



Model created in COMSOL Multiphysics 6.4

Geometric Parameter 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.

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.

By using the LiveLink interface for Solid Edge 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 model demonstrates how to synchronize the geometry between Solid Edge and COMSOL Multiphysics while updating dimensional parameters, and how to perform an optimization study.

Note: This model requires the Optimization Module, the Structural Mechanics Module, and the LiveLink interface for Solid Edge.

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 $9.3 \cdot 10^{-4} \text{ kg} \cdot \text{m}^2$.

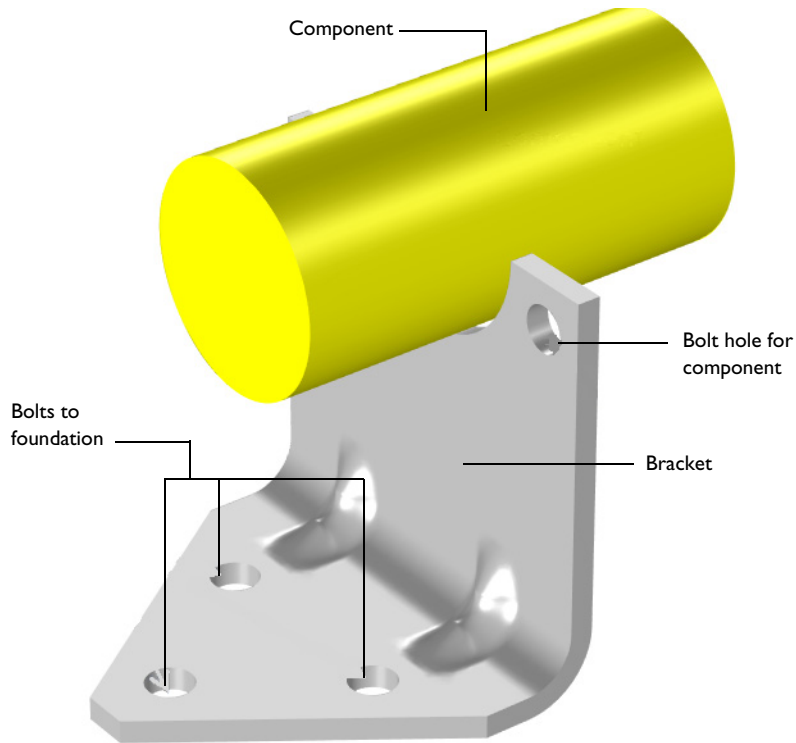


Figure 1: Bracket supporting a heavy component.

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
Z00	Vertical distance from the bend to the edge of the outer hole	8	30
Y00	Horizontal distance from the edge of the bracket to outer hole	3	29
WIND	Width of the indentation	8	20

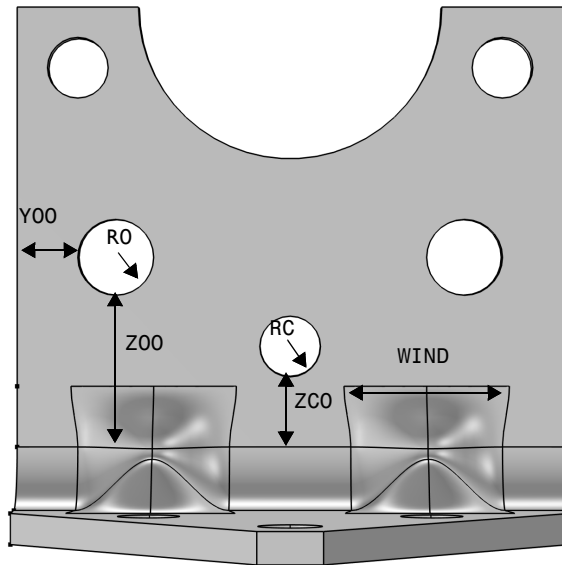


Figure 2: Optimization parameters.

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 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.
- 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](#).

