



Model created in COMSOL Multiphysics 6.4

Optimization of an Imported Bracket Geometry – with Parameter Expressions

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 as a way to reparameterize the already existing holes in the bracket in order 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 make sure that the holes will not come within close proximity of each other or the edges of the bracket. One way you can achieve this is by defining the constraints as parameter expressions based on geometrical relations, and the actual dimensions of the geometry. You can then enter the expressions and their allowed range into the constraint section of the optimization study settings. The solver can then be set to evaluate the design constraints before running the forward problem, and if a constraint expression is out of bounds the solver will continue to the next iteration.

The tutorial [Optimization of an Imported Bracket Geometry — with Measuring Dimensions](#) demonstrates an alternative way of setting up geometrical constraints by using measuring dimensions and parameter check features in the geometry sequence to abort the building of the geometry, and the solving of the forward problem for those combinations of the control parameters for which a constraint is outside the allowed range.

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 $7.1 \cdot 10^{-4} \text{ kg} \cdot \text{m}^2$, and the moment of inertia around the two transverse axes is $9.3 \cdot 10^{-4} \text{ kg} \cdot \text{m}^2$.

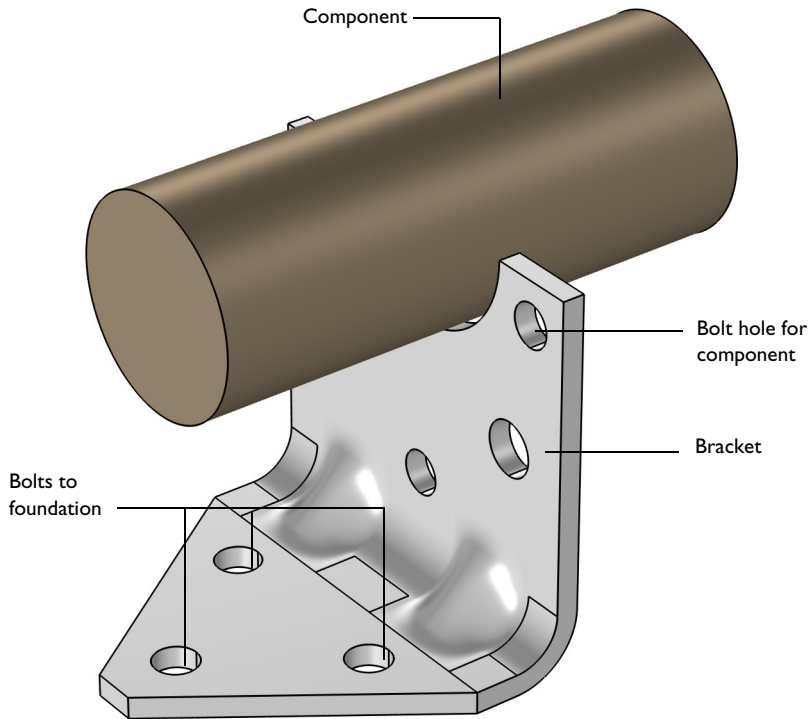


Figure 1: Bracket supporting a heavy component.

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 1: 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

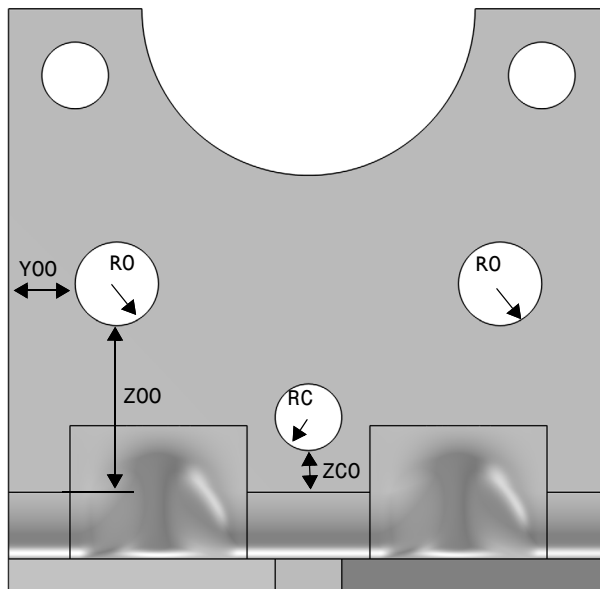


Figure 2: Geometrical parameters that are optimized in the study.

