Created in COMSOL Multiphysics 6.2

Multistudy Optimization of a Bracket

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.

In this model, the weight of a mounting bracket is reduced, given an upper bound on the stresses and a lower bound on the first natural frequency.

The bracket 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 in order 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.

Note: This application requires the Optimization 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^2, and the moment of inertia around the two transverse axes is $9.3 \cdot 10^{-4}$ kg·m^2.

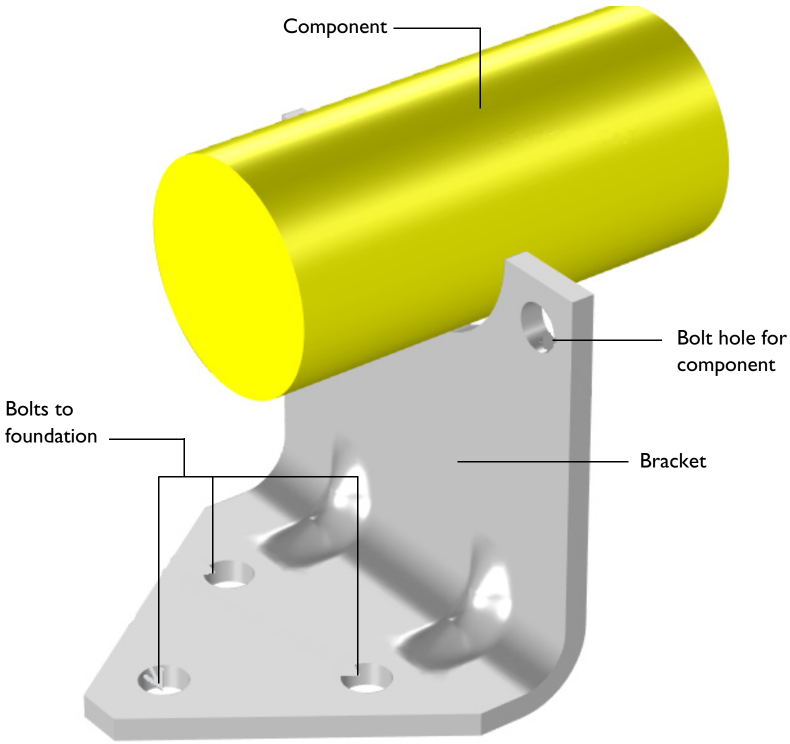


Figure 1: Bracket supporting a heavy component.

The idea is to reduce the weight by drilling holes in the vertical surface of the bracket, and at the same time change the dimensions of the indentations, in order to offset the loss in stiffness.

OPTIMIZATION PARAMETERS

Six 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

TABLE 1: GEOMETRICAL PARAMETERS.

Parameter	Description	Lower limit (mm)	Upper limit (mm)
r0	Radius of the outer hole	3	15
z0o	Vertical distance from the bend to the edge of the outer hole	8	30
y0o	Horizontal distance from the edge of the bracket to outer hole	3	29
wInd	Width of the indentation	8	20

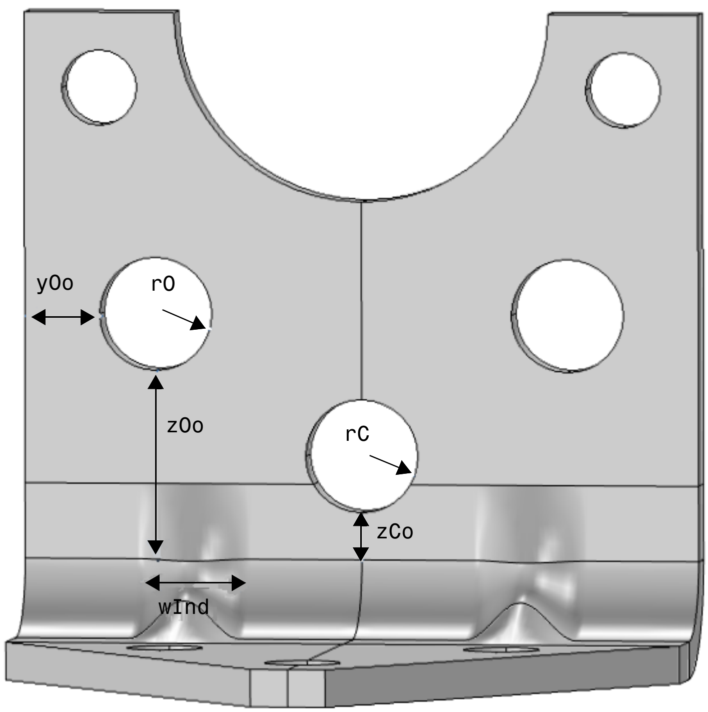


Figure 2: Optimization parameters.

