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. It 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 model requires the Structural Mechanics Module, the CAD 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 $7.1 \cdot 10^{-4}$ kgm^2, and the moment of inertia around the two transverse axes is $9.3 \cdot 10^{-4}$ kgm^2.

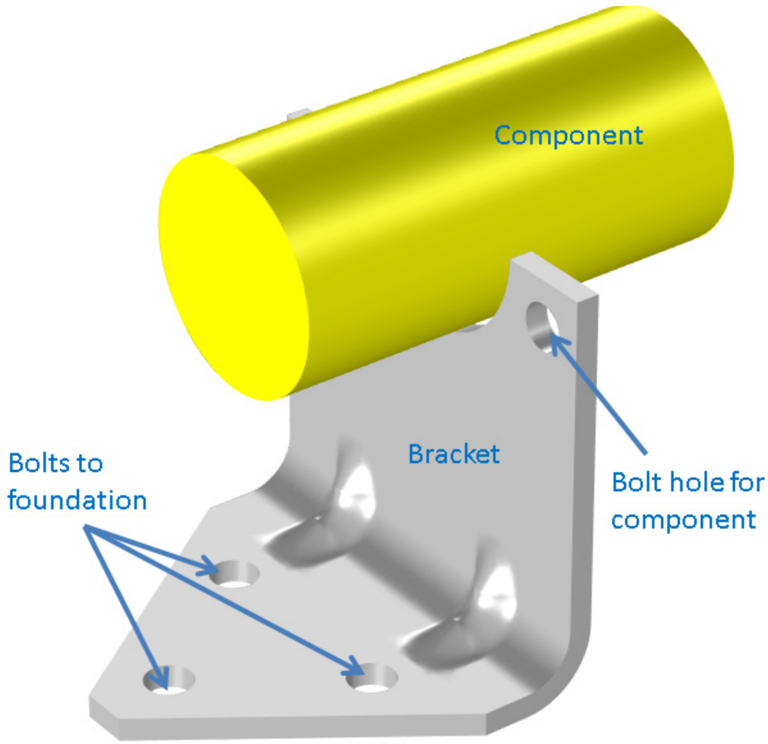

Figure 1: Bracket supporting a heavy component

The idea is to reduce the weigh by drilling holes in the vertical surface of the bracket, and at the same time changing 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
rO	Radius of the outer hole	3	15
zOo	Vertical distance from the bend to the edge of the outer hole	3	29
yOo	horizontal distance from the edge of the bracket to outer hole	8	30
$wInd$	Width of the indentation	8	20

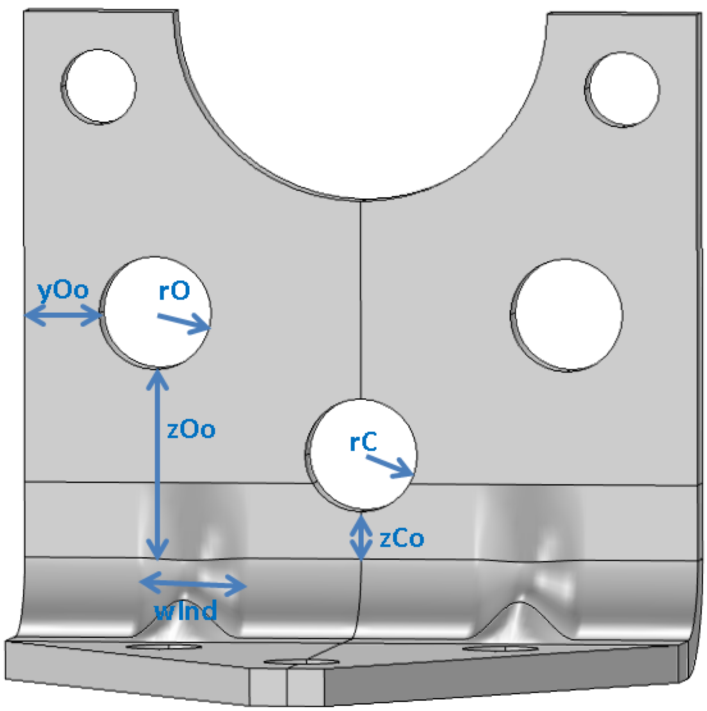

Figure 2: Optimization parameters

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 non-differentiable, since the location of the peak stress can jump from one place o 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.

