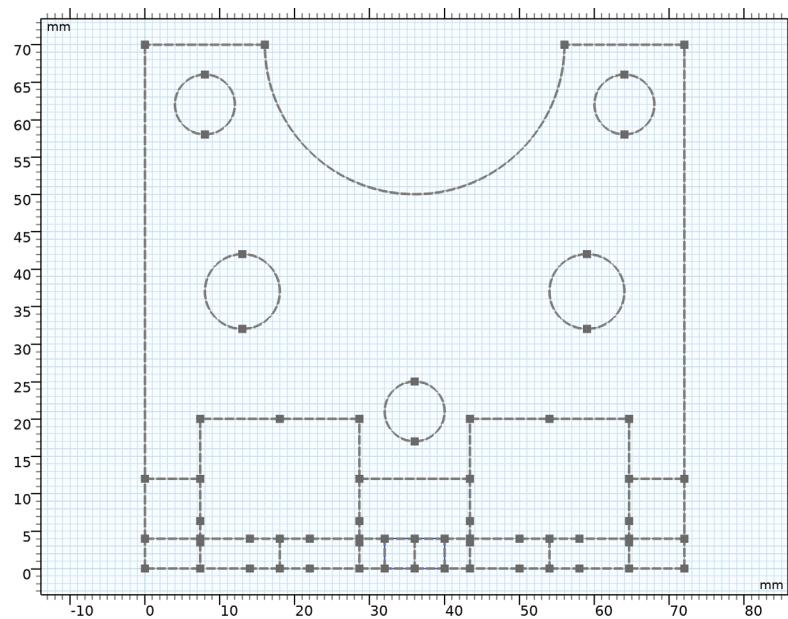
### *Original Geometry>Projection /*

- 1 In the **Work Plane** toolbar, click  **Projection**.
- 2 In the **Settings** window for **Projection**, locate the **Projection** section.
- 3 From the **Projection type** list, choose **Edges and vertices**.

Since you are only interested in the values of the measurements and you will not create geometry on this work plane, this projection can be set as construction geometry.

- 4 Locate the **Assigned Attributes** section. From the **Construction geometry** list, choose **On**.

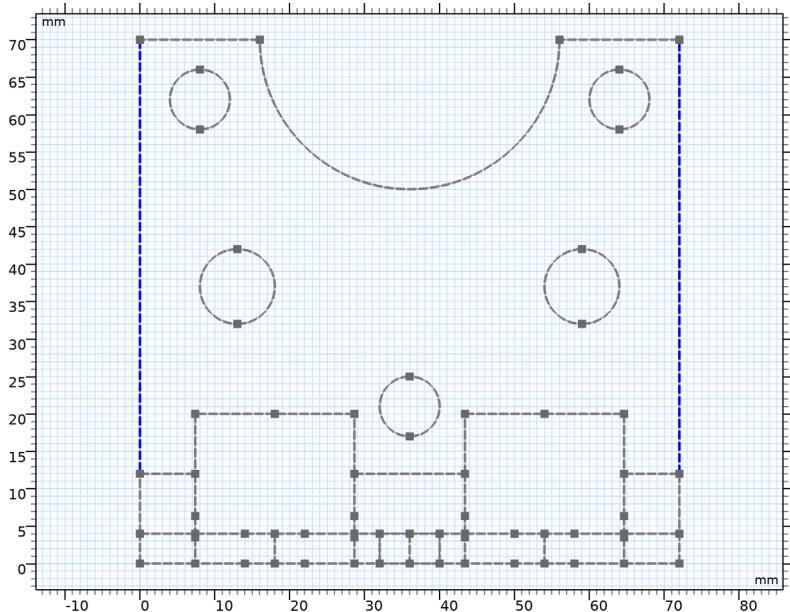
5 Click  **Build Selected.**



*Original Geometry>Distance 1*

I In the **Sketch** toolbar, click  **Distance**.

- 2** Select the two outer boundaries of the bracket, following the figure below, and apply **Distance** by clicking on the **Graphics** window.



Dimensions are created by default as constraining dimensions that control the geometry. The dimension is displayed in magenta in the Graphics window, indicating that the geometry is overdefined. This is expected as it cannot change the projection. Next, you will change this dimension into a measuring dimension that does not drive the measurements of the entities it is applied to, instead, it is updated according to the other features in the geometry sequence.

- 3** In the **Settings** window for **Distance**, locate the **Dimension Value** section. Click to clear the associated **Constrain Distance** toggle button (lock symbol).

- 4** Select the **Create measuring parameter** check box.

This creates a measuring parameter you can use in other geometry and non-geometry features.

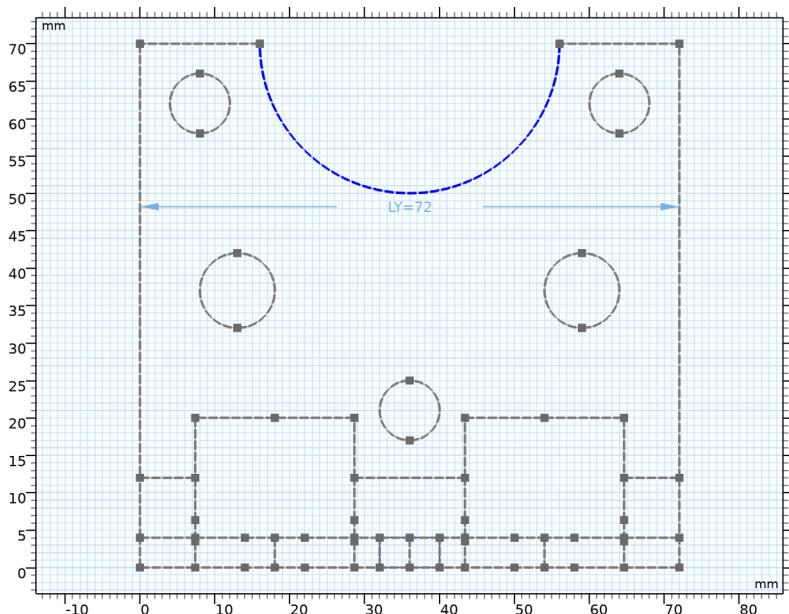
- 5** In the **Parameter name** text field, type LY.

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

*Original Geometry>Radius 1*

- I** In the **Sketch** toolbar, click **Radius**.

- 2** Select the edge of the mounting component hole, following the figure below, and apply **Radius** by clicking in the **Graphics** window.

