Created in COMSOL Multiphysics 5.6

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 Inventor 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 Inventor and COMSOL Multiphysics while updating dimensional parameters, and how to perform an optimization study.

Note: This application requires the Optimization Module, the Structural Mechanics Module, and the LiveLink interface for Inventor.

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.

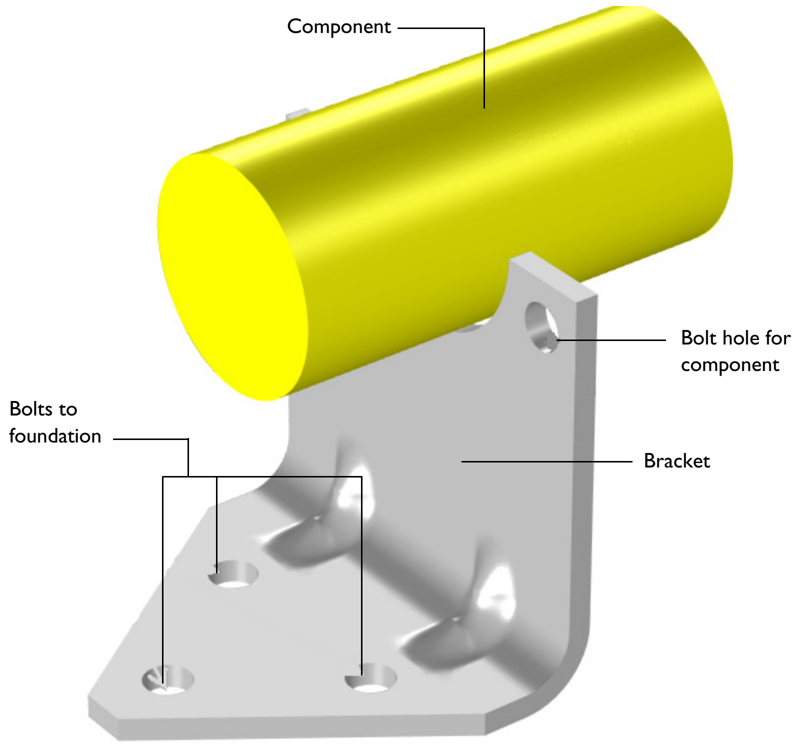


Figure 1: Bracket supporting a heavy component.

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
ZC0	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

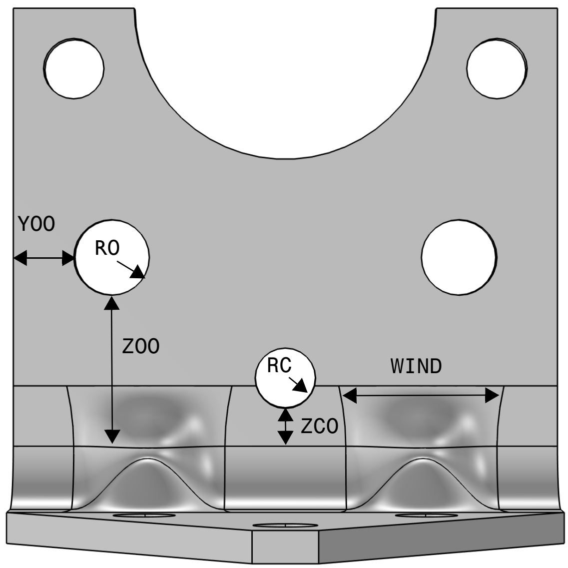


Figure 2: Optimization parameters.

CONSTRAINTS

- The lowest natural frequency must be at least 60 Hz.
- When exposed to a peak acceleration of 4*g* 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.

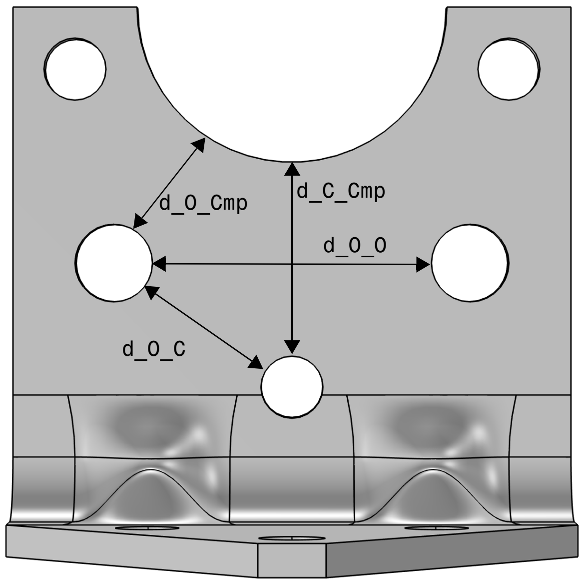


Figure 3: Geometrical constraints.