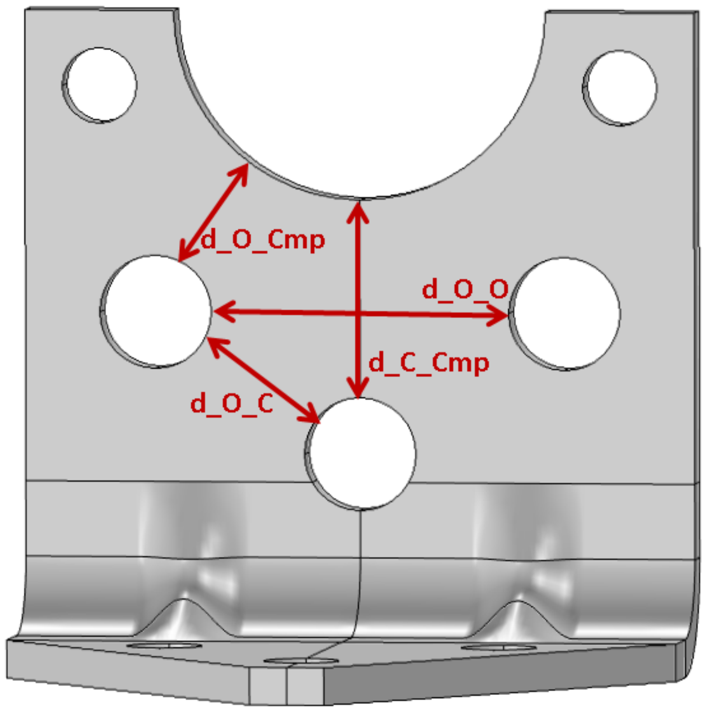

Figure 3: Geometrical constraints

Results and Discussion

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

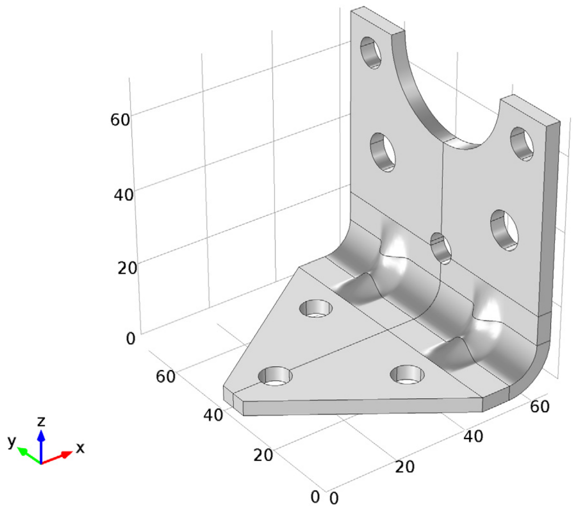

Figure 4: Initial geometry

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	9.3	3	15
zCo	4	1	23
rO	10	3	15
zOo	9.8	3	29
yOo	18.7	8	30
wInd	20	8	20

The weight of the optimized bracket is about 180 g, a reduction of 25 g from the original 205 g. The stresses from the shock load on the optimized geometry are shown in Figure 5

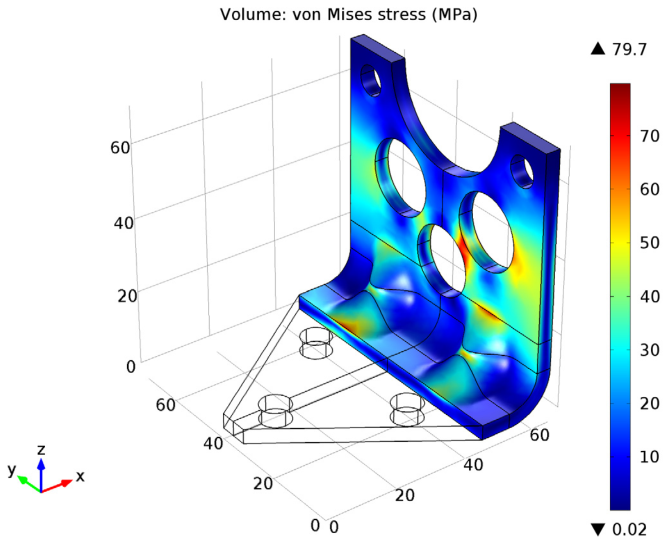

Figure 5: Stresses at peak load in the optimized design.

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

There are several possible arrangements of the holes that will give the same weight reduction within a small tolerance. It is thus 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.

Model Library path: `Optimization_Module/Shape_Optimization/multistudy_bracket_optimization`

Modeling Instructions

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

NEW

1 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 **Preset Studies>Eigenfrequency**.
6 Click **Done**.

DEFINITIONS

Parameters

1 On the **Model** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Model Library folder and double-click the file `multistudy_bracket_optimization_params.txt`.

ROOT

1 In the **Model Builder** window, click **Untitled.mph (root)**.
2 In the **Settings** window for Root, locate the **Unit System** section.
3 From the **Unit system** list, choose **MPa**.

Start building the geometry.