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.

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](#).

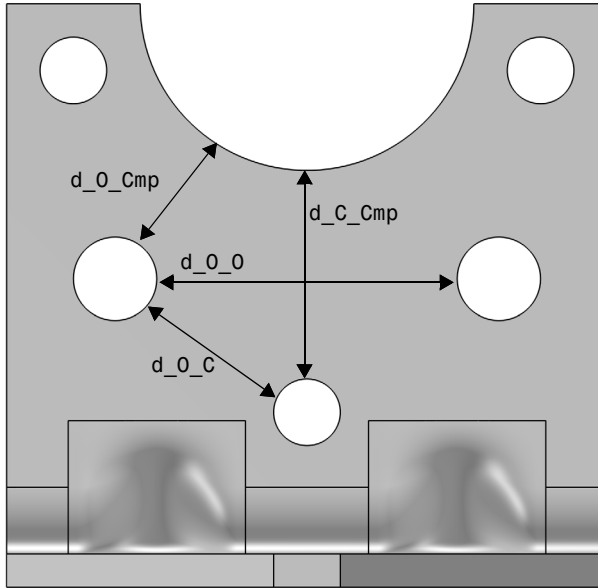


Figure 3: Geometrical constraints.

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

The geometrical constraints do not depend on the solution of the forward problem and are considered design constraints by the optimization solver. By choosing to strictly enforce design constraints in the optimization study settings, the solver evaluates all design

constraints before running the forward problem. If infeasible constraints are found, the forward problem is not run, and the optimization solver proceeds to the next iteration.

Results and Discussion

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

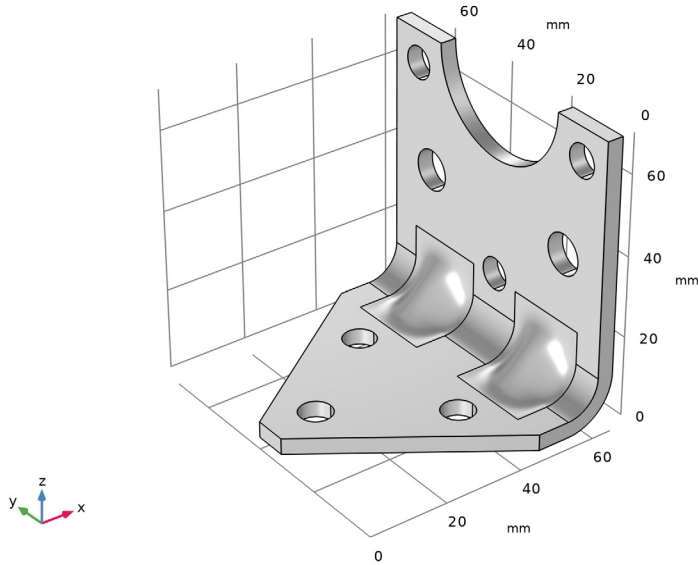


Figure 4: Initial geometry.

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	4.02	3	15
ZCO	1.55	1	23
RO	11.09	3	15
YOO	7.02	3	29
ZOO	10.93	8	30

The weight of the optimized bracket is about 185 g, a reduction of 19 g from the original 204.10 g. The stresses from the shock load on the optimized geometry are shown in [Figure 5](#).

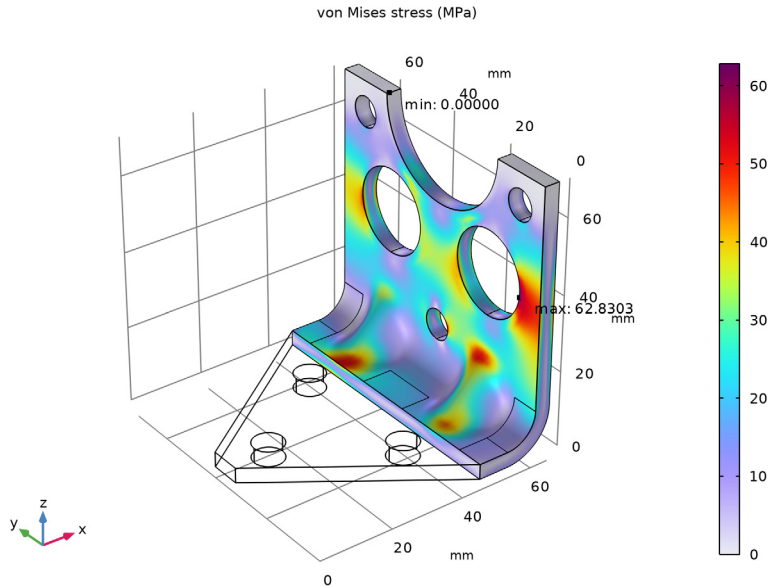


Figure 5: Stresses at peak load in the optimized design.