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 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.

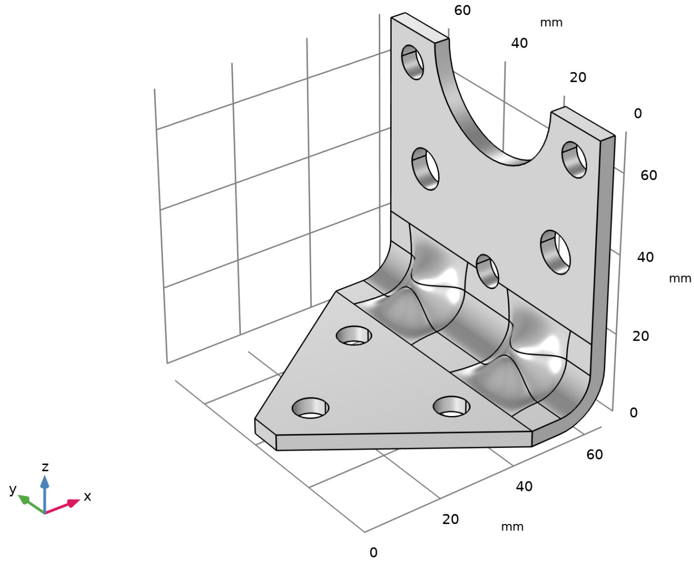


Figure 4: Initial geometry.

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)
LL_RC	7.34	3	15
LL_ZC0	1	1	23
LL_R0	10.76	3	15
LL_Y00	11.25	3	29
LL_Z00	16.37	8	30

The weight of the optimized bracket is about 179 g, a reduction of 21 g from the original 200.57 g. The stresses from the shock load on the optimized geometry are shown in Figure 5

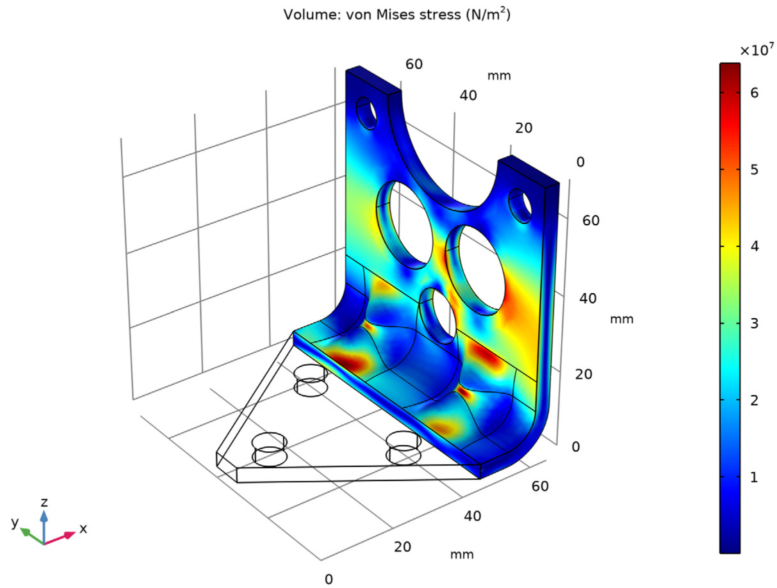


Figure 5: Stresses at peak load in the optimized design.

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 an Inventor design. Using LiveLink *for* Inventor you synchronize the geometry and parameters for the dimension of the bracket and the positioning of holes between Inventor 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 Inventor.

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_Inventor/Tutorials,` `_LiveLink_Interface/bracket_optimization_llinventor`

Modeling Instructions

You can set up this simulation both by working inside Inventor, using the embedded COMSOL simulation environment, and by working in the standalone COMSOL Desktop. Regardless which way you proceed, first you need to open the CAD file with the geometry in Inventor.