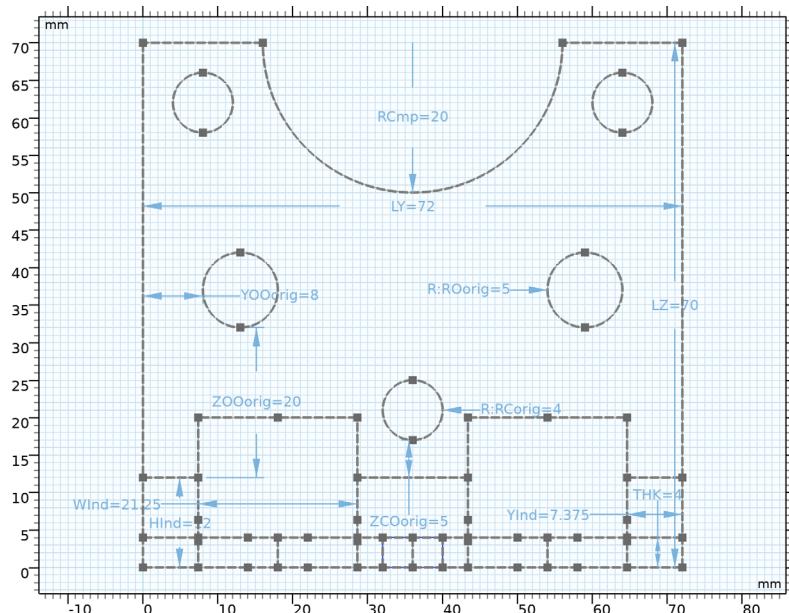


- 3** In the **Settings** window for **Radius**, locate the **Dimension Value** section. Click to clear the associated **Constrain Radius** toggle button (lock symbol).
- 4** Select the **Create measuring parameter** check box.
- 5** In the **Parameter name** text field, type **RCmp**.
- 6** Click **Build Selected**.

*Original Geometry>Distance 2-9, Radius 2-3*

I Proceed to create the measuring parameters listed in the table and shown in the image below by defining measuring dimensions according to the previous sections.

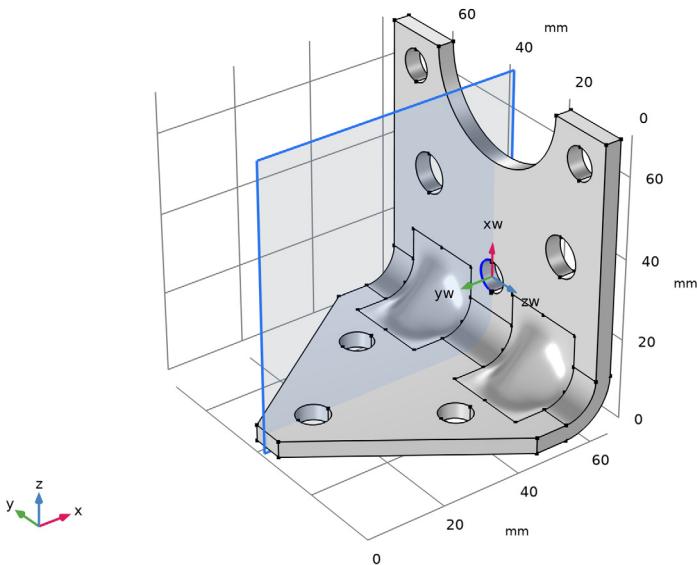
Parameter name	Description
ROorig	Original radius of outer holes
RCorig	Original radius of central hole
LZ	Bracket height
ZCOorig	Original distance from bend to bottom of central hole
THK	Plate thickness
YInd	Y position of the indentation
WInd	Width of indentation
HInd	Bend height
YOOrig	Original distance from edge to outer hole
ZOOrig	Original distance from bend to bottom of outer hole



### *Coordinate System - Central Hole*

Next, set up work planes for the two outer and the central hole. These define the coordinate systems you will use to change the hole size and position using the Transform Faces operation.

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, type Coordinate System - Central Hole in the **Label** text field.
- 3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Circle perpendicular**.
- 4 On the object **imp1**, select Edge 91 only.

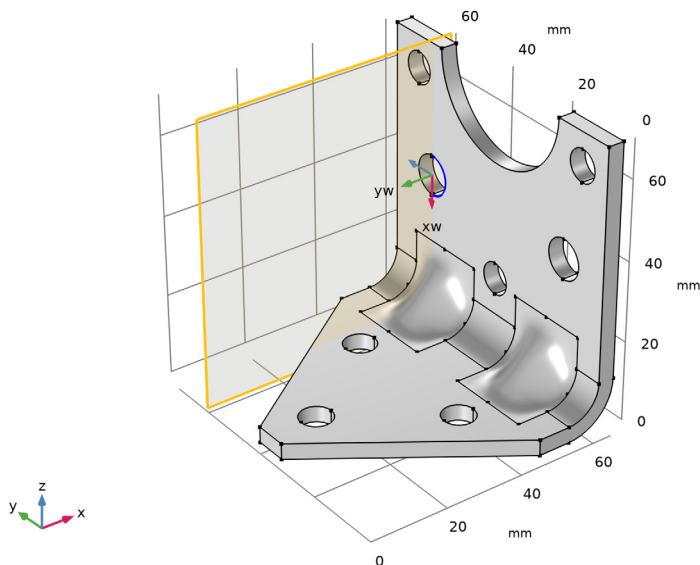


- 5 Click **Build Selected**.

### *Coordinate System - Outer Hole 1*

- 1 In the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, type Coordinate System - Outer Hole 1 in the **Label** text field.
- 3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Circle perpendicular**.

- 4 On the object **imp1**, select Edge 99 only.



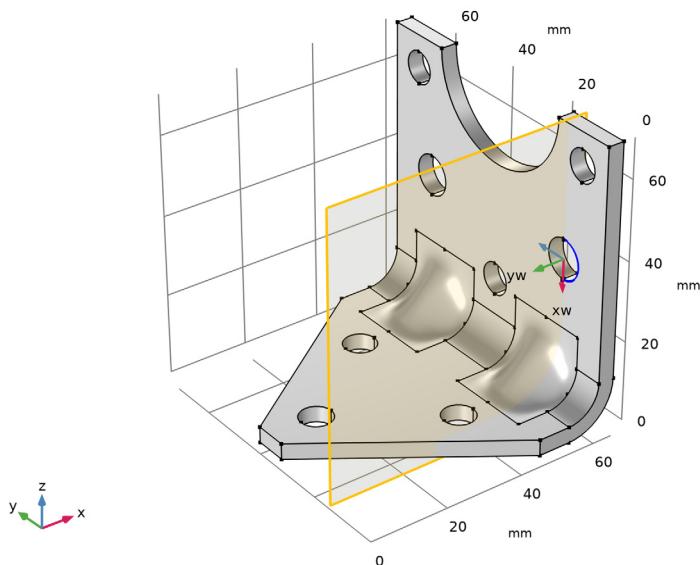
- 5 Select the **Reverse normal direction** check box.

- 6 Click **Build Selected**.

*Coordinate System - Outer Hole 2*

- 1 In the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, type **Coordinate System - Outer Hole 2** in the **Label** text field.
- 3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Circle perpendicular**.

- 4 On the object **imp1**, select Edge 81 only.