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_expressions

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

NEW

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



MODEL WIZARD

- 1 In the **Model Wizard** window, click .
- 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

The **Parameters I** node contains the parameter expressions used for the model setup. These include the parameters with the original, according to the imported CAD file, dimension values for the features we are optimizing, the parameter expressions used in the offset and transform faces operations that modify the geometry, the parameter expressions for the geometrical constraints, and the parameters used for the physics setup. The expressions in the file you will load into the model have been defined such that the control parameters for the optimization are the actual bolt hole radii and positions (that is, not the displacement relative to the imported CAD file).




- 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_expressions_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.

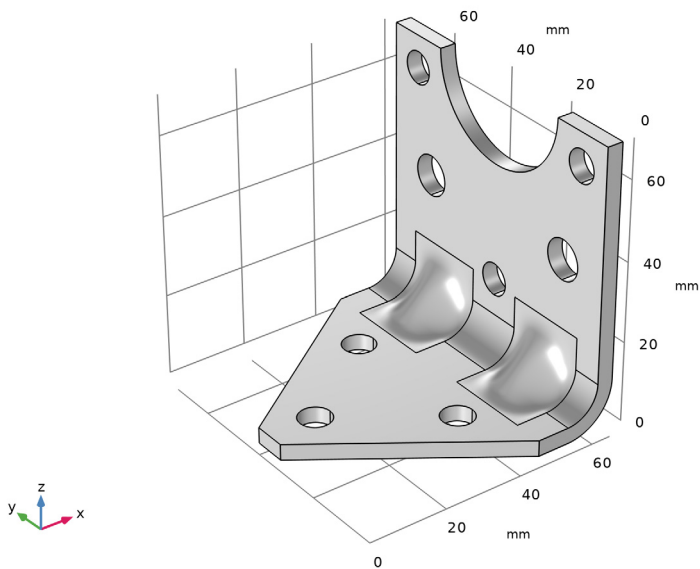
GEOMETRY I

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

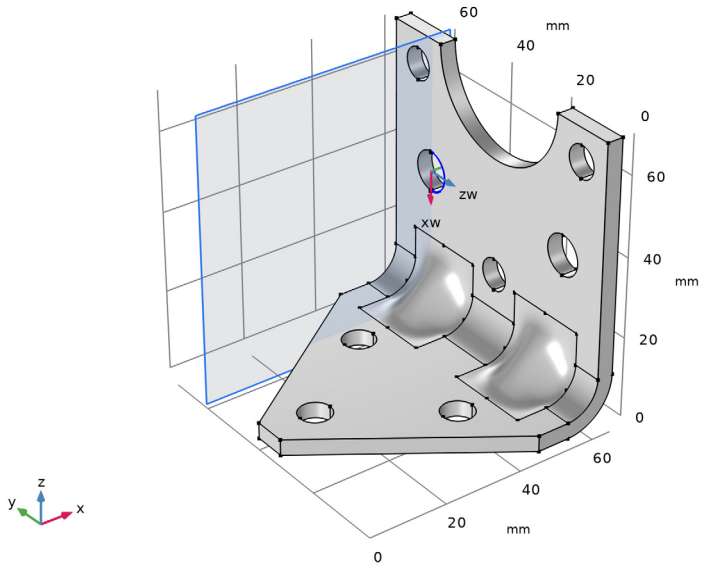
- 1 In the **Model Builder** window, under **Component 1** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
- 3 From the **Geometry representation** list, choose **CAD kernel**.

Import 1

- 1 In the **Geometry** toolbar, click  **Import**.
- 2 In the **Settings** window for **Import**, locate the **Source** section.
- 3 Click  **Browse**.
- 4 Browse to the model's Application Libraries folder and double-click the file `bracket_import_optimization_geom.step`.
- 5 Locate the **Length Unit** section. From the **Length unit** list, choose **From CAD document**.
With this option the length unit of the model 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 Locate the **Source** section. Click  **Import**.