CONSTRAINTS

- 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 equivalent 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.

- There must be at least 3 mm of material between two holes, or between a hole and an edge. This criterion is enforced both through the limits on the control parameters and as constraints. The geometrical constraints are shown in Figure 3.

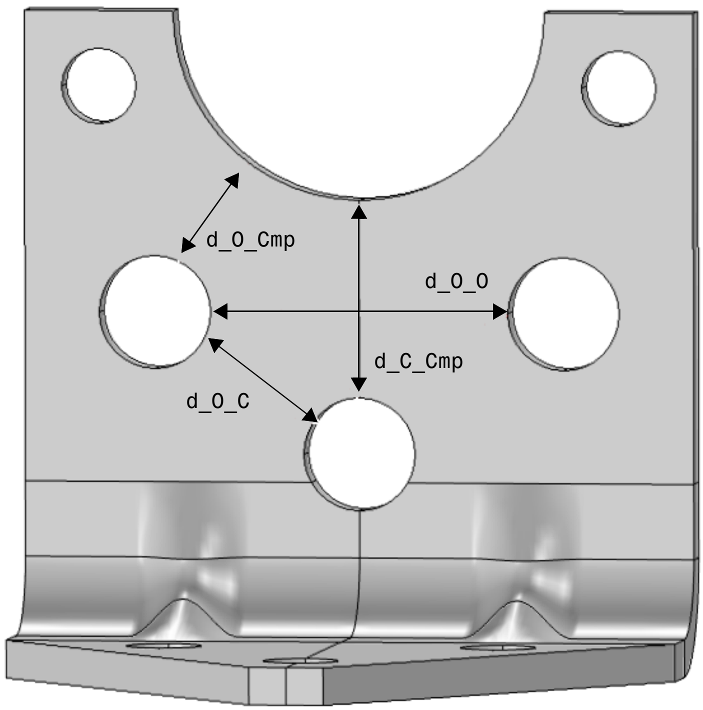


Figure 3: Geometrical constraints.

The COBYLA solver uses sampling in the control variable space to approximate both 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) constraint and bound violations.

Bounds and linear constraints are generally satisfied to high precision at the optimum point returned by the solver, but nonlinear constrains 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.

Results and Discussion

The initial geometry used in the optimization is shown in Figure 4. Three rather small holes have been introduced.

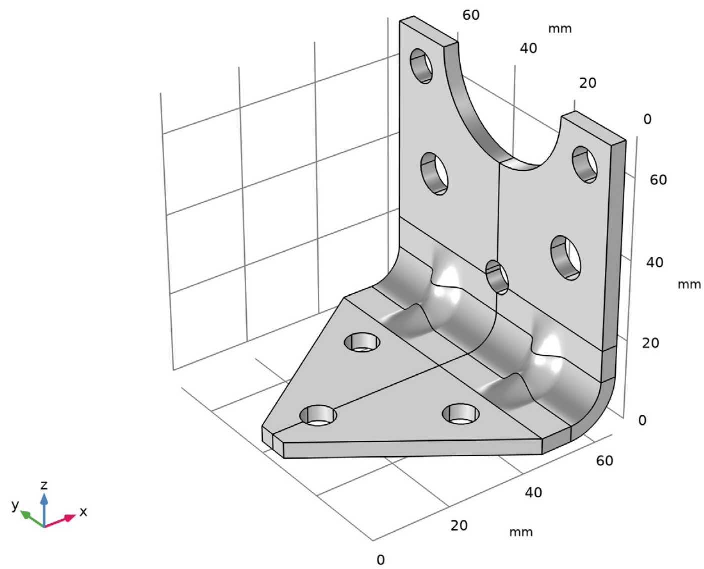


Figure 4: Initial geometry.

The optimal values of the geometrical parameters are shown in Table 2.

TABLE 2: OPTIMAL VALUES.