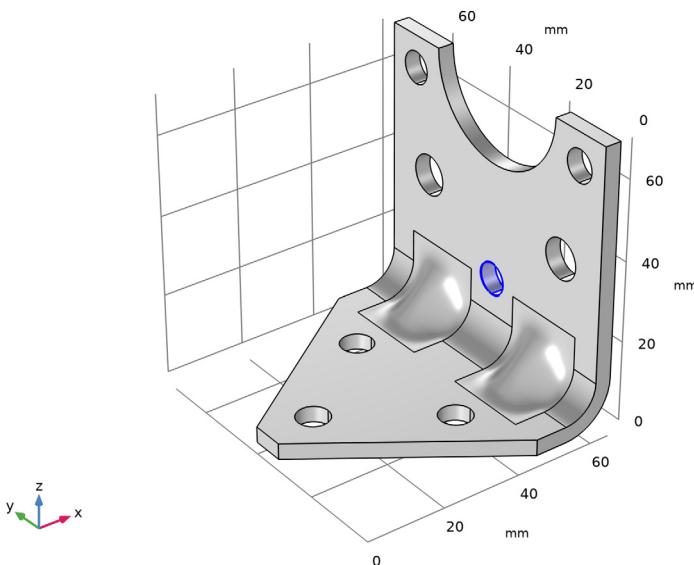
- 5 Click **Build Selected**.

#### *Z Displacement and Scaling - Central Hole*

For modifying the central hole, you will add a Transform Faces operation to the geometry sequence. Using this feature, you can both displace the hole and change its radius by scaling it.

- 1 In the **Geometry** toolbar, click **Editing** and choose **Transform Faces**.
- 2 In the **Settings** window for **Transform Faces**, type **Z Displacement and Scaling - Central Hole** in the **Label** text field.

- 3 On the object **imp1**, select Boundaries 31 and 32 only.



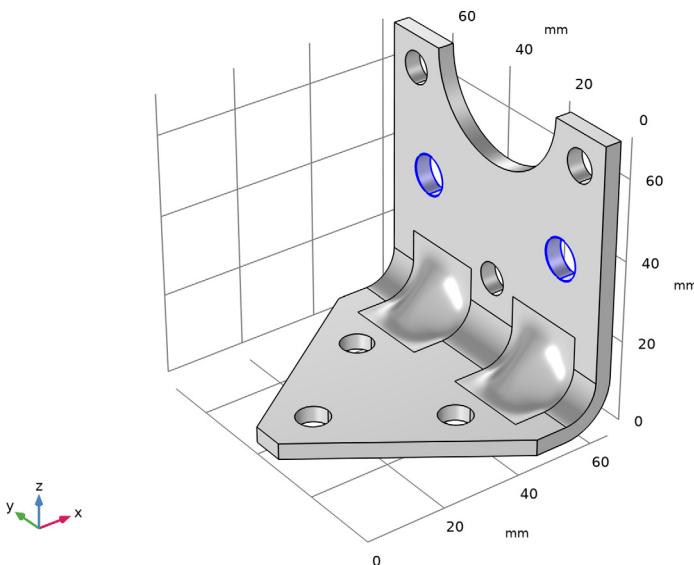
- 4 Locate the **Coordinate System** section. From the **Work plane** list, choose **Coordinate System - Central Hole**.
- 5 Locate the **Scaling** section. In the **Factor** text field, type **RC/RCorig**.
- 6 Locate the **Displacement** section. In the **xw** text field, type **ZCO+RC-RCorig-ZCOorig**.
- 7 Click **Build Selected**.

#### *Offset - Outer Holes*

For the two outer holes, use the Offset Faces operation to change the radius. Note that only one Offset Faces operation is needed since the holes have the same radius.

- 1 In the **Geometry** toolbar, click **Editing** and choose **Offset Faces**.
- 2 In the **Settings** window for **Offset Faces**, type **Offset - Outer Holes** in the **Label** text field.

- 3 On the object **tf1**, select Boundaries 28, 29, 34, and 35 only.



4 Locate the **Offset** section. In the **Distance** text field, type **R0-R0orig**.

5 Select the **Reverse side** check box.

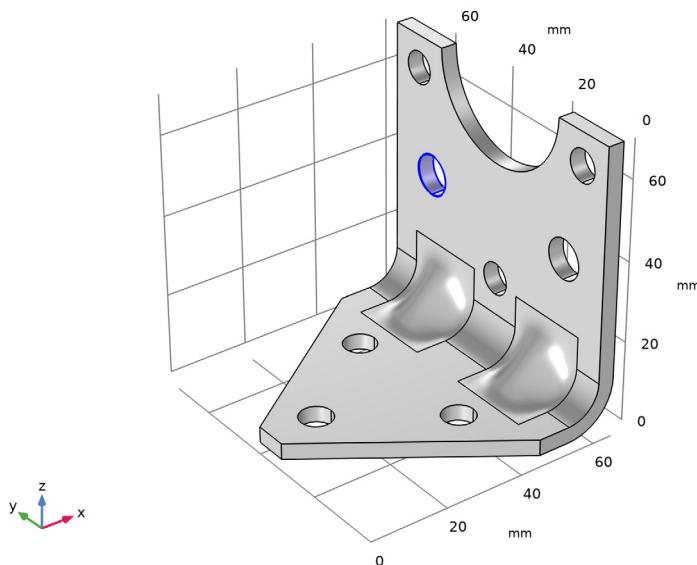
6 Click **Build Selected**.

#### *Displacement - Outer Hole 1*

Next, you can add the Transform Faces operations to displace the outer holes.

- 1 In the **Geometry** toolbar, click **Editing** and choose **Transform Faces**.
- 2 In the **Settings** window for **Transform Faces**, type **Displacement - Outer Hole 1** in the **Label** text field.

- 3 On the object **off1**, select Boundaries 34 and 35 only.



4 Locate the **Coordinate System** section. From the **Work plane** list, choose **Coordinate System - Outer Hole 1**.

5 Locate the **Displacement** section. In the **xw** text field, type  
- (Z00-Z00orig+R0-R0orig).

6 In the **zw** text field, type Y00-Y00orig+R0-R0orig.

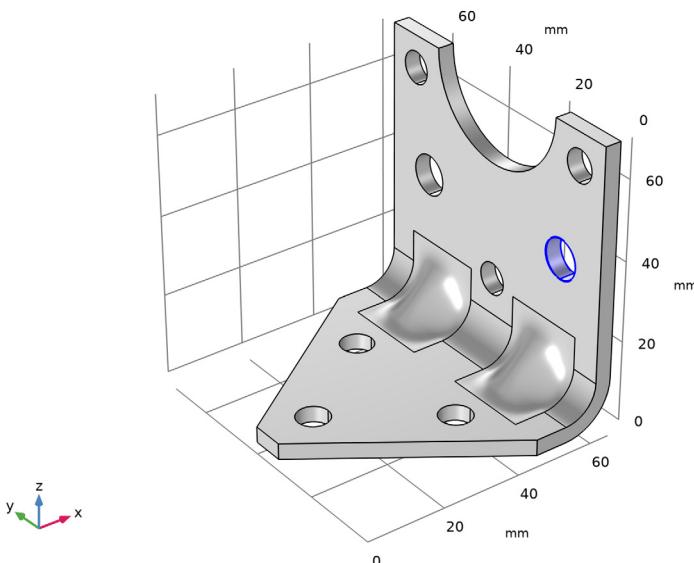
7 Click **Build Selected**.

