Created in COMSOL Multiphysics 5.6

Shape Optimization of an MBB Beam

Introduction

Shape optimization can be used to alter the geometry of an existing product to improve its performance, but it can also be used as a postprocessing step for topology optimization. This model takes the design found in the model Topology Optimization of an MBB Beam and uses the **Free Shape Boundary** feature to further improve the stiffness without increasing the mass. The magnitude of the deformations in shape optimization can sometimes be limited by inverted elements. However, this is less of an issue with topology-optimized designs, because the design is already near optimality.

Model Definition

The result of topology optimization can be converted to a geometry by taking the 0.5 contour of the *filtered material volume factor* or the *projected material volume factor*. In theory the two are identical, but in practice the fields are approximated with a linear representation on a refined mesh, and therefore it can be preferable to use the contour of the *filtered material volume factor*. This means that it is the contour of the field on the left in Figure 1 that will be the starting point of the model.

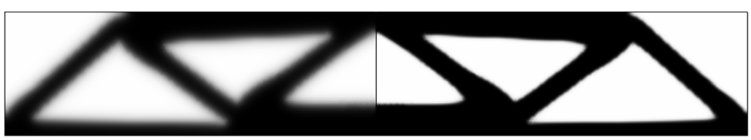

Figure 1: The result of Topology Optimization of an MBB Beam is shown with the filtered material volume factor to the left and the projected material volume factor to the right.

The model uses the **Free Shape Symmetry** feature to prevent the design from going outside the box of the original topology optimization problem.

Results and Discussion

The Helmholtz filter used by the **Density Model** in Topology Optimization of an MBB Beam is known to cause 90-degree angles on the boundaries of the design domain. The shape optimization changes this, because it is a detail that is not optimal, but it also re-positions the triangular holes as seen in Figure 2.

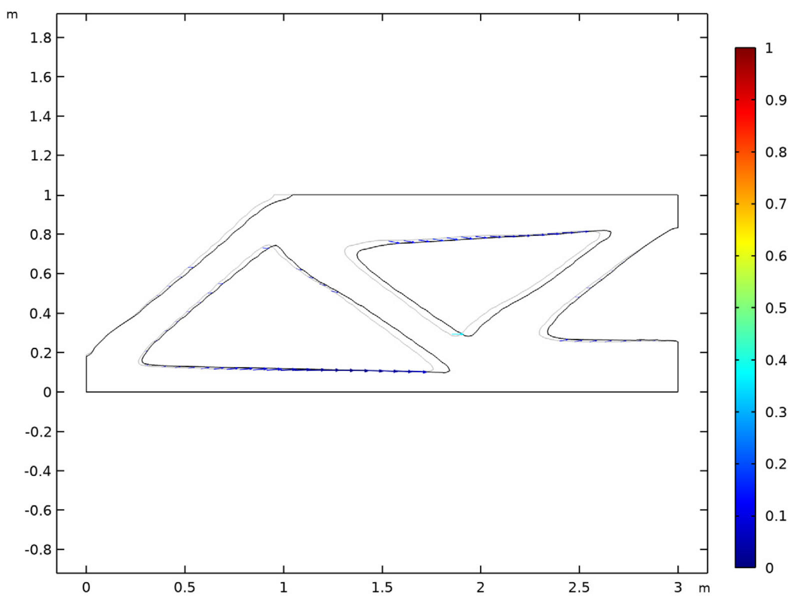

Figure 2: The default shape optimization plot shows the edges of the old and new geometry in gray and black, respectively. An arrow plot of the actual displacement is colored with the relative normal boundary displacement. The colors of the arrows thus indicate that the shape deformation is not limited by the maximum displacement.

Notes About the COMSOL Implementation

This model combines the Optimization and Solid Mechanics interfaces. The topology-optimized design can be extracted in several ways, but it is best to combine a **Filter** dataset, a **Mesh Part**, and an **Import** geometry feature because this allows recycling of the selections.

Application Library path: `Optimization_Module/Shape_Optimization/mbb_beam_shape_optimization`

Modeling Instructions

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

From the Application Libraries root, browse to the folder `Optimization_Module/Topology_Optimization` and double-click the file `mbb_beam_optimization.mph`.

RESULTS

Filter 1

1 In the **Results** toolbar, click **More Datasets** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Data** section.

3 From the **Dataset** list, choose **Optimization/Parametric Solutions 1 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `dtopo1.theta_f`.

5 Locate the **Filter** section. In the **Lower bound** text field, type `0.5`.

6 In the **Results** toolbar, click **Attributes** and choose **Create Mesh Part**.

MESH PART 1

1 In the **Model Builder** window, under **Global Definitions>Mesh Parts** right-click **Mesh Part 1** and choose **Build All**.

2 Right-click **Global Definitions>Mesh Parts>Mesh Part 1** and choose **Create Geometry from Mesh**.

GEOMETRY 2

In the **Home** toolbar, click **Build All**.