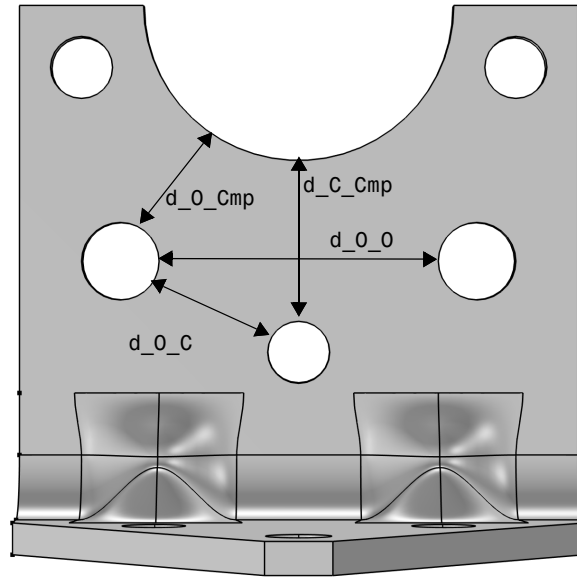


Figure 3: Geometrical constraints.

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 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.

Results and Discussion

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

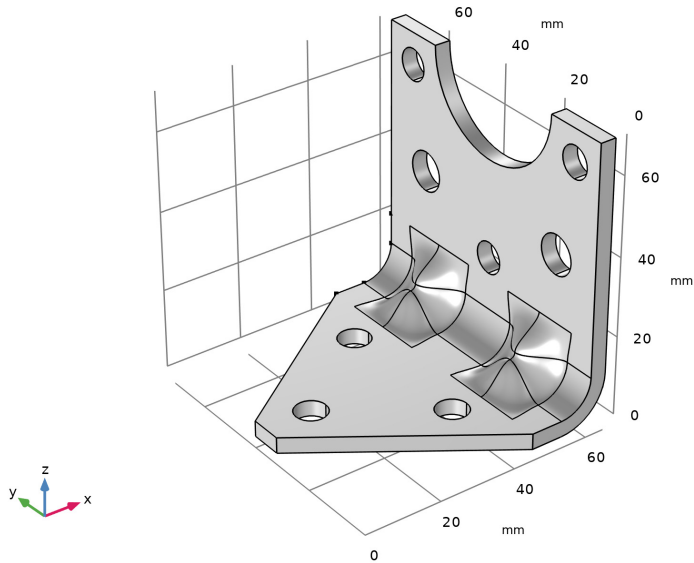


Figure 4: Initial geometry.

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)
LL_RC	6.94	3	15
LL_ZC0	6.43	1	23
LL_R0	10.39	3	15
LL_Y00	10.38	3	29
LL_Z00	17.69	8	30

The weight of the optimized bracket is about 181 g, a reduction of 20 g from the original 201 g. The stresses from the shock load on the optimized geometry are shown in [Figure 5](#)

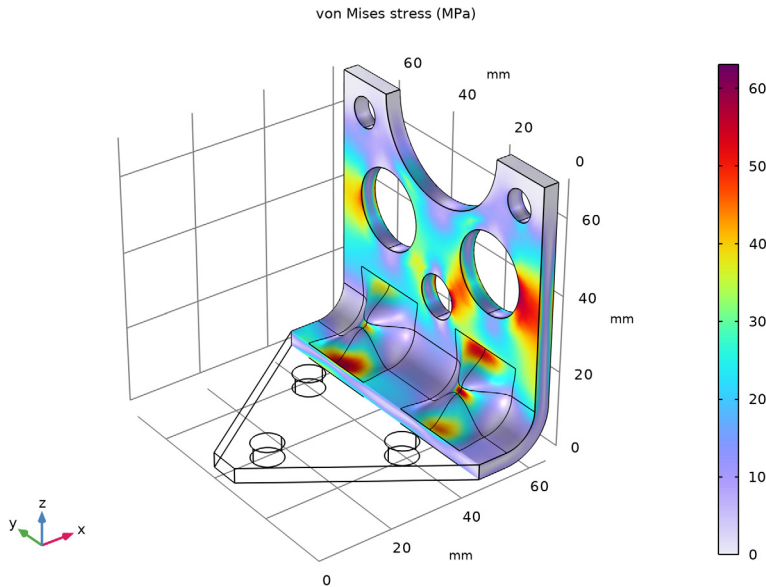


Figure 5: Stresses at peak load in the optimized design.