1. In Inventor open the file `bracket_optimization.ipt` located in the model's Application Library folder.
2. Switch to the COMSOL Desktop, and skip the next section. Or, continue below if you are working inside Inventor.

MODELING INSIDE INVENTOR

1. On the **COMSOL Multiphysics** tab click the **New** button.

   In case it is not already running, the COMSOL modeling environment will be started, and the geometry will be synchronized automatically.
2. Continue with step 2 under the Model Wizard section.

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 1

The geometry is already synchronized if you are modeling inside Inventor, and you can skip to step 4 in the section LiveLink for Inventor 1 (cad1).

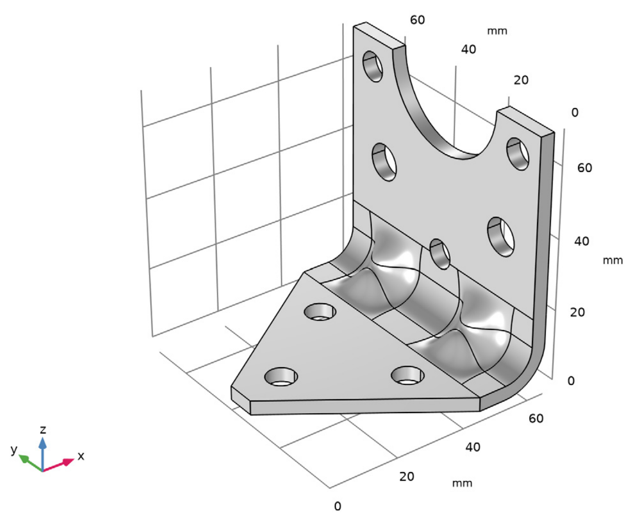
Make sure that the CAD Import Module kernel is used.

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 Inventor 1 (cad1)

1 Right-click **Component 1 (comp1)>Geometry 1** and choose **LiveLink Interfaces> LiveLink for Inventor**.
2 In the **Settings** window for **LiveLink for Inventor**, 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 dimensions, THK, LX, LZ, DCMP, BDIA, RC, ZC0, R0, Y00 and Z00, which are part of the Inventor model. In Inventor, the **Parameter Selection** button on the **COMSOL Multiphysics** tab allows you to select and view dimensions for synchronization. These dimensions 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 Inventor 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 Inventor dimensions 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 Inventor files for the components that they appear on. In Inventor, you can set-up selections using the **Selections** button on the **COMSOL Multiphysics** tab.
6. Right-click **LiveLink for Inventor 1 (cad1)** and choose **Build All**.

GLOBAL DEFINITIONS

Parameters 1

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

   The table already contains the automatically generated global parameters that are linked to the Inventor dimensions.

   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`.

MATERIALS

Add Material

From the **Home** menu, choose **Add Material**.

ADD MATERIAL

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

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

3 Click the right end of the **Add to Component** split button in the window toolbar.

4 From the menu, choose **Add to Component**.

5 From the **Home** menu, choose **Add Material**.

SOLID MECHANICS (SOLID)

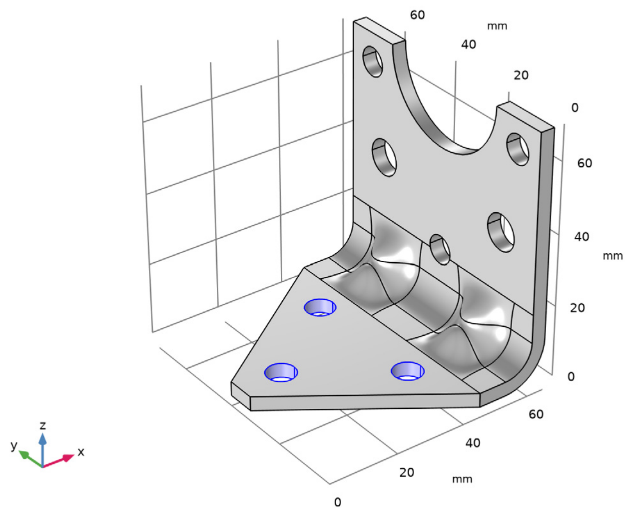
Fixed (Bolts)