Parameter	**Optimal value (mm)**	**Lower limit (mm)**	**Upper limit (mm)**
`rC`	11.8	3	15
`zCo`	2.7	1	23
`rO`	9.0	3	15
`zOo`	9.5	8	30
`yOo`	22.3	3	29
`wInd`	20	8	20

The weight of the optimized bracket is about 188 g, a reduction of 17 g from the original 205 g. The stresses from the shock load on the optimized geometry are shown in Figure 5

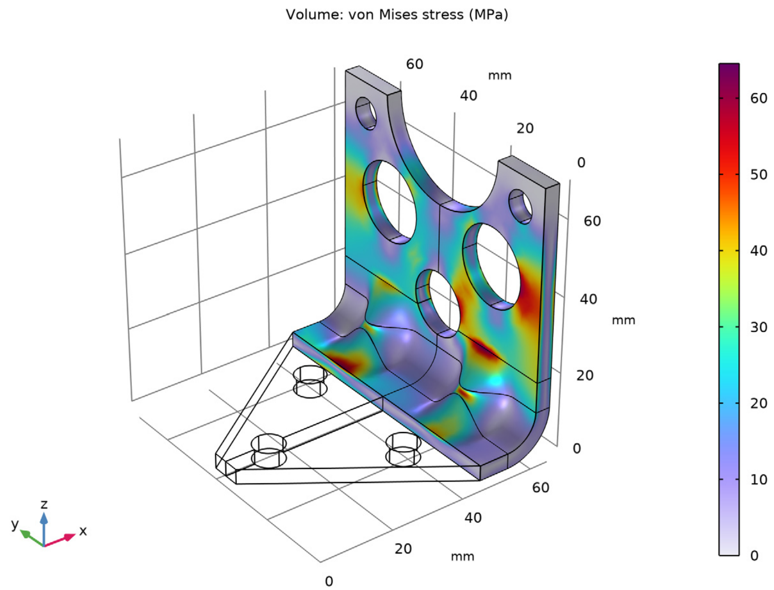


Figure 5: Stresses at peak load in the optimized design.

The optimal solution gives three fairly large holes, and the widest possible indentation.

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 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: `Structural_Mechanics_Module/Sensitivity_and_Optimization/multistudy_bracket_optimization`

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 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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `multistudy_bracket_optimization_parameters.txt`.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.
4 Locate the **Advanced** section. From the **Geometry representation** list, choose **CAD kernel**.
5 Select the **Design Module Boolean operations** check box.

The geometry sequence for the model (see Figure 1) is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

6 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
7 Browse to the model's Application Libraries folder and double-click the file `multistudy_bracket_optimization_geom_sequence.mph`.

8 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 (SOLID)

Fixed (Bolts)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** 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 10–13 and 15–22 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 48, 49, 52, 53, and 75–78 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

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, 8, 26, 30, 35, 39, 41, 44, 60, and 64 only.

Size 1

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

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

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

4 Select Boundaries 4, 8, 26, 30, 44, and 64 only.

5 Right-click **Size 1** and choose **Duplicate**.

Size 2

1 In the **Model Builder** window, click **Size 2**.

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

3 From the **Predefined** list, choose **Extra fine**.

4 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

5 Select Boundaries 35, 39, 41, and 60 only.

6 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.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Eigenfrequency Study** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, click to expand the **Results While Solving** section.

3 From the **Probes** list, choose **None**.

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

SOLID MECHANICS (SOLID)

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.

Step 1: Stationary

1 In the **Model Builder** window, under **Stationary Study** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 From the **Probes** list, choose **None**.

4 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 Domains 3–10 only.

Domain Probe 1 (dom1)

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 (comp1)>Solid Mechanics>Material properties>solid.rho - Density - kg/m³**.

Domain Probe 2 (dom2)

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 (comp1)>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 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**.

4 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 (comp1)>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 multistudy_bracket_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_O	3[mm]		Eigenfrequency

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

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

14 From the **Probes** list, choose **None**.

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

Solution 3 (sol3)

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

Run the optimization.

2 Click **Compute**.

RESULTS

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.