5 Select the **Reverse normal direction** checkbox.



6 Click  **Build Selected**.

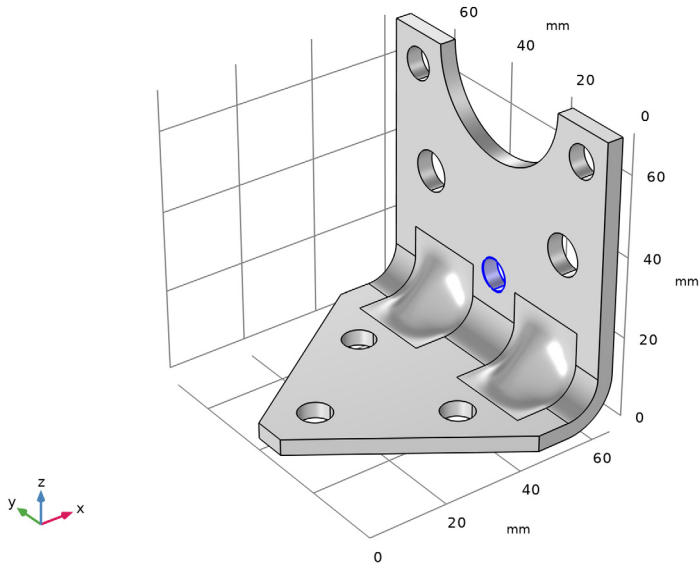
Coordinate System - Outer Hole 2

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**.

3 On the object **impl1**, 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 `scaleRC`.

6 Locate the **Displacement** section. In the **xw** text field, type `ZC0disp`.

7 Click  **Build Selected**.

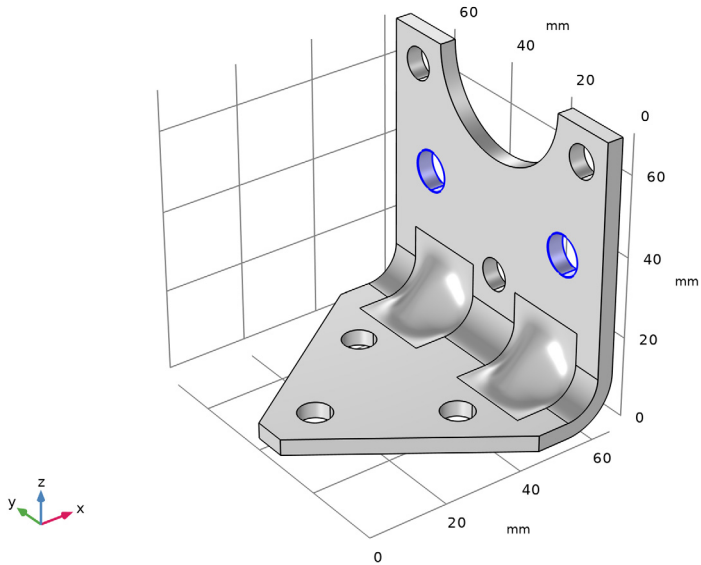
Offset - Outer Holes

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. And two Transform faces operations to displace each hole separately.

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 R0off.

5 Select the **Reverse side** checkbox.

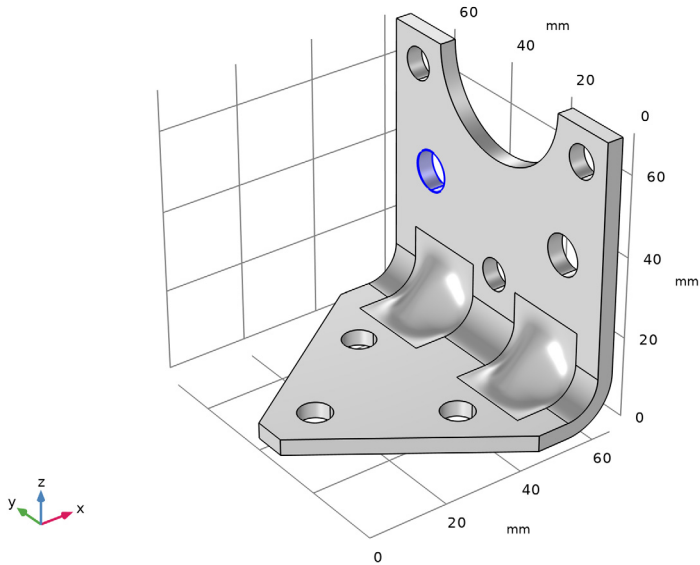
6 Click  **Build Selected**.