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



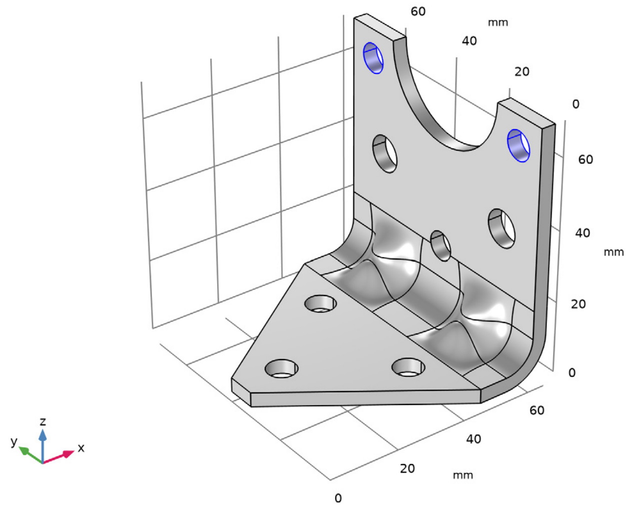
Rigid Connector (Mounted component)

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Connections>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 1

1 Right-click **Rigid Connector (Mounted component)** 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

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.

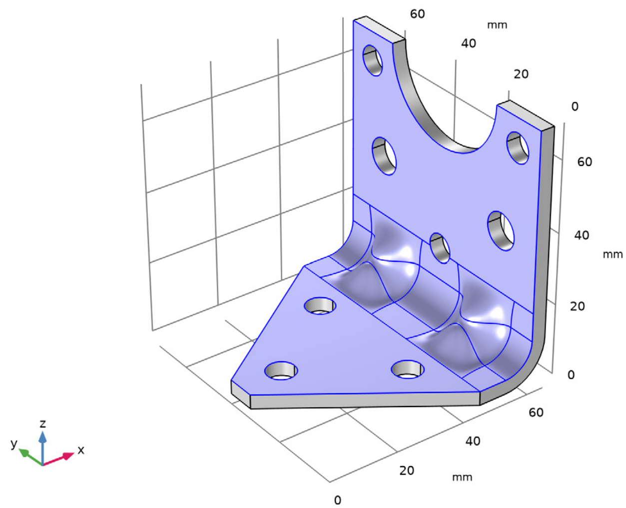
3 From the **Element size** list, choose **Finer**.

Free Triangular 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>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.

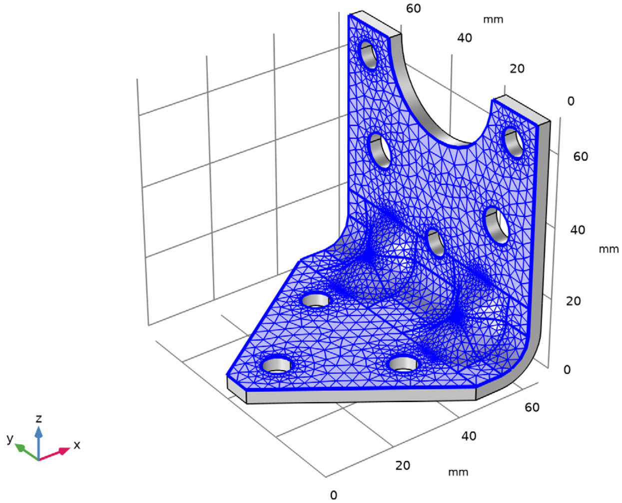
4 Locate the **Element Size Parameters** section. Select the **Minimum element size** check box.

