Created in COMSOL Multiphysics 6.2

Optimization of an Extruded MBB Beam

Introduction

Topology and shape optimization can be used to find and improve the design of products, but sometimes manufacturing constraints dictate that the design most be invariant in one of the dimensions, that is, an extruded geometry is desired. If 3-dimensional effects play little role, a 2D optimization can be used. Otherwise, one has to perform a 3D simulation and restrict the optimization to preserve the extruded property of the geometry.

This model is inspired by Topology Optimization of an MBB Beam, but the geometry is forced to be invariant in the z direction. The result is transferred to a second component in which shape optimization is performed, while still preserving the invariance in the z direction.

Model Definition

The model uses the *Density Model* feature to impose a minimum length scale on the *filtered material volume factor*. This is only defined on a symmetry plane, but it can be accessed as a z-invariant field in the volume by defining a **General Extrusion** operator.

The shape optimization uses equation-based modeling to define control variables and Helmholtz filters on the edges that form the intersection of a symmetry plane and the optimized boundaries. A second **General Extrusion** operator is used to transfer the variables of the Helmholtz filters from the edges to the optimized boundaries. Dirichlet boundary conditions are imposed on the Helmholtz filter to restrict the shape optimization to the box of the topology optimization.

Results and Discussion

The result of the topology optimization is shown in Figure 1. The model accounts for out-of-plane displacements, but the design is identical to the 2D result in the model Topology Optimization of an MBB Beam.

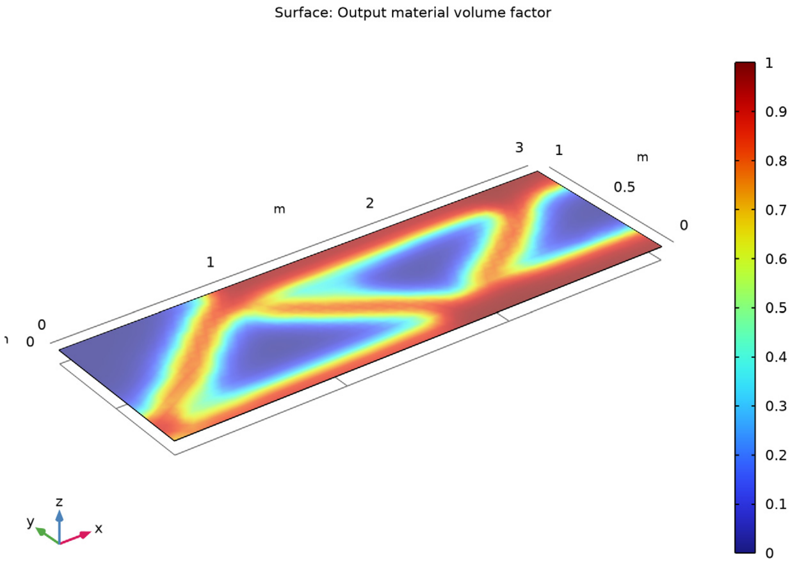


Figure 1: The filtered material volume factor is plotted on the z symmetry plane associated with the Density Model. An extrusion operator is used to transfer the variable to the volume.

The second **General Extrusion** will be more robust if it is used on an extruded design. There are several ways to achieve this, but in this model we will combine a **Filter dataset**, a **Mesh part** and a geometry **Import** feature to transfer a 2D version of the design. This is then extruded as shown in Figure 2.

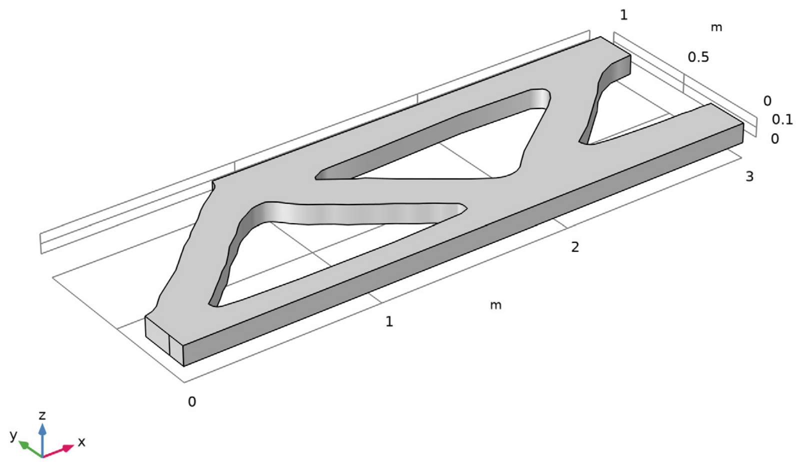


Figure 2: The topology optimized design has been transferred to a second component using an extrusion operator and a filter dataset pointing to a cut plane dataset.