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 bracket geometry you are using in this model comes from Solid Edge design. Using LiveLink *for* Solid Edge you synchronize the geometry and parameters for the dimension of the bracket and the positioning of holes between Solid Edge and COMSOL

Multiphysics. In order for this to work you need to have both programs running during modeling, and you need to make sure that the bracket file is the active file in Solid Edge.

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: LiveLink_for_Solid_Edge/Tutorials,
_LiveLink_Interface/bracket_optimization_llse


Modeling Instructions

- 1 In Solid Edge open the file `bracket_optimization.par` located in the model's Application Library folder.
- 2 Switch to the COMSOL Desktop.




COMSOL DESKTOP

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**.

GEOMETRY I

Make sure that the CAD Import Module kernel is used.

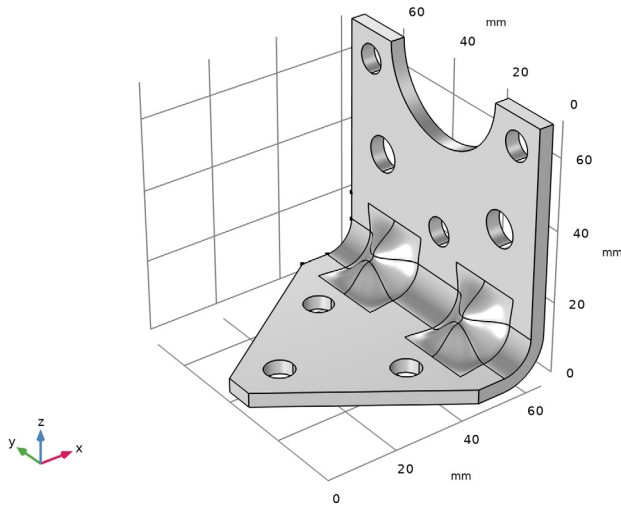
- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
- 3 From the **Geometry representation** list, choose **CAD kernel**.

LiveLink for Solid Edge 1 (cad1)

- 1 In the **Home** toolbar, click  **LiveLink** and choose **LiveLink for Solid Edge**.
- 2 In the **Settings** window for **LiveLink for Solid Edge**, locate the **Synchronize** section.


3 Click **Synchronize**.

After a few moments the geometry of the bracket appears in the **Graphics** window.



- 4** Click to expand the **Parameters in CAD Package** section. The table contains ten variables, THK, LX, LZ, DCMP, BDIA, RC, ZCO, RO, Y00 and Z00, which are part of the Solid Edge model. In Solid Edge, the **Parameter Selection** button on the **COMSOL Multiphysics** tab allows you to select and view variables for synchronization. These variables are retrieved, and appear in the **CAD name** column of the table. The corresponding entries in the **COMSOL name** column, LL_THK, LL_LX and so on, are global parameters in the COMSOL model. These are automatically generated during synchronization, and are assigned the values of the linked Solid Edge dimensions. The parameter values are displayed in the **COMSOL value** column.

Global parameters in a model allow you to parameterize settings and can be controlled by the optimization solver to perform parametric sweeps. Thus, by linking Solid Edge variables to COMSOL global parameters, the optimization solver can automatically update and synchronize the geometry for each new value in a sweep.

- 5** Click to expand the **Boundary Selections** section. The selections listed here are user defined selections saved in the Solid Edge files for the components that they appear on. In Solid Edge, you can set up selections using the **Selections** button on the **COMSOL Multiphysics** tab.
- 6** In the **Home** toolbar, click  **Build All**.



GLOBAL DEFINITIONS

Parameters I



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

The table already contains the automatically generated global parameters that are linked to the Solid Edge variables.

Based on the parameters in the table you can define expressions to constrain the positioning of the holes while optimizing the bracket. Later on you will set up the optimization solver to take into account these geometric constraints. Now, continue with loading the expressions for the geometric constraints and the parameters needed to define the physics. Since the parameter file contains all parameters, including the already synchronized ones, clear the table first to avoid duplicates.

- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Clear Table**.
- 4 Click  **Load from File**.
- 5 Browse to the model's Application Libraries folder and double-click the file `bracket_optimization_parameters.txt`.