Displacement - Outer Hole 1

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 Z00disp.

6 In the **zw** text field, type Y00disp.

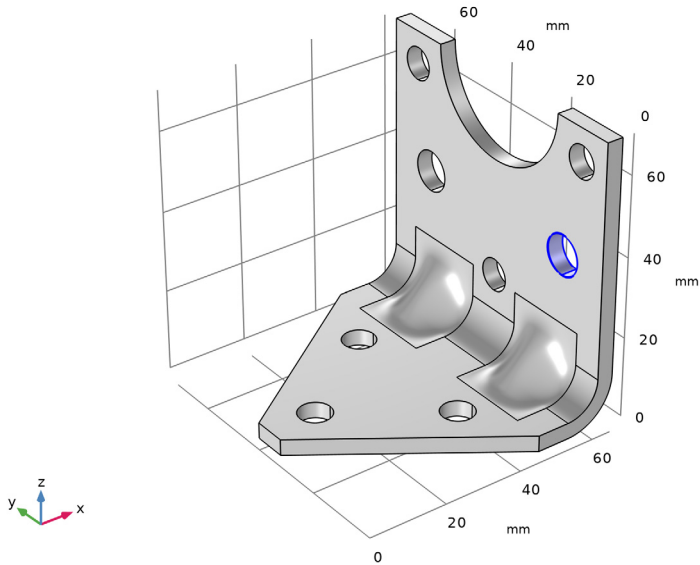
7 Click  **Build Selected**.

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 Z00disp.

6 In the **zw** text field, type Y00disp.

7 Click  **Build Selected**.

Work Plane 4

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

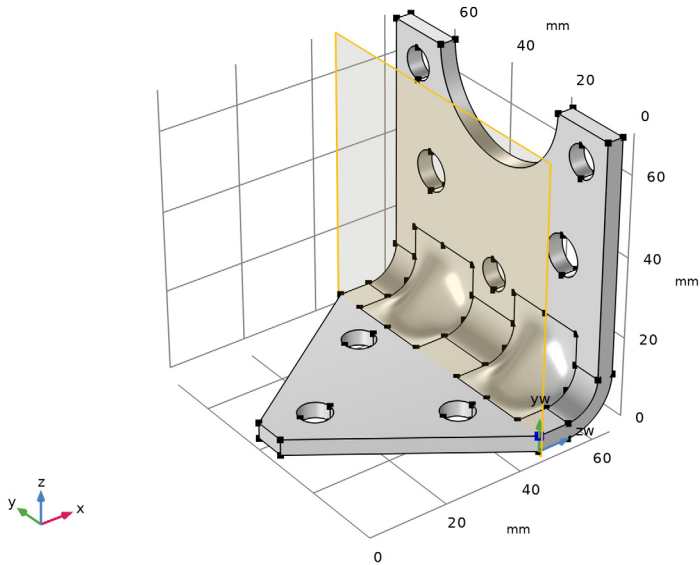
1 In the **Geometry** toolbar, click  **Work Plane**.

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 I

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**.



At this stage, the geometry set up 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 any potential meshing issues. First, create-coordinate based selections that contain the edges for the indentation faces.

Left Edges Indent

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 **7[mm]**.
- 7 In the **y maximum** text field, type **22[mm]**.
- 8 In the **z minimum** text field, type **13[mm]**.
- 9 In the **z maximum** text field, type **22[mm]**.
- 10 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. Click **New**.
- 11 In the **New Cumulative Selection** dialog, 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-22[mm]**.
- 7 In the **y maximum** text field, type **LY-7[mm]**.
- 8 In the **z minimum** text field, type **13[mm]**.
- 9 In the **z maximum** text field, type **22[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