Finally, the result of the shape optimization is shown in Figure 3 as the initial volume in gray on top of the optimized volume in red (transparency is enabled). The 90-degree angle near the top boundary is removed, because it is an artifact of the Helmholtz filter and thus not optimal (similar to Shape Optimization of an MBB Beam).

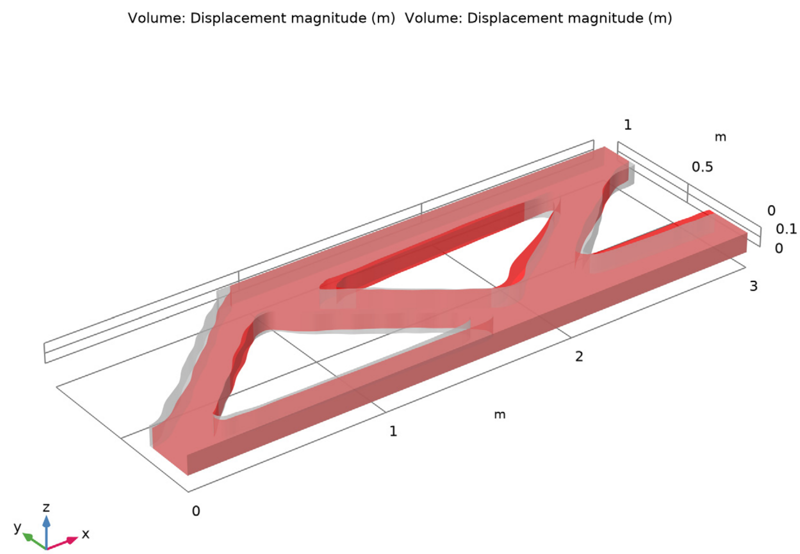


Figure 3: The plot shows the initial and optimized geometries in gray and red, respectively.

Notes About the COMSOL Implementation

This model combines the Optimization, Solid Mechanics and Deforming Geometry interfaces. The model uses a **Filter** dataset to transfer the geometry between components. An alternatively method is to export the edges as a text file with a section-wise format and import them as an interpolation curve. The interpolation curve has a parameter that can be used to straighten out the wiggles, but this approach requires more geometry operations to identify and delete the void domain.

Finally, the plot with transparency suffers from z-fighting artifacts on the **Symmetry/Roller** boundaries, but this is rectified by shrinking one of the volumes slightly.

Application Library path: `Optimization_Module/Design_Optimization/mbb_beam_extruded_optimization`

Modeling Instructions

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 **Preset Studies for Selected Physics Interfaces> Optimization>Topology Optimization, Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
a	3[m]	3 m	Beam half width
b	1[m]	1 m	Beam height
c	0.1[m]	0.1 m	Beam half depth
L1	0.1[m]	0.1 m	Support width
volfrac	0.5	0.5	Maximum volume fraction

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `a`.
4 In the **Height** text field, type `b`.

Work Plane 1 (wp1)>Point 1 (pt1)

1 In the **Work Plane** toolbar, click **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **yw** text field, type `L1`.

Work Plane 1 (wp1)>Point 2 (pt2)

1 In the **Work Plane** toolbar, click **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **xw** text field, type `a-L1/2`.
4 In the **yw** text field, type `b`.

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (m)
c

Symmetry z Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Symmetry z Boundary` 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 **z maximum** text field, type `c*0.001`.
5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
6 In the **Geometry** toolbar, click **Build All**.

The model geometry is now complete.

MATERIALS

Topology Link 1 (toplnk1)

In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Topology Link 1 (toplnk1)** and choose **Delete**.

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 Global Materials** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

SOLID MECHANICS (SOLID)

Roller 1

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

2 Select Boundaries 3 and 8 only.

Prescribed Displacement 1

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

2 Select Boundary 1 only.

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

4 From the **Displacement in y direction** list, choose **Prescribed**.

This is effectively a roller condition along the x-axis, but it is applied on a vertical boundary to avoid bending stiffness.

Boundary Load 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

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

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

0	x
-100[kN]	y
0	z

MESH 1

Create a swept mesh along the extrusion direction of the geometry.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Symmetry z Boundary**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

TOPOLOGY OPTIMIZATION

Density Model 1 (dtopo1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Topology Optimization** click **Density Model 1 (dtopo1)**.

2 In the **Settings** window for **Density Model**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Symmetry z Boundary**.

5 Locate the **Control Variable Initial Value** section. In the θ_0 text field, type `volfrac`.

DEFINITIONS

Use an extrusion operator to transfer the filtered design variable, `theta_f_bnd`, to the volume. This approach guarantees an extruded geometry.