ADD MATERIAL

- 1 In the **Materials** 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 right end of the **Add to Component** split button in the window toolbar.
- 5 From the menu, choose **Add to Component**.
- 6 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

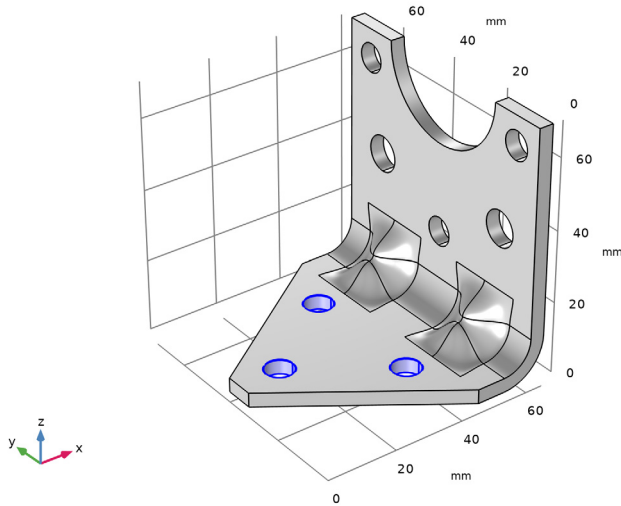
Fixed (Bolts)

- 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 Locate the **Boundary Selection** section. From the **Selection** list, choose **Fixed (Bolts)**.



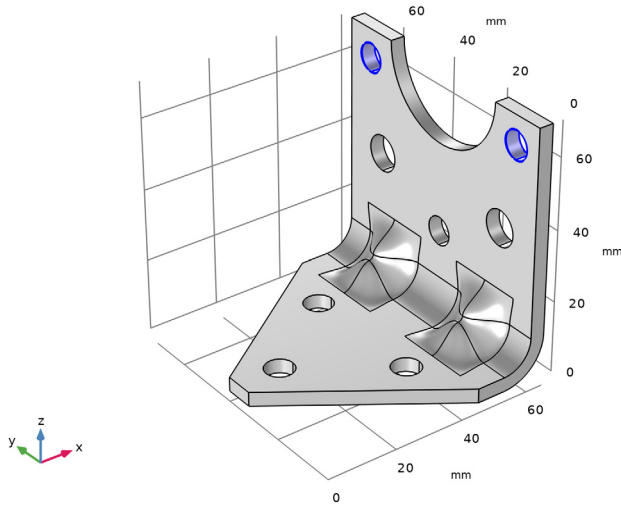
Rigid Connector (Mounted component)

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 Locate the **Boundary Selection** section. From the **Selection** list, choose **Rigid Connector (Mounted comp)**.



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

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

$LL_LX - LL_THK / 2$	x
$(4 * LL_BDIA + LL_DCMP) / 2$	y
LL_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

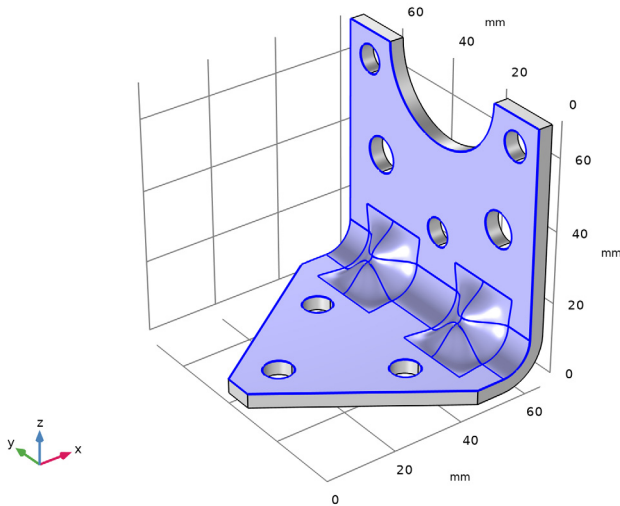
$IXCmp$	0	0
0	$IYZCmp$	0
0	0	$IYZCmp$

MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
- 3 From the **Element size** list, choose **Finer**.