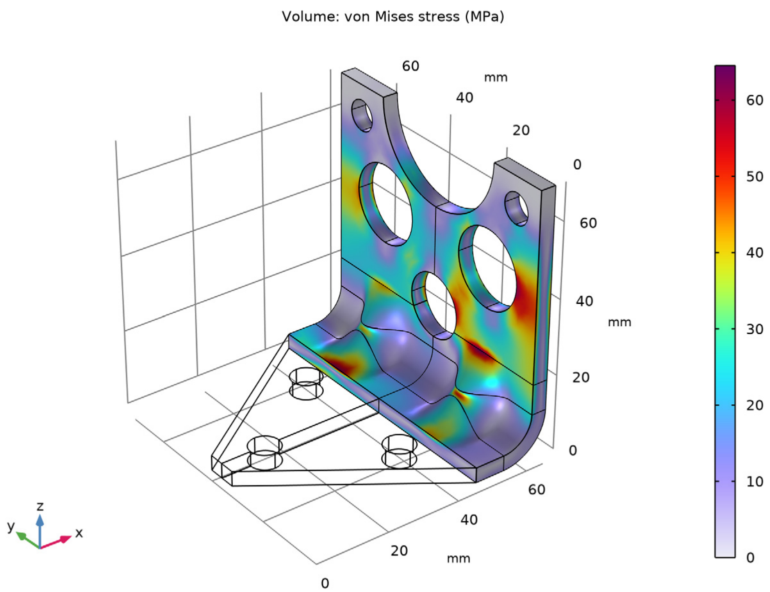
On the last line of **Global Constraints Table 7** you will find the values of the natural frequency and maximum stress in the optimized configuration, as well as the values of the other constraints.

Examine the stress distribution in the optimized configuration.

Stress in Optimized Region

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results> Stress in Optimized Region**.

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



Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

GLOBAL DEFINITIONS

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `multistudy_bracket_optimization_geom_sequence_parameters.txt`.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

4 Locate the **Advanced** section. From the **Geometry representation** list, choose **CAD kernel**.

5 Select the **Design Module Boolean operations** check box.

Work Plane 1 (wp1)

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 **zy-plane**.

4 In the **x-coordinate** text field, type `lX-rOut`.

5 Click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Parametric Curve 1 (pc1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Expressions** section.

3 In the **xw** text field, type `if((abs(s-0.5)<wInd/lY),dInd/2*(1+cos(pi*lY/if(wInd>4[mm],wInd,4[mm])*(s-0.5))),0)`.

4 In the **yw** text field, type `s*lY/2`.

5 Click **Build Selected**.

Work Plane 1 (wp1)>Copy 1 (copy1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **pc1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **xw** text field, type `thk`.

Work Plane 1 (wp1)>Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **xw** text field, type `thk`.

Work Plane 1 (wp1)>Copy 2 (copy2)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **ls1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

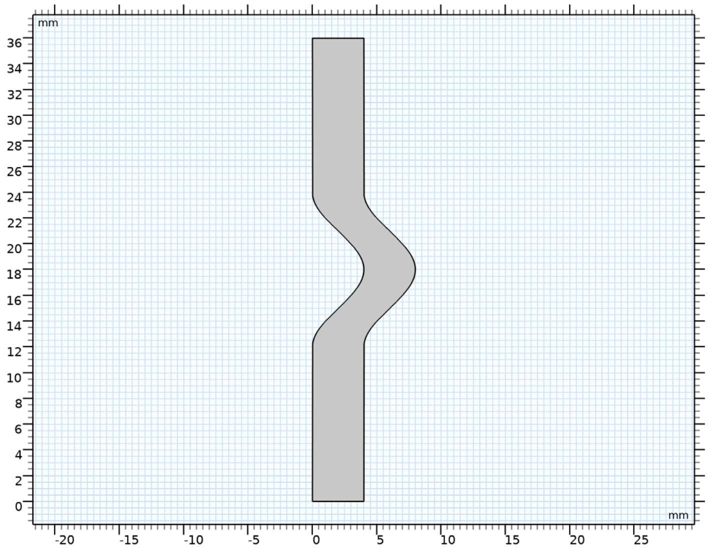
4 In the **yw** text field, type `lY/2`.

Work Plane 1 (wp1)>Convert to Solid 1 (csol1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Convert to Solid**, click **Build Selected**.



Revolve 1 (rev1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `90`.