*Displacement - Outer Hole 2*

1 In the **Geometry** toolbar, click **Editing** and choose **Transform Faces**.

2 In the **Settings** window for **Transform Faces**, type Displacement - Outer Hole 2 in the **Label** text field.

- 3 On the object **tf2**, select Boundaries 28 and 29 only.



4 Locate the **Coordinate System** section. From the **Work plane** list, choose **Coordinate System - Outer Hole 2**.

5 Locate the **Displacement** section. In the **xw** text field, type  
- (Z00-Z00orig+R0-R0orig).

6 In the **zw** text field, type Y00-Y00orig+R0-R0orig.

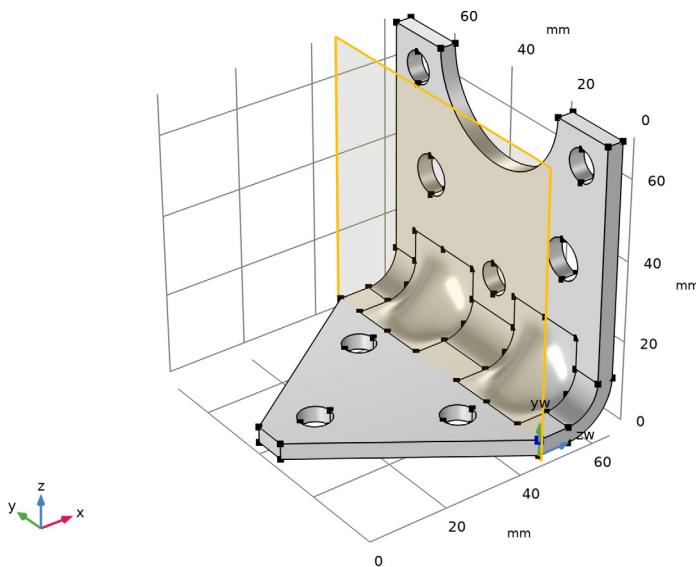
7 Click **Build Selected**.

#### Work Plane 5

You will define a new work plane that you will use for partitioning the bracket into two domains since the evaluation of the maximum stress is needed only for the upper part of the bracket.

- 1 In the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
- 3 From the **Plane** list, choose **yz-plane**.
- 4 From the **Offset type** list, choose **Through vertex**.

- 5 On the object **tf3**, select Point 18 only.



- 6 Click **Build Selected**.

#### *Partition Objects 1*

- 1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
- 2 Select the object **tf3** only.
- 3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
- 4 From the **Partition with** list, choose **Work plane**.
- 5 Click **Build Selected**.

#### *Geometrical Constraints*

Next, add a work plane that you will use for projecting the modified geometry. Here you will set up measuring dimensions that define the parameters for the geometrical constraints used for the optimization.

- 1 In the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, type **Geometrical Constraints** in the **Label** text field.
- 3 Locate the **Plane Definition** section. From the **Plane** list, choose **yz-plane**.

*Geometrical Constraints>Plane Geometry*

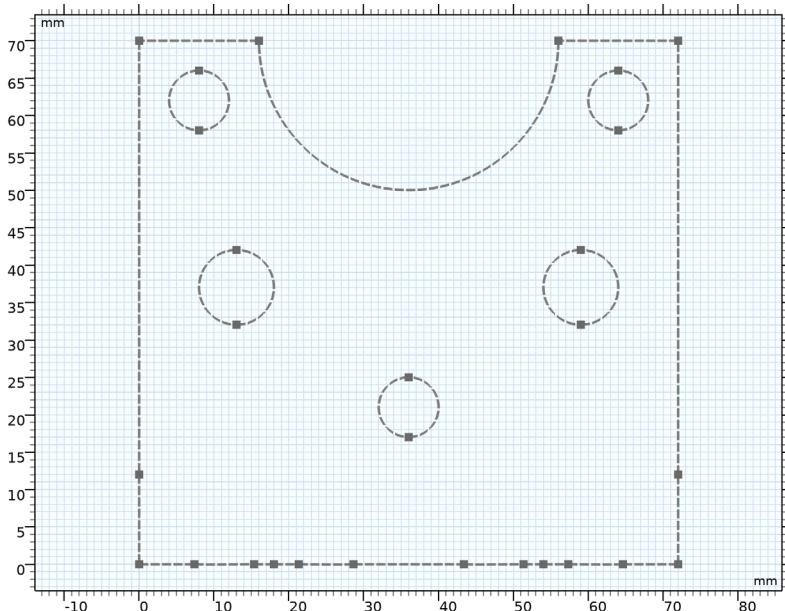
- 1 In the **Model Builder** window, click **Plane Geometry**.

For a cleaner look in the Graphics window, disable the in-plane visualization of the outline of the 3D geometry.

- 2 In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.
- 3 Find the **In-plane visualization of 3D geometry** subsection. Clear the **Coincident entities (blue)** check box.
- 4 Clear the **Intersection (green)** check box.
- 5 Clear the **Projection (gray)** check box.

*Geometrical Constraints>Projection 1*

- 1 In the **Work Plane** toolbar, click  **Projection**.
- 2 In the **Settings** window for **Projection**, locate the **Assigned Attributes** section.
- 3 From the **Construction geometry** list, choose **On**.
- 4 Click  **Build Selected**.



### *Geometrical Constraints>Point 1*

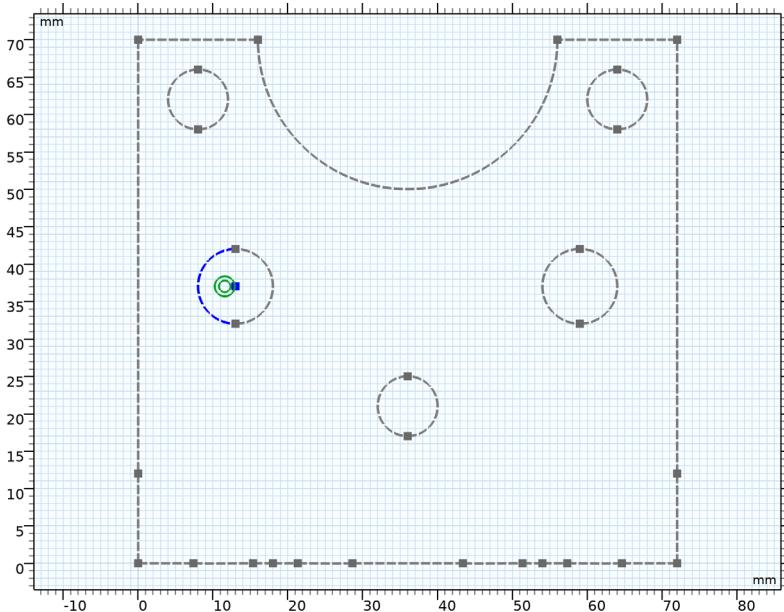
Next, create points that you will constrain to the center of the projected circles. Later, you will use these points for setting up the measuring dimensions for the geometrical constraints.

- 1 In the **Sketch** toolbar, click  **Point**.
- 2 Apply the **Point** by clicking in the **Graphics** window near the left outer hole.
- 3 Locate the **Assigned Attributes** section. Select the **Construction geometry** check box.