5 In the associated text field, type `0.2`.

6 Select the **Curvature factor** check box.

7 In the associated text field, type `0.38`.

8 Click **Build Selected**.



Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

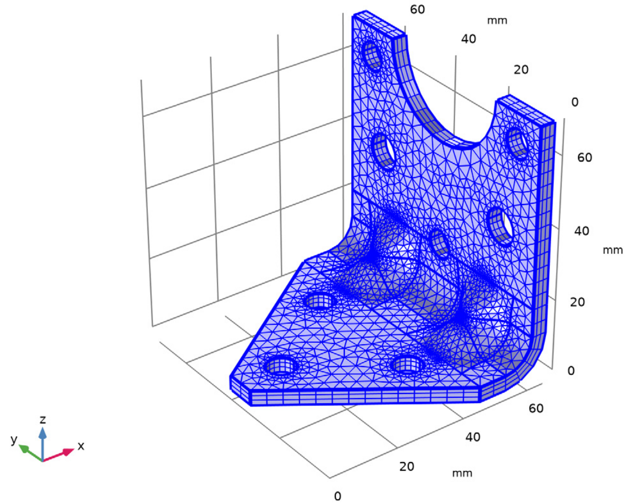
Distribution 1

1 In the **Model Builder** window, 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 Click **Compute**.

SOLID MECHANICS (SOLID)

Add the peak loads, and perform a stationary study.

Body load 4g on bracket

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>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

Force 4g on mounted component

1 In the **Model Builder** window, right-click **Rigid Connector (Mounted component)** 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

ROOT

From the **Home** menu, choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

4 From the **Home** menu, choose **Add Study**.

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 Click **= Compute**.

DEFINITIONS

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

Integration 1 (intop1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Nonlocal Couplings>Integration**.

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

3 From the **Selection** list, choose **All domains**.

Maximum 1 (maxop1)

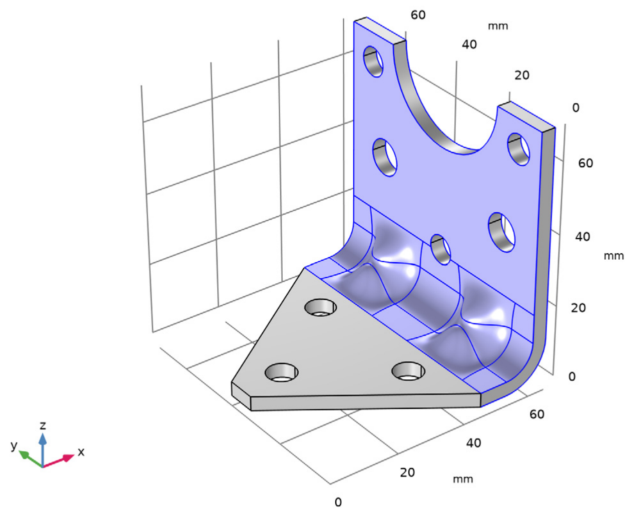
1 In the **Model Builder** window, right-click **Definitions** and choose **Nonlocal Couplings> Maximum**.

The boundaries for which the maximum stress is going to be evaluated are defined as **Selections** in Inventor.

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

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

4 From the **Selection** list, choose **Maximum Stress**.



Variables 1

1 Right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
mass	intop1(solid.rho)	kg	Bracket mass
maxStress	maxop1(solid.mises)	N/m²	Maximum stress

RESULTS

Add a plot for monitoring the geometry and stresses in the optimized region.

Stress in Optimized Region

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress in Optimized Region` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Stationary Study/Solution 2 (sol2)**.

Volume 1

1 Right-click **Stress in Optimized Region** and choose **Volume**.

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

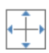
3 In the **Expression** text field, type `solid.mises`.

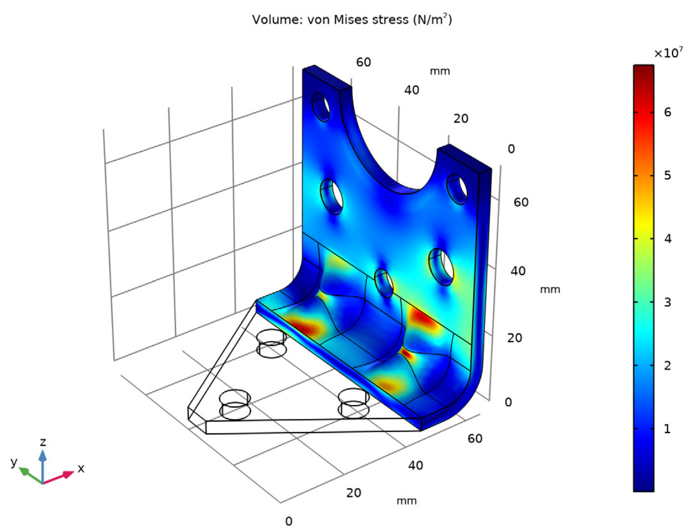
Filter 1

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.