5 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

6 Find the **Point on the revolution axis** subsection. In the **x** text field, type `lX-rOut`.

7 In the **z** text field, type `rOut`.

8 Find the **Direction of revolution axis** subsection. In the **y** text field, type `-1`.

9 Click **Build All Objects**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `lX-rOut-2*thk`.

4 In the **Depth** text field, type `lY/2`.

5 In the **Height** text field, type `thk`.

6 Click **Build All Objects**.

Loft 1 (loft1)

1 In the **Geometry** toolbar, click **Loft**.

2 In the **Settings** window for **Loft**, locate the **General** section.

3 Clear the **Unite with input objects** check box.

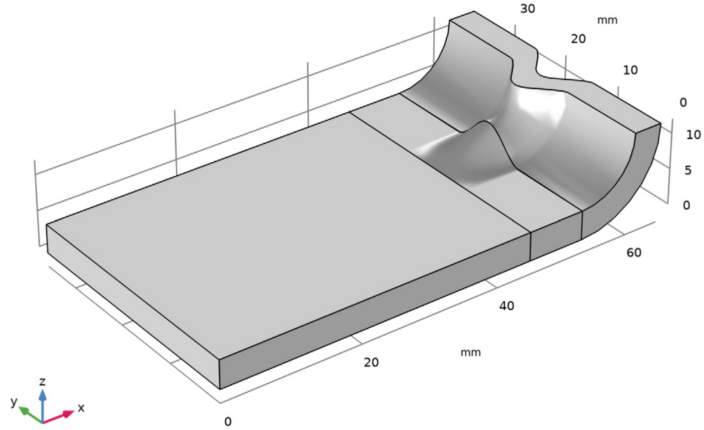
4 Click to expand the **Start Profile** section. Click to select the **Activate Selection** toggle button for **Start profile**.

5 On the object **rev1**, select Boundary 1 only.

6 Click to expand the **End Profile** section. Click to select the **Activate Selection** toggle button for **End profile**.

7 On the object **blk1**, select Boundary 5 only.

8 Click **Build All Objects**.



Mirror 1 (mir1)

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

2 Select the object **loft1** only.

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

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

5 Locate the **Point on Plane of Reflection** section. In the **x** text field, type `lX-rOut`.

6 In the **z** text field, type `rOut`.

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

8 Click **Build All Objects**.

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `thk`.

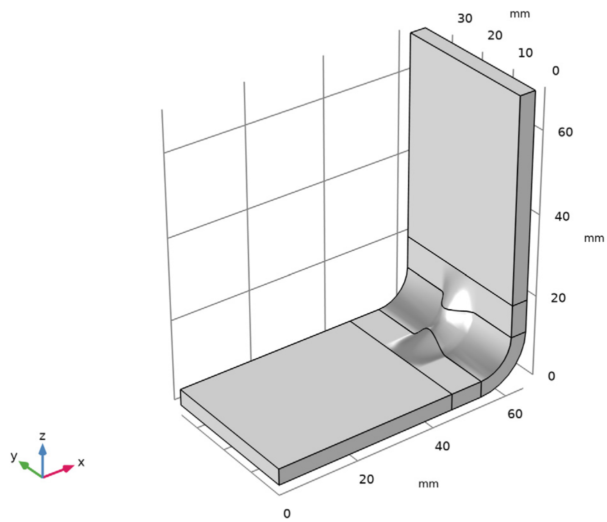
4 In the **Depth** text field, type `lY/2`.

5 In the **Height** text field, type `lZ-rOut-2*thk`.

6 Locate the **Position** section. In the **x** text field, type `lX-thk`.

7 In the **z** text field, type `rOut+2*thk`.

8 Click **Build All Objects**.



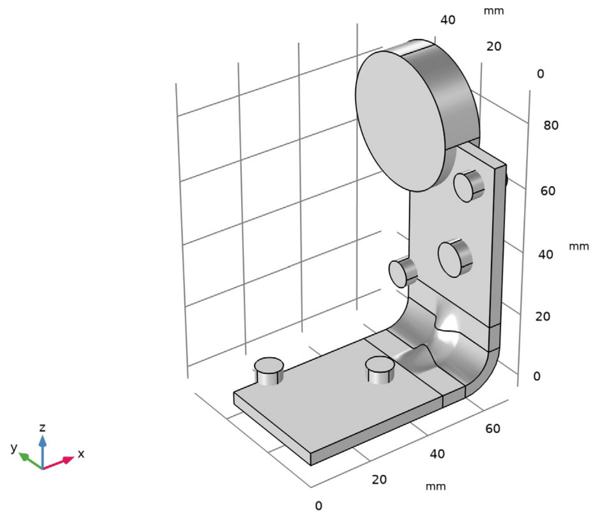
Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder** six times.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape**, **Position**, and **Axis** sections and type the following settings for the six newly created nodes.