### *Geometrical Constraints>Concentric 1*

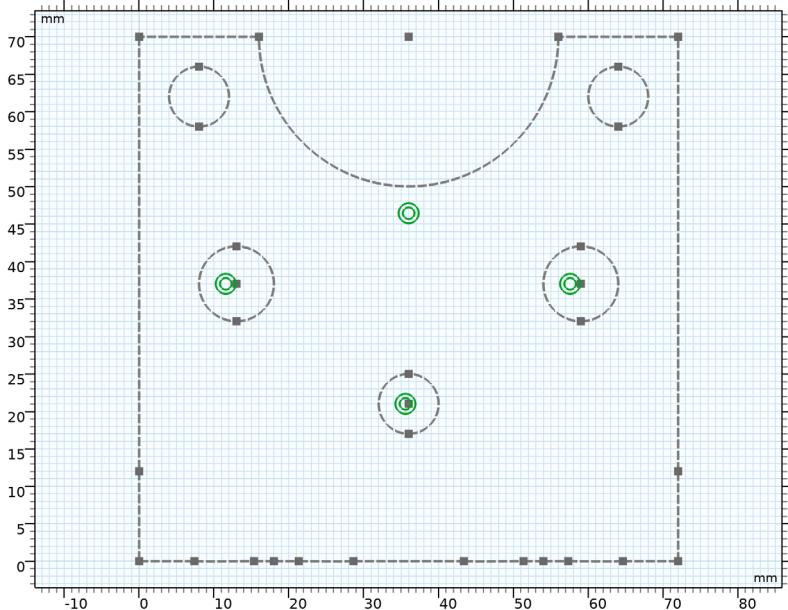
The Concentric constraint ensures that the point and the circle always stay concentric as the geometry is updated during the optimization study.

- 1 In the **Sketch** toolbar, click  **Concentric**.
- 2 Select one edge of the left outer hole and Point 1 and apply the **Concentric** constraint by clicking in the **Graphics** window.



#### *Geometrical Constraints>Point 2-4, Concentric 2-4*

Proceed to create three more points and apply the **Concentric** constraints. When done, you should have points at the center of the circles that are the projections of the optimized holes and at the center of the half circle at the top, as shown in the figure below.



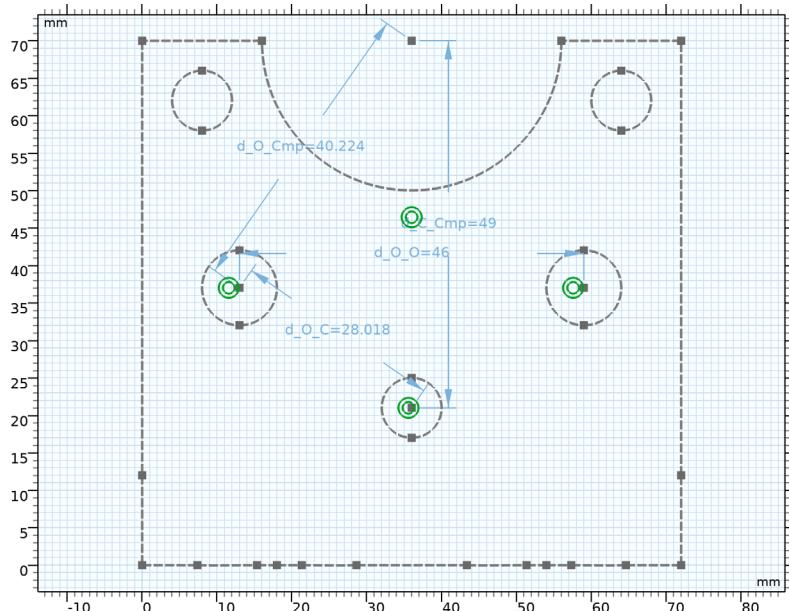
#### *Geometrical Constraints>Distance 1*

- 1 In the **Sketch** toolbar, click **Distance**.
- 2 Select the point in the left outer hole and the mounting component and apply **Distance** by clicking in the **Graphics** window.
- 3 In the **Settings** window for **Distance**, locate the **Dimension Value** section. Click to clear the associated **Constrain Distance** toggle button (lock symbol).
- 4 Select the **Create measuring parameter** check box.
- 5 In the **Parameter name** text field, type **d\_O\_Cmp**.

### Geometrical Constraints>Distance 2-4

- I Add three more **Distance** constraints by following the steps in the section above to create the following dimension parameters.

Parameter name	Description
d_O_O	Distance between outer holes
d_O_C	Distance between outer and central holes
d_C_Cmp	Distance between central hole and component



### Parameter Check I

To ensure that the modified bracket geometry fulfills the geometrical constraints, add **Parameter Check** features where you can enter conditions using the previously defined dimension parameters.

- I In the **Model Builder** window, right-click **Geometry 1** and choose **Programming> Parameter Check**.
- 2 In the **Settings** window for **Parameter Check**, locate the **Parameter Check** section.
- 3 In the **Condition** text field, type  $d\_O\_Cmp < 3[\text{mm}] + R_O + R_{Cmp}$ .

- 4 In the **Message** text field, type Distance between outer hole and component smaller than 3 mm.

5 Click  **Build Selected**.

#### *Parameter Check 2*

- 1 In the **Geometry** toolbar, click  **Programming** and choose **Parameter Check**.
- 2 In the **Settings** window for **Parameter Check**, locate the **Parameter Check** section.
- 3 In the **Condition** text field, type  $d\_O\_C < 3[\text{mm}] + R_O + R_C$ .
- 4 In the **Message** text field, type Distance between outer hole and central hole smaller than 3 mm.

5 Click  **Build Selected**.

#### *Parameter Check 3*

- 1 In the **Geometry** toolbar, click  **Programming** and choose **Parameter Check**.
- 2 In the **Settings** window for **Parameter Check**, locate the **Parameter Check** section.
- 3 In the **Condition** text field, type  $d\_O\_O < 3[\text{mm}] + 2 * R_O$ .
- 4 In the **Message** text field, type Distance between outer holes smaller than 3 mm.

5 Click  **Build Selected**.

#### *Parameter Check 4*

- 1 In the **Geometry** toolbar, click  **Programming** and choose **Parameter Check**.
- 2 In the **Settings** window for **Parameter Check**, locate the **Parameter Check** section.
- 3 In the **Condition** text field, type  $d\_C\_Cmp < 3[\text{mm}] + R_C + R_Cmp$ .
- 4 In the **Message** text field, type Distance between central hole and component less than 3 mm.

5 Click  **Build Selected**.

#### *Left Edges Indent*

At this stage, the geometry setup is ready except for adding a safeguard in case the edges of the central hole and the edges of the faces defining the indentation intersect or come close to each other. This may result in very short edges or narrow face regions, which could become difficult to mesh. Since the indentation edges connect continuous faces, we can use the Ignore Edges virtual operation to hide them from the mesher and avoid potential meshing issues. First, create coordinate-based selections that contain the edges for the indentation faces.

- 1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.