Free Triangular 1

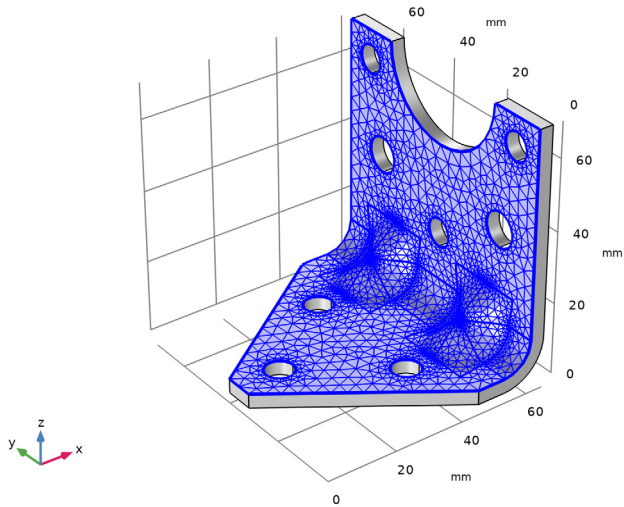
- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Boundary Mesh**.




Size 1

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section.
- 5 Select the **Minimum element size** checkbox. In the associated text field, type 0.2.
- 6 Select the **Curvature factor** checkbox. In the associated text field, type 0.38.

7 Click  **Build Selected.**



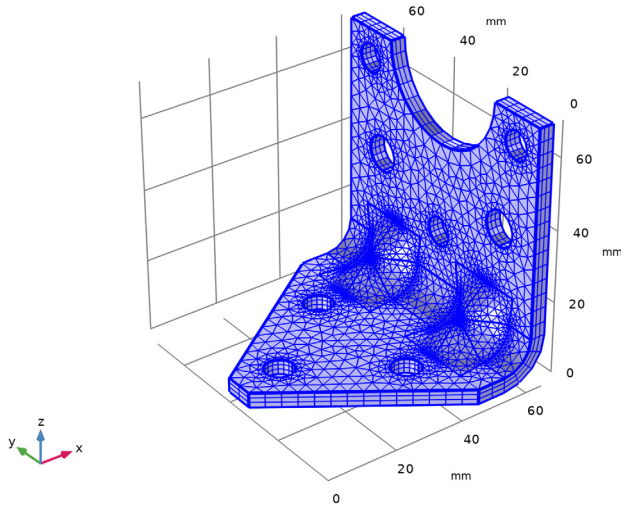
Swept 1

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 (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

$4 \cdot g_{\text{const}} \cdot \text{solid.rho}$	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

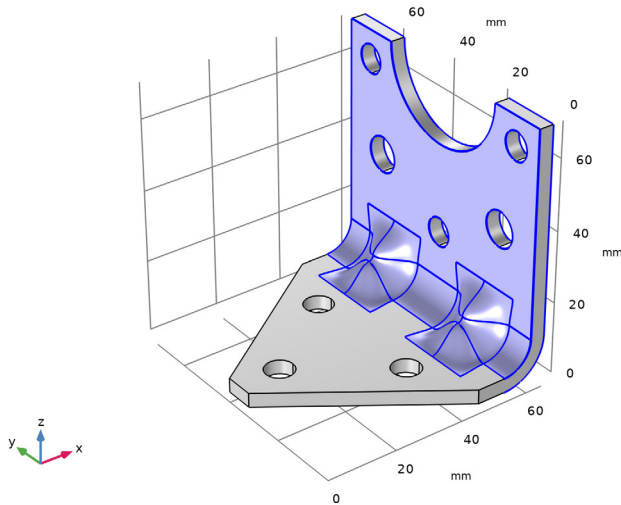
Prepare for the optimization by adding variables for the bracket mass and the maximum stress.

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**.

Boundary Probe 1 (bnd1)

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Boundary Probe**.
- 2 In the **Settings** window for **Boundary Probe**, type maxStress in the **Variable name** text field.
- 3 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.
The boundaries for which the maximum stress is going to be evaluated are defined as **Selections** in Solid Edge.
- 4 Locate the **Source Selection** section. From the **Selection** list, choose **Maximum Stress**.