GEOMETRY 1

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 Import Module kernel**.
4 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

1 On 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`.

Parametric Curve 1 (pc1)

1 On the **Geometry** 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 `dInd/2*(1+cos(pi*lY/wInd*(s-0.5)))*(abs(s-0.5)<wInd/lY)`.

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

Plane Geometry

Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Parametric Curve 1 (pc1)** and choose **Build Selected**.

Copy 1 (copy1)

1 On 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`.

Bézier Polygon 1 (b1)

1 On the **Work plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **2**, set **xw** to `thk`.

Copy 2 (copy2)

1 On the **Work plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **b1** only.

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

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

Convert to Solid 1 (csol1)

On the **Work plane** toolbar, click **Conversions** and choose **Convert to Solid**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** right-click **Convert to Solid 1 (csol1)** and choose **Build Selected**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.
2 In the **Settings** window for Revolve, locate the **Revolution Angles** section.
3 In the **End angle** text field, type `90`.
4 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.
5 Find the **Point on the revolution axis** subsection. In the **x** text field, type `lX-rOut`.
6 In the **z** text field, type `rOut`.
7 Find the **Direction of revolution axis** subsection. In the **y** text field, type `-1`.
8 Click the **Build All Objects** button.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for Block, locate the **Size** 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 the **Build All Objects** button.

Loft 1 (loft1)

1 On 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. Locate the **Start Profile** section. Find the **Start profile** subsection. Select the **Active** toggle button.
5 On the object **rev1**, select Boundary 1 only.
6 Click to expand the **End profile** section. Locate the **End Profile** section. Find the **End profile** subsection. Select the **Active** toggle button.
7 On the object **blk1**, select Boundary 5 only.
8 Click the **Build All Objects** button.

Mirror 1 (mir1)

1 On 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 the **Build All Objects** button.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for Block, locate the **Size** 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 the **Build All Objects** button.
9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `dCmp/2`.
4 In the **Height** text field, type `3*thk`.
5 Locate the **Position** section. In the **x** text field, type `lX-2*thk`.
6 In the **y** text field, type `lY/2`.
7 In the **z** text field, type `lZ`.
8 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.
9 Click the **Build All Objects** button.

Cylinder 2 (cyl2)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `bDia/2`.

4 Locate the **Position** section. In the **y** text field, type `bDia`.

5 In the **z** text field, type `lZ-bDia`.

6 Click the **Build All Objects** button.

Cylinder 3 (cyl3)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 2 (cyl2)** and choose **Duplicate**.

2 In the **Settings** window for Cylinder, locate the **Position** section.

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

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

5 In the **z** text field, type `-thk`.

6 Locate the **Axis** section. From the **Axis type** list, choose **z-axis**.

7 Click the **Build All Objects** button.

Cylinder 4 (cyl4)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 3 (cyl3)** and choose **Duplicate**.

2 In the **Settings** window for Cylinder, locate the **Position** section.

3 In the **x** text field, type `1.5*bDia`.

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

5 Click the **Build All Objects** button.

Cylinder 5 (cyl5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `r0`.

4 Locate the **Position** section. In the **y** text field, type `y0`.

5 In the **z** text field, type `z0`.

6 Click the **Build All Objects** button.

Cylinder 6 (cyl6)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 5 (cyl5)** and choose **Duplicate**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `rC`.
4 Locate the **Position** section. In the **y** text field, type `lY/2`.
5 In the **z** text field, type `zC`.
6 Click the **Build All Objects** button.

Work Plane 2 (wp2)

1 On 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`.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.
2 Click the **Zoom Extents** button on the **Graphics** toolbar.
3 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.
4 Find the **Added segments** subsection. Click **Add Linear**.
5 Find the **Control points** subsection. In row **2**, set **xw** to `lY/2-bDia/2`.
6 Find the **Added segments** subsection. Click **Add Linear**.
7 Find the **Control points** subsection. In row **2**, set **yw** to `lX-rOut-2*thk`.
8 In row **2**, set **xw** to `0`.
9 Find the **Added segments** subsection. Click **Add Linear**.
10 Find the **Control points** subsection. Click **Close Curve**.

Work Plane 2 (wp2)

Right-click **Component 1 (comp1)>Geometry 1>Work Plane 2 (wp2)>Plane Geometry>Bézier Polygon 1 (b1)** and choose **Build Selected**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.
2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (mm)
3*thk

4 Right-click **Component 1 (comp1)>Geometry 1>Extrude 1 (ext1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the objects **rev1**, **blk1**, **mir1**, **blk2**, and **loft1** only.
3 In the **Settings** window for Difference, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the objects **cyl5**, **cyl6**, **cyl3**, **cyl2**, **ext1**, **cyl1**, and **cyl4** only.
6 Click the **Build All Objects** button.

Mirror 2 (mir2)

1 On 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 Click the **Build All Objects** button.
8 Click the **Zoom Extents** button on the **Graphics** toolbar.