Name	Radius	Height	x	y	z	Axis type
Cylinder 1	dCmp/2	3*thk	lX-2*thk	lY/2	lZ	x-axis
Cylinder 2	bDia/2	3*thk	lX-2*thk	bDia	lZ-bDia	x-axis
Cylinder 3	bDia/2	3*thk	lX-rOut-2*thk-bDia	lY/4	-thk	z-axis
Cylinder 4	bDia/2	3*thk	1.5*bDia	lY/2	-thk	z-axis
Cylinder 5	r0	3*thk	lX-2*thk	y0	z0	x-axis
Cylinder 6	rC	3*thk	lX-2*thk	lY/2	zC	x-axis

3 Right-click **Geometry 1** and choose **Build All Objects**.



Work Plane 2 (wp2)

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 **yx-plane**.

4 In the **z-coordinate** text field, type `2*thk`.

5 Click **Go to Plane Geometry**.

Work Plane 2 (wp2)>Polygon 1 (pol1)

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

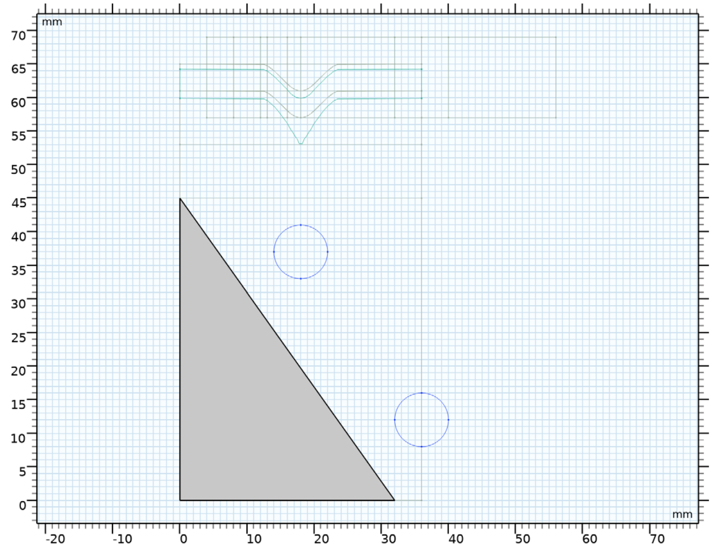
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **xw** text field, type `0 lY/2-bDia/2 lY/2-bDia/2 0 0 0`.

5 In the **yw** text field, type `0 0 0 lX-rOut-2*thk lX-rOut-2*thk 0`.

6 Click **Build Selected**.



Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
`3*thk`

4 Click **Build Selected**.

Difference 1 (dif1)

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

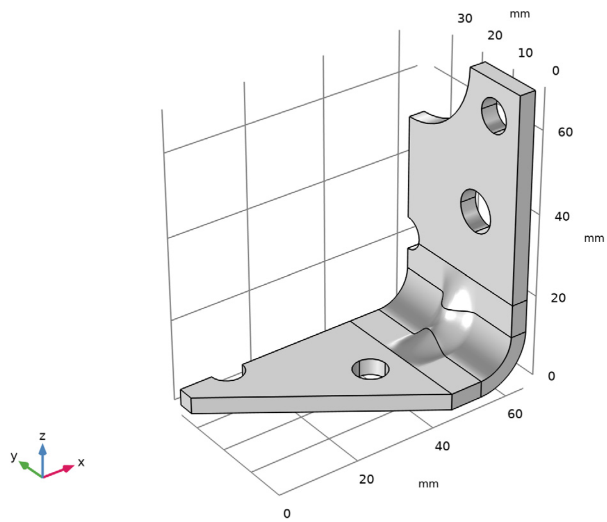
2 Select the objects **blk1**, **blk2**, **loft1**, **mir1**, and **rev1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **cyl1**, **cyl2**, **cyl3**, **cyl4**, **cyl5**, **cyl6**, and **ext1** only.

6 Click **Build All Objects**.



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

4 Locate the **Point on Plane of Reflection** section. In the **y** text field, type `lY/2`.

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

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

7 Select the object **dif1** only.

8 Click **Build All Objects**.

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