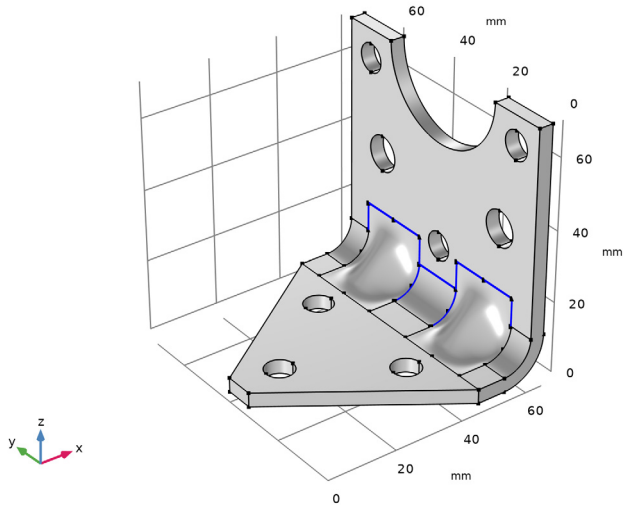
- 1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.

- 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 $LY/2-8[mm]$.
- 7 In the **y maximum** text field, type $LY/2+8[mm]$.
- 8 In the **z minimum** text field, type $0[mm]$.
- 9 In the **z maximum** text field, type $12[mm]$.
- 10 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Ignore Edges Selection**.
- 11 Click  **Build Selected**.

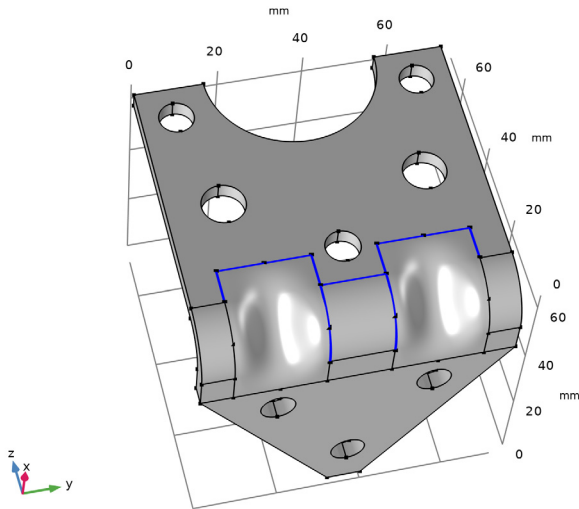
Ignore Edges I

- 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**.

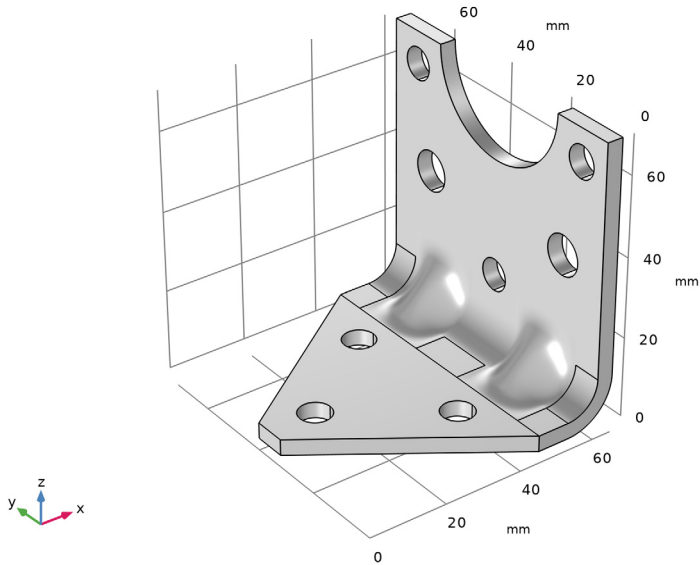


Change the model view to see the edges on the bottom side of the bracket.





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 the **Add to Component** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

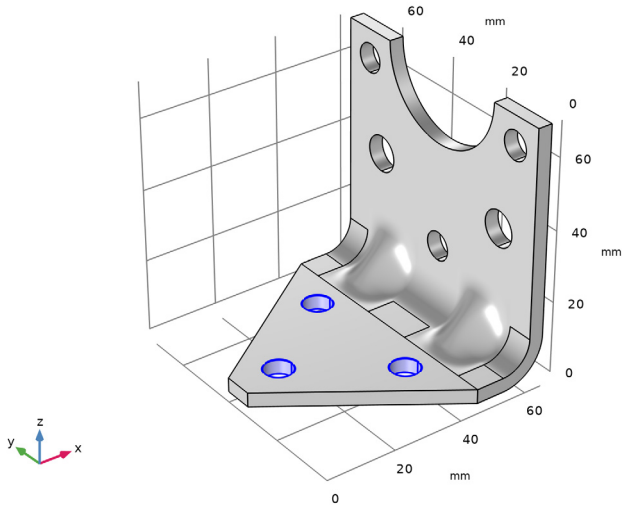
SOLID MECHANICS

Fixed (Bolts)

- 1 In the **Physics** toolbar, click  **Boundaries** 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.