Moving Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Moving Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `a*0.001`.

5 In the **x maximum** text field, type `a*0.999`.

6 In the **y minimum** text field, type `b*0.001`.

7 In the **y maximum** text field, type `b*0.999`.

Roller Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Complement Selection**.

2 In the **Settings** window for **Complement Selection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click + **Add**.

5 In the **Add** dialog box, in the **Selections to invert** list, choose **Load Boundary (Import 1)**, **Symmetry y (Import 1)**, **Symmetry x (Import 1)**, and **Moving Boundaries**.

6 Click **OK**.

7 In the **Settings** window for **Complement Selection**, type `Roller Boundaries` in the **Label** text field.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Optimization**.

5 Click **Add to Component 2** in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

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 2 (SOLID2)

Roller 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Solid Mechanics 2 (solid2)** and choose **Roller**.

2 In the **Settings** window for **Roller**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry x (Import 1)**.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry y (Import 1)**.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in y direction** check box.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Load Boundary (Import 1)**.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

0	x
-100[kN]	y

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

Free Triangular 2

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Maximum element growth rate** text field, type `Inf`.

4 In the **Curvature factor** text field, type `Inf`.

5 In the **Resolution of narrow regions** text field, type `0`.

6 Click **Build All**.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

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

6 In the **Model Builder** window, click the root node.

7 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

VERIFICATION

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

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

RESULTS

Applied Loads (solid2)

In the **Model Builder** window, under **Results** right-click **Applied Loads (solid2)** and choose **Delete**.

Topology Optimization

In the **Model Builder** window, right-click **Topology Optimization** and choose **Delete**.

DEFINITIONS (COMP2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Definitions**.

Free Shape Domain 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Free Shape Domain**.

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

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

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Symmetry/Roller**.

2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Roller Boundaries**.

Free Shape Boundary 1

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

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

3 From the **Selection** list, choose **Moving Boundaries**.

4 Locate the **Control Variable Settings** section. In the d_{max} text field, type `0.1`.

OPTIMIZATION

Step 1: Stationary

1 In the **Model Builder** window, expand the **Optimization** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 2)**.

VERIFICATION

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 2)**.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

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

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Shape Optimization

1 In the **Model Builder** window, right-click **Study 3** and choose **Optimization>Shape Optimization**.

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

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

4 Clear the **Move limits** check box.

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

6 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 2 (comp2)>Solid Mechanics 2>Global> comp2.solid2.Ws_tot - Total elastic strain energy - J**.

7 Locate the **Objective Function** section. From the **Objective scaling** list, choose **Initial solution based**.

8 Locate the **Control Variables** section. In the table, clear the **Solve for** check box for **Density Model 1 (dtopo1)**.

9 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 2 (comp2)>Definitions>Free Shape Domain 1> comp2.fsd1.area - Free shape area**.

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

Expression	Lower bound	Upper bound
comp2.fsd1.area/a/b		volfrac

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

RESULTS

Applied Loads (solid2)

In the **Model Builder** window, under **Results** right-click **Applied Loads (solid2)** and choose **Delete**.

Topology Optimization

In the **Model Builder** window, right-click **Topology Optimization** and choose **Delete**.

STUDY 3

Shape Optimization

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

2 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Shape Optimization**.

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

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

7 In the **Study** toolbar, click **Compute**.

RESULTS

Shape Optimization

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

2 In the **Model Builder** window, under **Results** click **Shape Optimization**.

3 In the **Shape Optimization** toolbar, click **Plot**.

Shape Optimization (alternative plot)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Shape Optimization (alternative plot)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Shape Optimization/ Solution 6 (6) (sol6)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Shape Optimization (alternative plot)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Verification/Solution 5 (4) (sol5)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

Surface 2

1 In the **Model Builder** window, right-click **Shape Optimization (alternative plot)** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

Line 1

1 Right-click **Shape Optimization (alternative plot)** and choose **Line**.

2 In the **Settings** window for **Line**, locate the **Data** section.

3 From the **Dataset** list, choose **Verification/Solution 5 (4) (sol5)**.

4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

5 Select the **Radius scale factor** check box.

6 In the associated text field, type `0.0035`.

7 From the **Coloring** list, choose **Uniform**.

8 From the **Color** list, choose **Gray**.

Line 2

1 Right-click **Shape Optimization (alternative plot)** and choose **Line**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

6 From the **Line type** list, choose **Tube**.

7 Select the **Radius scale factor** check box.

8 In the associated text field, type `0.0035`.

9 In the **Shape Optimization (alternative plot)** toolbar, click **Plot**.

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

Shaped Optimized Stress (solid2)

1 In the **Model Builder** window, under **Results** click **Stress (solid2) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Shaped Optimized Stress (solid2)` in the **Label** text field.