Model created in COMSOL Multiphysics 6.4

Bracket — Eigenfrequency Shape Optimization

Introduction

Shape optimization in a structural mechanics context can be used with different objectives, but this model focuses on the case of eigenfrequency optimization. The model demonstrates how to identify the shape deformation so that the lowest eigenfrequency is maximized. Such investigations can occur during the concept design or at later stages.

Manufacturing constraints might prevent all of the performance improvements gained by the shape optimization, but in many cases the deformed design can be used to tweak the design and/or the manufacturing process to achieve improved performance. Having some idea about the magnitude of the potential performance improvements can also be useful when deciding whether to change the manufacturing tool/process.

The various examples based on a bracket geometry form a suite of tutorials that summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module. In addition, this example requires the Optimization Module.

Model Definition

The model shows how to determine the optimal shape deformation of a bracket geometry. The bracket is symmetric about the plane $x = 0$ and is made of a linear elastic material, structural steel. The optimization preserves the symmetry of the design using the **Mirror Symmetry** feature and the **Free Shape Boundary** feature supports regularization by

- imposing a maximum displacement for the shape deformation
- applying a filter length, effectively limiting the slope of the deformation to 2
- restricting the shape deformation to occur in the normal direction

The thickness of the metal sheet is preserved by copying the shape deformation using a **General Extrusion** operator and a **Prescribed Deformation** feature.

The bracket is optimized with respect to the lowest natural frequency, but the eigenmode with the lowest frequency can change due to shape change, and therefore the first six eigenfrequencies are computed in every optimization iteration. The MMA optimization algorithm is then used to maximize the lowest eigenfrequency, but the algorithm considers all eigenfrequencies in every iteration.

Results

The result of the optimization is shown in Figure 1. The optimization increases the moment of inertia around the *z*-axis by making a bulge in the *x* direction. This results in an eigenfrequency that is twice as high as the initial value.

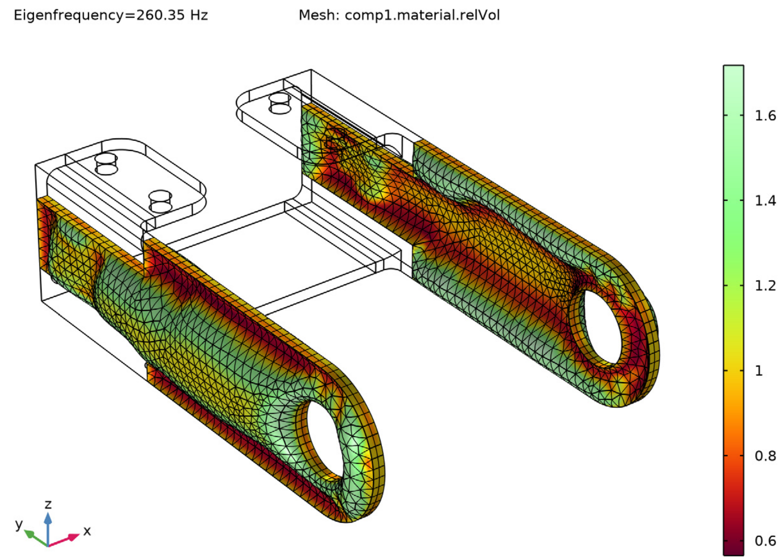

Figure 1: The mesh quality is plotted after the shape optimization has deformed the elements.

Notes About the COMSOL Implementation

The filter equation introduced by the **Free Shape Boundary** feature requires a **Stationary** solver. Computation of the eigenfrequency requires an **Eigenvalue Solver**, but gradient-based optimization is only supported over a single study step. To circumvent this limitation, one has to use the **Stationary Then Eigenfrequency** study step. This study step is designed specifically for optimization, and it creates both solvers such that the one-way coupling between them is accounted for in the computation of the gradient.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_eigenfrequency_shape_optimization`

Modeling Instructions

This example starts from an existing model from the Structural Mechanics Module Application Library.

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

Browse to the model's Application Libraries folder and double-click the file `bracket_eigenfrequency.mph`.