- 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

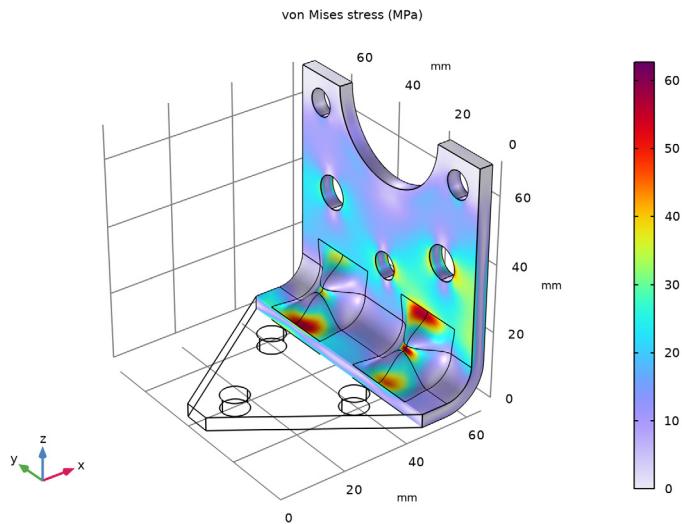
- 1 In the **Model Builder** window, expand the **Stress in Optimized Region** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.

Deformation

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

Filter 1


- 1 Right-click **Volume 1** and choose **Filter**.
- 2 In the **Settings** window for **Filter**, locate the **Element Selection** section.
- 3 In the **Logical expression for inclusion** text field, type $x > LL_LX - 5 * LL_THK$.
- 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 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 (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 `bracket_optimization_ctrlvars.txt`.

11 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound	Evaluate for
<code>real(freq)</code>	<code>minFreq</code>		Eigenfrequency
<code>comp1.maxStress/ maxStressLimit</code>		1	Stationary
<code>d_O_Cmp</code>	3 [mm]		Eigenfrequency
<code>d_C_Cmp</code>	3 [mm]		Eigenfrequency
<code>d_O_C</code>	3 [mm]		Eigenfrequency
<code>d_O_0</code>	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

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**.

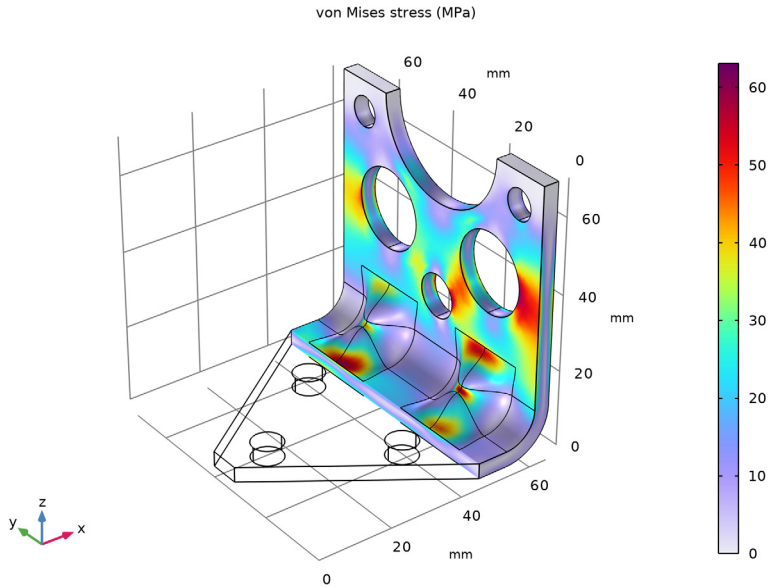
RESULTS

Stress in Optimized Region

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

Examine the stress distribution in the optimized configuration.

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.

1 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, synchronize the optimum values and rebuild the geometry.

GLOBAL DEFINITIONS

Parameters I

1 In the **Model Builder** window, expand the **Global Definitions > Parameters I** node, then click **Parameters I**.

2 In the **Settings** window for **Parameters**, click  **Case I**.

GEOMETRY I

- 1 In the **Geometry** toolbar, click  **Build All**.
- 2 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry I**.