- 2 In the **Settings** window for **Box Selection**, type **Left Edges Indent** in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
- 4 Locate the **Box Limits** section. In the **x minimum** text field, type **LX-THK**.
- 5 In the **x maximum** text field, type **LX**.
- 6 In the **y minimum** text field, type **YInd**.
- 7 In the **y maximum** text field, type **WInd**.
- 8 In the **z minimum** text field, type **HInd+1[mm]**.
- 9 In the **z maximum** text field, type **HInd+10[mm]**.
- 10 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. Click **New**.
- 11 In the **New Cumulative Selection** dialog box, type **Ignore Edges Selection** in the **Name** text field.
- 12 Click **OK**.
- 13 In the **Settings** window for **Box Selection**, click  **Build Selected**.

#### *Right Edges Indent*

- 1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.
- 2 In the **Settings** window for **Box Selection**, type **Right Edges Indent** in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
- 4 Locate the **Box Limits** section. In the **x minimum** text field, type **LX-THK**.
- 5 In the **x maximum** text field, type **LX**.
- 6 In the **y minimum** text field, type **LY-WInd**.
- 7 In the **y maximum** text field, type **LY-YInd**.
- 8 In the **z minimum** text field, type **HInd+1[mm]**.
- 9 In the **z maximum** text field, type **HInd+10[mm]**.
- 10 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Ignore Edges Selection**.
- 11 Click  **Build Selected**.

#### *Central Edges Indent*

- 1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.

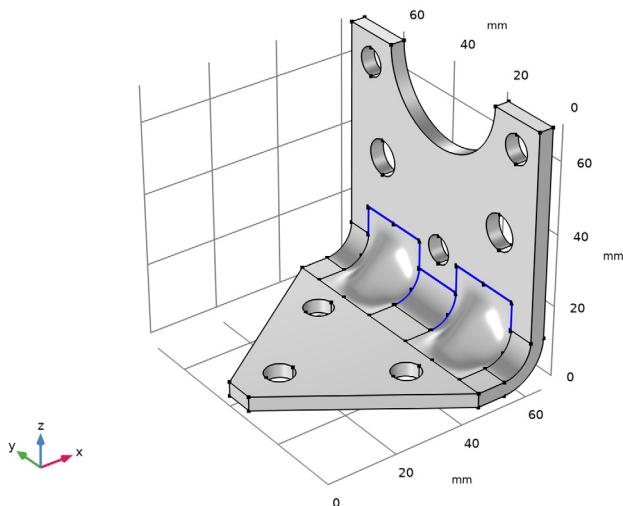
- 2 In the **Settings** window for **Box Selection**, type Central Edges Indent in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
- 4 Locate the **Box Limits** section. In the **x minimum** text field, type LX-2\*THK.
- 5 In the **x maximum** text field, type LX.
- 6 In the **y minimum** text field, type LY/2-YInd.
- 7 In the **y maximum** text field, type LY/2+YInd.
- 8 In the **z minimum** text field, type 0.
- 9 In the **z maximum** text field, type HInd+1 [mm].
- 10 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Ignore Edges Selection**.

II Click  **Build Selected**.

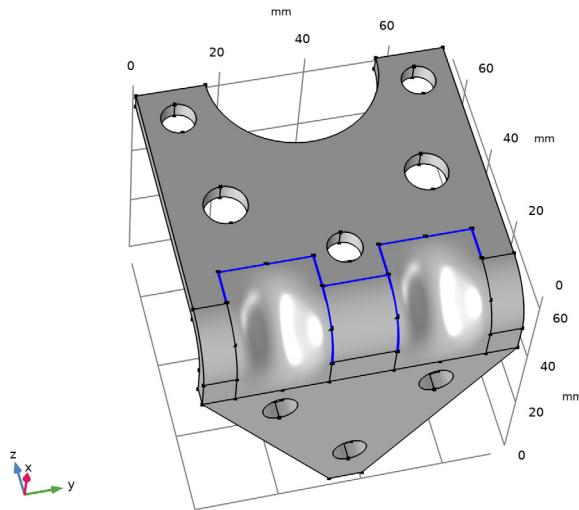
#### *Ignore Edges /*

- 1 In the **Geometry** toolbar, click  **Virtual Operations** and choose **Ignore Edges**.
- 2 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

- 3 From the **Edges to ignore** list, choose **Ignore Edges Selection**.

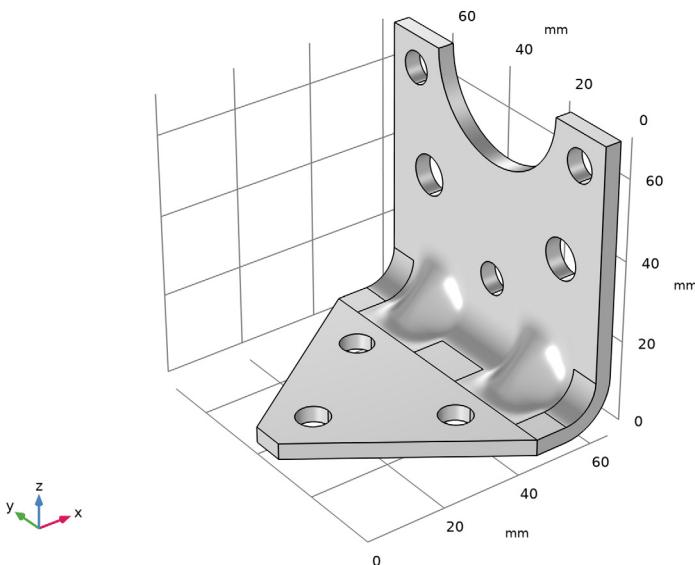


Change the model view to see the edges on the bottom side of the bracket.



- 4 Click the **Go to Default View** button in the **Graphics** toolbar.

- 5 In the **Geometry** toolbar, click  **Build All**.



#### 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

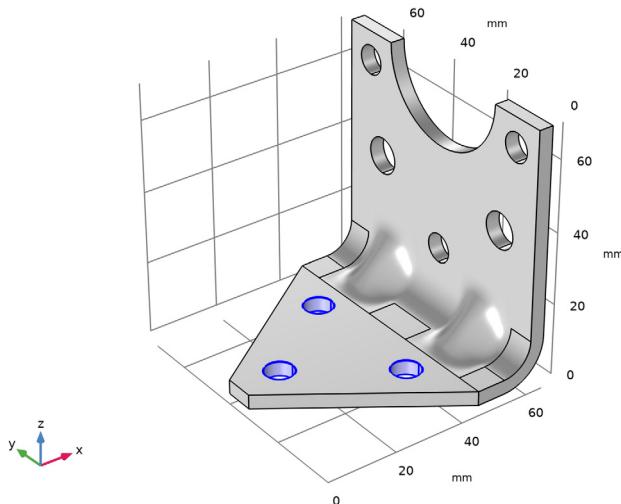
*Fixed (Bolts)*

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

The exact way the bolts clamp the bracket to the foundation is not important for the results in the part being optimized.

- 2 In the **Settings** window for **Fixed Constraint**, type **Fixed (Bolts)** in the **Label** text field.