COMPONENT 1 (COMP1)

Free Shape Domain 1

1 In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Free Shape Domain**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

Free Shape Boundary 1

1 In the **Shape Optimization** toolbar, click **Free Shape Boundary**.

2 Select Boundary 19 only.

3 In the **Settings** window for **Free Shape Boundary**, locate the **Control Variable Settings** section.

4 From the d_{max} list, choose **User defined**.

5 In the table, enter the following settings:

	Lock	Lower bound (m)	Upper bound (m)
X		-2[cm]	2[cm]
Y	√	-0.02	0.02
Z	√	-0.02	0.02

6 Locate the **Filtering** section. From the R_{min} list, choose **User defined**.

7 In the text field, type `1[cm]`.

Preserve the symmetry of the bracket using a **Mirror Symmetry** feature.

Mirror Symmetry 1

1 In the **Shape Optimization** toolbar, click **Mirror Symmetry**.

2 Select Domain 9 only.

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

4 From the **p** list, choose **User defined**.

5 From the **n** list, choose **User defined**.

DEFINITIONS

Preserve the thickness of the bracket by copying the deformation to the other side of the bracket arm using a **General Extrusion** operator and a **Prescribed Deformation** feature.

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.

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

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

4 Select Boundary 19 only.

5 Locate the **Destination Map** section. In the **x-expression** text field, type `10[cm]*sign(Xg)`.

6 In the **y-expression** text field, type `Yg`.

7 In the **z-expression** text field, type `Zg`.

8 Locate the **Source** section. From the **Source frame** list, choose **Geometry (Xg, Yg, Zg)**.

COMPONENT 1 (COMP1)

Prescribed Deformation 1

1 In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Deformed Geometry** > **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 1 only.

5 Locate the **Prescribed Deformation** section. Specify the *dx* vector as

`genext1(material.dX)`	X
`genext1(material.dY)`	Y
`genext1(material.dZ)`	Z

The **Boundary Load** feature is not used for eigenfrequency analysis, so it can be deleted.

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics (solid)** node.
2 Right-click **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Boundary Load 1** and choose **Delete**.

MESH 1

Increase the mesh resolution on the selection of the **Free Shape Boundary** to resolve the shape deformation better.

Edge 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Mesh 1** node.
2 Right-click **Component 1 (comp1)** > **Mesh 1** > **Edge 1** and choose **Build Selected**.

Free Triangular 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Mesh 1** > **Free Tetrahedral 1** node.
2 Right-click **Mesh 1** and choose **More Generators** > **Free Triangular**.
3 Select Boundaries 1 and 72 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 **Extremely fine**.

Free Tetrahedral 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** right-click **Free Tetrahedral 1** and choose **Build All**.

STUDY 2

In the **Model Builder** window, expand the **Study 2** node.

Solver Configurations

In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** node.

Solution 2 (sol2), Step 1: Stationary, Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 2**, Ctrl-click to select **Step 1: Stationary**, **Step 2: Eigenfrequency**, and **Solver Configurations** > **Solution 2 (sol2)**.

2 Right-click and choose **Delete**.

Step 1: Stationary Then Eigenfrequency

In the **Study** toolbar, click **More Study Steps** and choose **Eigenfrequency** > **Stationary Then Eigenfrequency**.

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **MMA**.

4 In the **Maximum number of iterations** text field, type `25`.

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

Expression	Description
`real(freq)`	

6 From the **Type** list, choose **Maximization**.

7 From the **Solution** list, choose **Minimum of objectives**.

8 Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Initial solution based**.

STUDY 1: INITIAL DESIGN

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

2 In the **Settings** window for **Study**, type `Study 1: Initial Design` in the **Label** text field.

STUDY 2: SHAPE OPTIMIZATION

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

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

3 Locate the **Study Settings** section. Select the **Generate default plots** checkbox.

4 In the **Study** toolbar, click **Get Initial Value**.

STUDY 2: SHAPE OPTIMIZATION

Shape Optimization

1 In the **Model Builder** window, under **Study 2: Shape Optimization** click **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, click to expand the **Output** section.

3 Select the **Plot** checkbox.

4 In the table, enter the following settings:

Plot group	**Plot window**
Shape Optimization	Graphics

5 In the **Home** toolbar, click **= Compute**.