MATERIALS

On the **Model** toolbar, click **More Windows** and choose **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>Structural steel**.
3 Click **Add to Component** in the window toolbar.
4 On the **Model** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

On 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.

Fixed Constraint 1

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

2 Select Boundaries 10–13 and 15–22 only.

3 On 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.

Rigid Connector 1

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

2 Select Boundaries 48, 49, 52, 53, and 75–78 only.

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

4 Specify the $\mathbf{X}_c$ vector as

`lX-thk/2`	X
`lY/2`	Y
`lZ`	Z

Mass and Moment of Inertia 1

1 On 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 **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
2. Select Boundaries 4, 8, 26, 30, 35, 39, 41, 44, 60, and 64 only.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for Size, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.
4. Click the **Build All** button.

Swept 1

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

Distribution 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 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 `2`.
4. Click the **Build All** button.

Run an eigenfrequency study on the initial geometry.

STUDY 1

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. On the **Model** toolbar, click **Compute**.

Add the peak loads, and a perform a stationary study.

SOLID MECHANICS (SOLID)

Body Load 1

1. On 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)

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Rigid Connector (Mounted component)** and choose **Applied Force**.

Applied Force 1

1 In the **Settings** window for Applied Force, type `Force 4g on mounted component` in the **Label** text field.

2 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

On the **Model** toolbar, click **More Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select study** tree, select **Preset Studies>Stationary**.

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

4 On the **Model** toolbar, click **More Windows** and choose **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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 On the **Model** toolbar, click **Compute**.

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

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **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 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.

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

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

4 Select Boundaries 25, 26, 29, 30, 34, 35, 38, 39, 41, 44, 60, 64, and 81–84 only.

Variables 1a

1 In the **Model Builder** window, 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)`	t	`Bracket mass`
`maxStress`	`maxop1(solid.mises)`	MPa	`Maximum stress`

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

RESULTS

3D Plot Group 3

1 On the **Model** toolbar, click **Add Plot Group** 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 **Data set** list, choose **Stationary study/Solution 2**.

4 Right-click **Results>3D Plot Group 3** and choose **Volume**.

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

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

7 Right-click **Results>3D Plot Group 3>Volume 1** and choose **Filter**.

8 In the **Settings** window for Filter, locate the **Element Selection** section.

9 In the **Logical expression for inclusion** text field, type `dom>2`.

10 Click the **Zoom Extents** button on the **Graphics** toolbar.

Set up the optimization study.

ROOT

On the **Model** toolbar, click **More Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select study** tree, select **Custom Studies>Empty Study**.

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

4 On the **Model** toolbar, click **More Windows** and choose **Add Study** to close the **Add Study** window.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

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

3 On the **Study** toolbar, click **Optimization**.

OPTIMIZATION

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

2 In the **Model Builder** window, right-click **Optimization** and choose **Study Reference**.

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

4 Locate the **Study Reference** section. From the **Study reference** list, choose **Eigenfrequency study**.

5 Right-click **Optimization** and choose **Study Reference**.

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

7 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 **BOBYQA**.

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

Expression	Description	Evaluate for
`comp1.mass`	`Bracket mass`	Stationary {ref2}

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

The first eigenfrequency is to be used in the optimization.

6 Locate the **Objective Function** section. From the **Solution** list, choose **Use first**.

7 Locate the **Control Variables and Parameters** section. Click **Load from File**.

8 Browse to the model's Model Library folder and double-click the file `multistudy_bracket_optimization_ctrlvars.txt`.

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

Expression	Lower bound	Upper bound	Evaluate for
`real(comp1.solid.freq)`	`minFreq`		Eigenfrequency {ref}
`comp1.maxStress`		`maxStress`	Stationary {ref2}
`d_O_Cmp`	3		Eigenfrequency {ref}
`d_C_Cmp`	3		Eigenfrequency {ref}
`d_O_C`	3		Eigenfrequency {ref}
`d_O_O`	3		Eigenfrequency {ref}

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

11 From the **Plot group** list, choose **3D Plot Group 3**.

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

Parametric 1

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

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

3 Clear the **Stop if error** check box.

Parametric 2

1 In the **Model Builder** window, under **Optimization>Job Configurations** click **Parametric 2**.

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

3 Clear the **Stop if error** check box.

Parametric 3

1 In the **Model Builder** window, under **Optimization>Job Configurations** click **Parametric 3**.

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

3 Clear the **Stop if error** check box.

Run the optimization.

4 On the **Study** toolbar, click **Compute**.

RESULTS

On the last line of **Objective Table 2**, you will find the optimal set of parameters and the minimum weight.

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.

Examine the stress distribution in the optimized configuration.

3D Plot Group 3

On the **3D plot group** toolbar, click **Plot**.