**3** Select Boundaries 6–11 only.



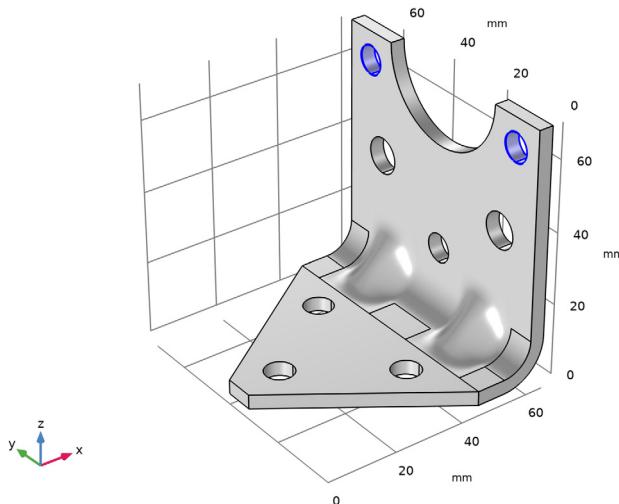
*Rigid Connector (Mounted component)*

- 1** In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

The attached component has a high stiffness, and is bolted to the two upper bolt holes. It is modeled as being rigid, with only mass properties.

- 2** In the **Settings** window for **Rigid Connector**, type **Rigid Connector (Mounted component)** in the **Label** text field.

- 3** Select Boundaries 28, 29, 38, and 39 only.



- 4** Locate the **Center of Rotation** section. From the list, choose **User defined**.

- 5** Specify the  $\mathbf{X}_c$  vector as

LX - THK/2	x
LY/2	y
LZ	z

*Mass and Moment of Inertia /*

- 1** In the **Physics** toolbar, click **Attributes** and choose **Mass and Moment of Inertia**.

- 2** In the **Settings** window for **Mass and Moment of Inertia**, locate the **Mass and Moment of Inertia** section.

- 3** In the *m* text field, type *mCmp*.

- 4** From the list, choose **Diagonal**.

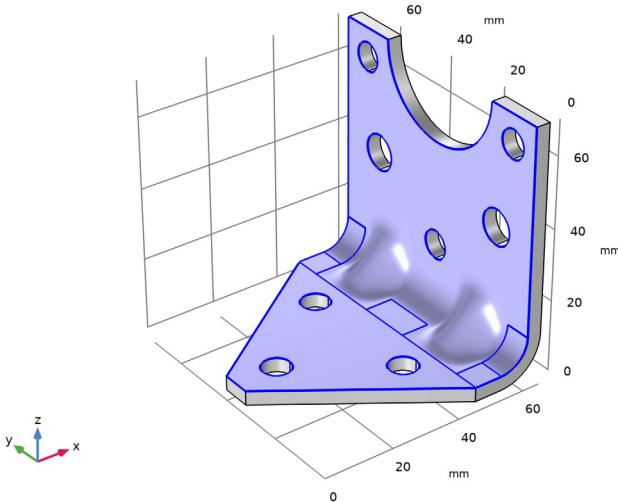
- 5** In the **I** table, enter the following settings:

IXCmp	0	0
0	IYZCmp	0
0	0	IYZCmp

## MESH 1

### Free Triangular 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 Select Boundaries 4, 15, 17, 19, 21, 24, and 26 only.



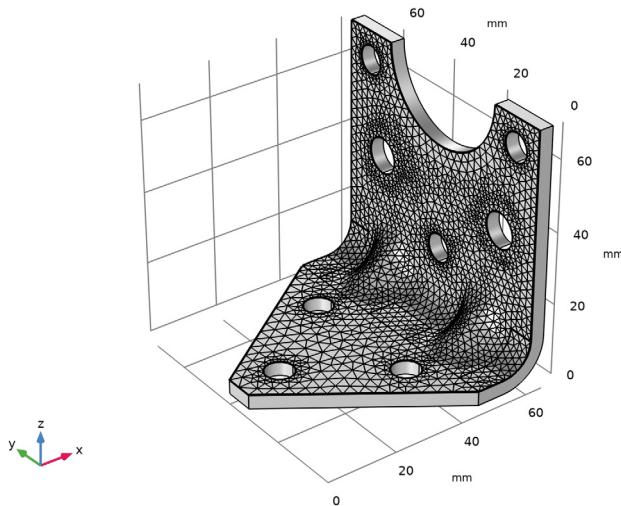
### Size 1

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Boundary 4 only.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

### Size 2

- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
- 2 Select Boundaries 15, 17, 19, 21, 24, and 26 only.
- 3 In the **Settings** window for **Size**, locate the **Element Size** section.
- 4 From the **Predefined** list, choose **Extra fine**.

**5** Click  **Build Selected**.



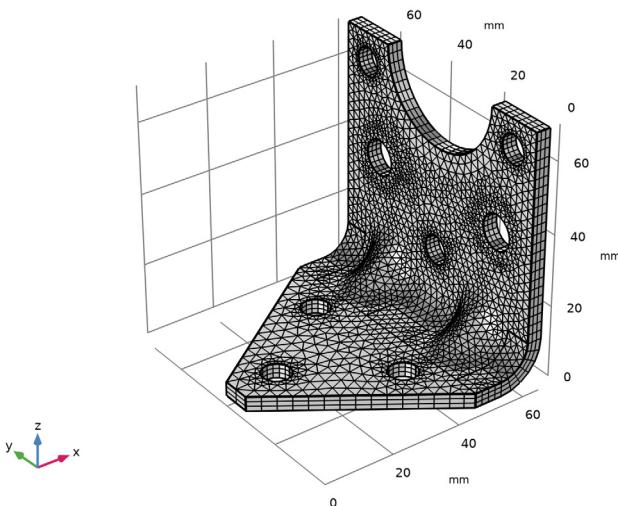
#### *Swept 1*

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

#### *Distribution 1*

- 1** Right-click **Swept 1** and choose **Distribution**.
- 2** In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3** In the **Number of elements** text field, type 3.

- 4 Click  **Build All**.



### EIGENFREQUENCY STUDY

Run an eigenfrequency study on the initial geometry.

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type **Eigenfrequency Study** in the **Label** text field.
- 3 In the **Home** toolbar, click  **Compute**.

### SOLID MECHANICS

Add the peak loads, and perform a stationary study.

*Body load 4g on bracket*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Body Load**.
- 2 In the **Settings** window for **Body Load**, type **Body load 4g on bracket** in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Force** section. Specify the  $\mathbf{F}_V$  vector as

4*g_const*solid.rho	x
4*g_const*solid.rho	y
4*g_const*solid.rho	z

*Rigid Connector (Mounted component)*

In the **Model Builder** window, click **Rigid Connector (Mounted component)**.

*Force 4g on mounted component*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Applied Force**.
- 2 In the **Settings** window for **Applied Force**, type **Force 4g** on mounted component in the **Label** text field.
- 3 Locate the **Applied Force** section. Specify the **F** vector as

4*g_const*mCmp	x
4*g_const*mCmp	y
4*g_const*mCmp	z

#### **ADD STUDY**

- 1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

#### **STATIONARY STUDY**

- 1 In the **Model Builder** window, click **Study 2**.
- 2 In the **Settings** window for **Study**, type **Stationary Study** in the **Label** text field.
- 3 In the **Home** toolbar, click  **Compute**.