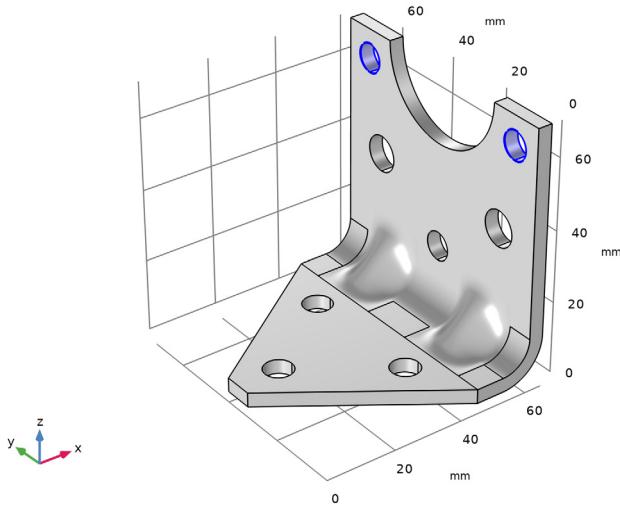
Rigid Connector (Mounted component)

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 I

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**.

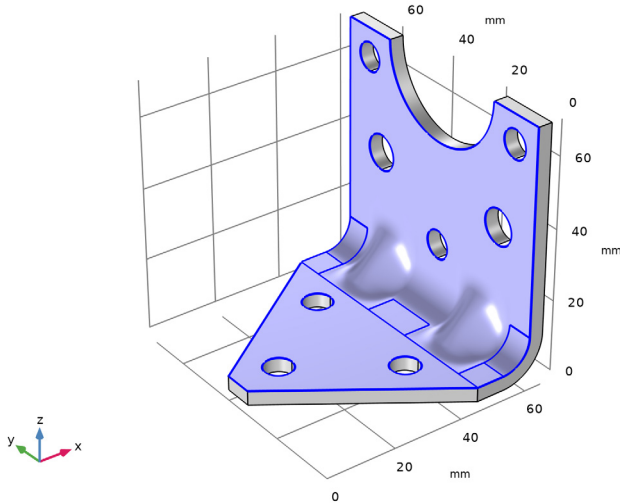
5 Specify the \mathbf{I} matrix as

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.



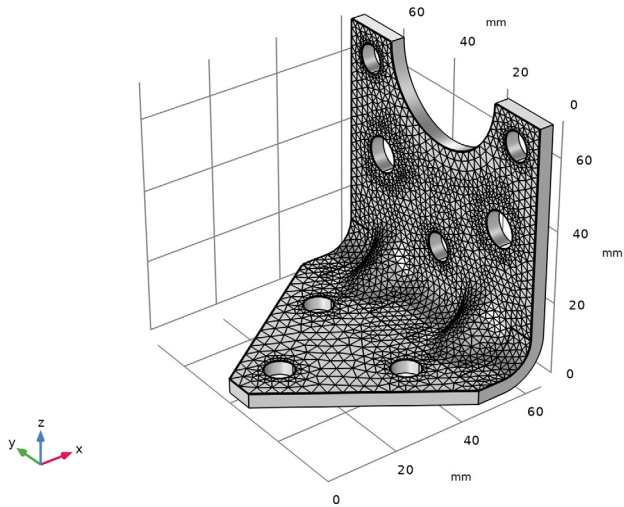
Size 1

- 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.**



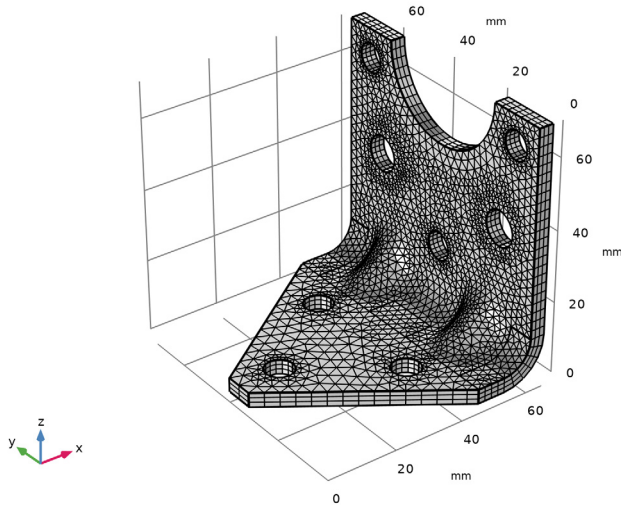
Swept 1

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 **Study** 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