Set up the optimization study.

ROOT

From the **Home** menu, choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.
2 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.
3 Click **Add Study** in the window toolbar.
4 From the **Home** menu, choose **Add Study**.

OPTIMIZATION STUDY

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

Optimization

Right-click **Optimization Study** and choose **Optimization>Optimization**.

Eigenfrequency

1 Right-click **Optimization Study** 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 Right-click **Optimization Study** 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**.

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 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
`comp1.mass`	`Bracket mass`	Stationary

7 Locate the **Optimization Solver** section. In the **Optimality tolerance** text field, type `0.1`.

The first eigenfrequency is to be used in the optimization.

8 Locate the **Objective Function** section. 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
`real(comp1.solid.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**.

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 **Model Builder** window, right-click **Optimization Study** and choose **Show Default Solver**.

Run the optimization.

2 Right-click **Optimization Study** and choose **Compute**.

RESULTS

On the last line of **Objective Table 2** 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 6** 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.

To avoid scrolling through a large number of table entries, insert a **Global Evaluation** which shows only the optimized parameters.

1 In the table, enter the following settings:

Name	Description
LL_THK	Plate thickness
LL_LX	Bracket length
LL_LZ	Bracket height
LL_DCMP	Diameter of component
LL_BDIA	Diameter of mounting bolts
LL_RC	Radius of central hole
LL_ZCO	Distance from bend to bottom of central hole
LL_RO	Radius of outer holes
LL_YOO	Distance from edge to outer hole
LL_ZOO	Distance from bend to bottom of outer hole

Global Evaluation 1

1 In the **Model Builder** window, expand the **Results>Tables** node.

2 Right-click **Results>Derived Values** and choose **Global Evaluation**.

Instead of adding each parameter manually load a txt file

3 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

4 From the **Dataset** list, choose **Optimization Study/Solution 3 (sol3)**.

5 Locate the **Expressions** section. Click **Load from File**.

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

7 Click next to **Evaluate**, then choose **New Table**.

TABLE

Go to the **Table** window.

RESULTS

Global Evaluation 1

1 In the **Model Builder** window, click **Global Evaluation 1**.

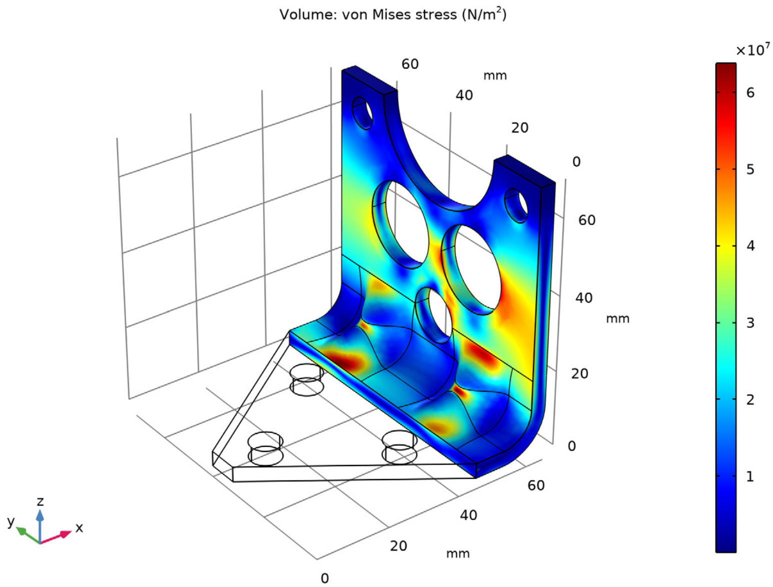
2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

Examine the stress distribution in the optimized configuration.

Stress in Optimized Region

1 In the **Model Builder** window, right-click **Stress in Optimized Region** and choose **Plot**.

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



As a last step, update the parameters in Inventor.

GEOMETRY 1

LiveLink for Inventor 1 (cad1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **LiveLink for Inventor 1 (cad1)**.

2 In the **Settings** window for **LiveLink for Inventor**, locate the **Parameters in CAD Package** section.

3 Click **Update Parameters from CAD** in the upper-right corner of the section.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build All**.