#### **DEFINITIONS**

Prepare for the optimization by adding variables for the bracket mass and the maximum stress.

*Stress Optimization Domain*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Stress Optimization Domain** in the **Label** text field.
- 3 Select Domain 2 only.

*Domain Probe 1*

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Domain Probe**.

- 2 In the **Settings** window for **Domain Probe**, type mass in the **Variable name** text field.
- 3 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.
- 4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Material properties>solid.rho - Density - kg/m<sup>3</sup>**.

#### *Domain Probe 2*

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Domain Probe**.
- 2 In the **Settings** window for **Domain Probe**, type maxStress in the **Variable name** text field.
- 3 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.
- 4 Locate the **Source Selection** section. From the **Selection** list, choose **Stress Optimization Domain**.
- 5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress>solid.mises - von Mises stress - N/m<sup>2</sup>**.

## **RESULTS**

Modify the default stress plot to monitor the geometry and stresses in the optimized region.

#### *Stress in Optimized Region*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, type Stress in Optimized Region in the **Label** text field.

#### *Volume 1*

- 1 In the **Model Builder** window, expand the **Stress in Optimized Region** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.

#### *Deformation*

- 1 In the **Model Builder** window, expand the **Volume 1** node.
- 2 Right-click **Deformation** and choose **Delete**.

#### *Selection 1*

- 1 In the **Model Builder** window, right-click **Volume 1** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.

- 3 From the **Selection** list, choose **Stress Optimization Domain**.

#### *Marker 1*

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

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

#### **ROOT**

Set up the optimization study.

#### **ADD STUDY**

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

- 2 Go to the **Add Study** window.

- 3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

- 4 Click **Add Study** in the window toolbar.

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

#### **OPTIMIZATION STUDY**

In the **Settings** window for **Study**, type **Optimization Study** in the **Label** text field.

#### *Optimization*

In the **Study** toolbar, click  **Optimization** and choose **Optimization**.

#### *Eigenfrequency*

- 1 In the **Study** toolbar, click  **Study Reference**.

- 2 In the **Settings** window for **Study Reference**, type **Eigenfrequency** in the **Label** text field.

- 3 Locate the **Study Reference** section. From the **Study reference** list, choose **Eigenfrequency Study**.

#### *Stationary*

- 1 In the **Study** toolbar, click  **Study Reference**.

- 2 In the **Settings** window for **Study Reference**, type **Stationary** in the **Label** text field.

- 3 Locate the **Study Reference** section. From the **Study reference** list, choose **Stationary Study**.

#### *Optimization*

- 1 In the **Model Builder** window, click **Optimization**.

- 2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.

- 3 From the **Method** list, choose **COBYLA**.

- 4 Find the **Solver settings** subsection. Clear the **Stop if error** check box.
- 5 Locate the **Constraints** section. Select the **Enforce design constraints strictly** check box.
- 6 Click **Replace Expression** in the upper-right corner of the **Objective Function** section.  
From the menu, choose **Component 1>Definitions>comp1.mass - Domain Probe 1 - kg**.
- 7 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
comp1.mass	Bracket mass	Stationary

The first eigenfrequency is to be used in the optimization.

- 8 From the **Solution** list, choose **Use first**.
- 9 Locate the **Control Variables and Parameters** section. Click  **Load from File**.
- 10 Browse to the model's Application Libraries folder and double-click the file `bracket_import_optimization_ctrlvars.txt`.
- II Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound	Evaluate for
real(freq)	minFreq		Eigenfrequency
comp1.maxStress/ maxStressLimit		1	Stationary

- I2 Locate the **Output While Solving** section. Select the **Plot** check box.

- I3 From the **Plot group** list, choose **Stress in Optimized Region**.

If some configurations are not valid, the optimization procedure should still continue. The default is to stop if an error occurs.

### *Solution 3*

- 1 In the **Study** toolbar, click  **Show Default Solver**.

Run the optimization.

- 2 Click  **Compute**.

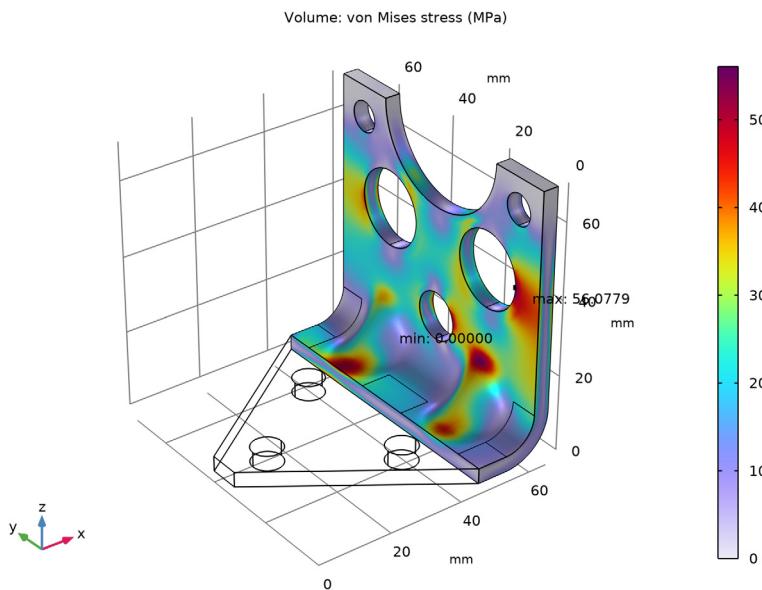
## RESULTS

### *Stress in Optimized Region*

Examine the stress distribution in the optimized configuration.

- I In the **Model Builder** window, under **Results** click **Stress in Optimized Region**.

- 2 In the **Stress in Optimized Region** toolbar, click  **Plot**.



On the last line of **Global Constraints Table 7**, you will find the values of the natural frequency and maximum stress in the optimized configuration, as well as the values of the other constraints.

#### OBJECTIVE TABLE 3

On the last line of **Objective Table 3**, you will find the optimal set of parameters and the minimum weight. Note that the value in the **Objective** column can be colored orange if the solution violates a constraint slightly but is still accepted within the tolerances.

- I In the **Objective Table 3** table, right-click the last row and select **Copy Selected Rows to New Parameter Cases**.

This last step creates a new parameter case with the optimum parameter values. Next, you can rebuild the geometry using the optimized parameters.

#### GLOBAL DEFINITIONS

*Parameters I*

- 1 In the **Model Builder** window, under **Global Definitions>Parameters I** click **Case 2**.
- 2 In the **Settings** window for **Case**, type Optimized Values in the **Label** text field.
- 3 In the **Model Builder** window, click **Parameters I**.

- 4** In the **Settings** window for **Parameters**, click **Optimized Values**.

#### **GEOMETRY I**

- 1** In the **Home** toolbar, click **Build All**.
- 2** In the **Model Builder** window, under **Component 1** click **Geometry 1**.