$4 \cdot g_{\text{const}} \cdot \text{solid.rho}$	x
$4 \cdot g_{\text{const}} \cdot \text{solid.rho}$	y
$4 \cdot g_{\text{const}} \cdot \text{solid.rho}$	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 the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.


STATIONARY STUDY

- 1 In the **Settings** window for **Study**, type Stationary Study in the **Label** text field.
- 2 In the **Study** 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/m³**.

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/m²**.

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 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 I

- 1 Right-click **Volume I** 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 the **Add Study** button 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.


General Optimization

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

Eigenfrequency

- 1 In the **Study** toolbar, click  **More Study Extensions** and choose **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  **More Study Extensions** and choose **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**.


General Optimization

- 1 In the **Model Builder** window, click **General Optimization**.
- 2 In the **Settings** window for **General Optimization**, locate the **Optimization Solver** section.
- 3 From the **Method** list, choose **COBYLA**.

- 4 Click to expand the **Solver Settings** section. Clear the **Stop if error** checkbox.
- 5 Find the **Constraint settings** subsection. Select the **Enforce design constraints strictly** checkbox.
- 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`.
- 11 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
d_O_Cmp	3 [mm]		Eigenfrequency
d_C_Cmp	3 [mm]		Eigenfrequency
d_O_C	3 [mm]		Eigenfrequency
d_O_0	3 [mm]		Eigenfrequency

- 12 Click to expand the **Output** section. Select the **Plot** checkbox.
- 13 In the table, enter the following settings:

Plot group	Plot window
Stress in Optimized Region	Graphics

- 14 Select the **Copy optimized parameters to a parameter case** checkbox.

This last step automatically creates a new parameter case with the optimum parameter values.

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**.

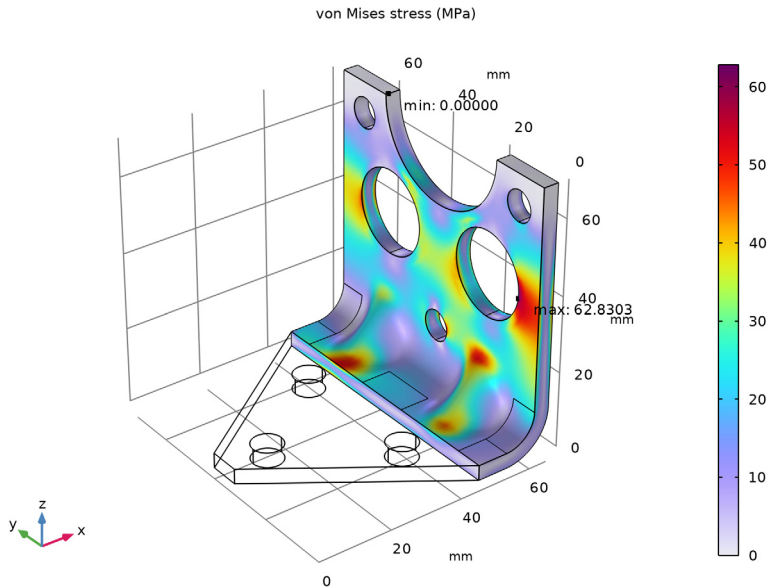
RESULTS

Stress in Optimized Region

Examine the stress distribution in the optimized configuration.

1 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 row 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 row 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.


Next, rebuild the geometry using the optimum parameter values.

GLOBAL DEFINITIONS

Parameters 1

- 1 In the **Model Builder** window, expand the **Global Definitions > Parameters 1** node, then click **Case 2**.
- 2 In the **Settings** window for **Case**, type Optimum values in the **Label** text field.
- 3 In the **Model Builder** window, click **Parameters 1**.
- 4 In the **Settings** window for **Parameters**, click  **Optimum values**.

GEOMETRY 1

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