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 From the **Selection** list, choose **Symmetry z Boundary**.
5 Locate the **Destination Map** section. In the **x-expression** text field, type `X`.
6 In the **y-expression** text field, type `Y`.
7 In the **z-expression** text field, type `0`.
8 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **All domains**.
5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
E_SIMP	mat1.Enu.E* genext1(dtopo1.theta_p)	Pa	Penalized Young's modulus
theta_f	genext1(dtopo1.theta_f)		Extruded filtered material volume factor

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
3 From the E list, choose **User defined**. In the associated text field, type `E_SIMP`.

TOPOLOGY OPTIMIZATION

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Topology Optimization` in the **Label** text field.

Initialize the study to create a default plot to display while solving.

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

The surface plot can visualize intermediate design variables, but now that the optimization has finished, it makes sense to change the filter dataset so that the threshold volume plot represents the optimized geometry.

Topology Optimization

1 In the **Model Builder** window, click **Topology Optimization**.

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

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

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

Expression	Lower bound	Upper bound
`comp1.dtopo1.theta_avg`		`volfrac`

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

6 From the **Plot group** list, choose **Output material volume factor**.

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

RESULTS

Topology Optimization

In the **Model Builder** window, expand the **Results>Topology Optimization** node.

Surface 1

1 In the **Model Builder** window, expand the **Results>Topology Optimization> Output material volume factor** node, then click **Surface 1**.

2 In the **Output material volume factor** toolbar, click **Plot**.

Filter

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Filter**.

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

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

Threshold

1 In the **Model Builder** window, under **Results>Topology Optimization** click **Threshold**.

2 In the **Threshold** toolbar, click **Plot**.

Create a **Cut Plane** dataset for a 2D plot group, so that the design can be exported.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

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

3 From the **Dataset** list, choose **Topology Optimization/Solution 1 (sol1)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

Filter 2

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 **Cut Plane 1**.

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

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

6 Locate the **Evaluation** section. From the **Smoothing** list, choose **None**.

7 Clear the **Use derivatives** check box.

8 Right-click **Filter 2** 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**.

GEOMETRY 2

Import 1 (imp1)

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Clear the **Form solids from surface objects** check box.

3 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

4 From the **Show in physics** list, choose **Boundary selection**.

Extrude 1 (ext1)

1 In the **Geometry** toolbar, click **Extrude**.

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

3 From the **Input faces** list, choose **Import 1**.

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

Distances (m)
c

5 Select the **Reverse direction** check box.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **y** text field, type `L1`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **y** text field, type `L1`.

7 In the **z** text field, type `c`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `a-L1/2`.

5 In the **y** text field, type `b`.

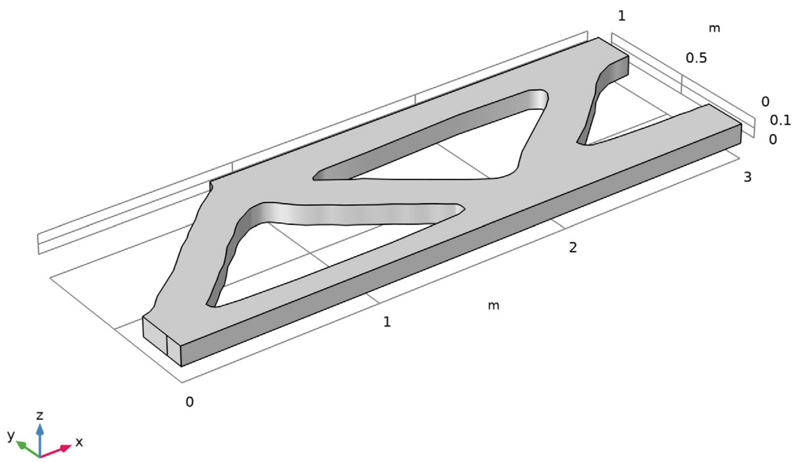
6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type `a-L1/2`.

8 In the **y** text field, type `b`.

9 In the **z** text field, type `c`.

10 In the **Geometry** toolbar, click **Build All**.



The geometry should now look like that in Figure 1.

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`.

8 In the **z minimum** text field, type `c*0.001`.

9 In the **z maximum** text field, type `c*0.999`.

Symmetry x Boundary

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

2 In the **Settings** window for **Box Selection**, type `Symmetry x Boundary` 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*0.999`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Roller Support

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

2 In the **Settings** window for **Box Selection**, type `Roller Support` 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 maximum** text field, type `a*0.001`.

5 In the **y maximum** text field, type `L1*1.001`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Symmetry z Edges

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

2 In the **Settings** window for **Adjacent Selection**, type `Symmetry z Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, select **Import 1** in the **Input selections** list.

7 Click **OK**.

Moving Boundaries Edges

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, select **Moving Boundaries** in the **Input selections** list.

7 Click **OK**.

Load Boundary

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

2 In the **Settings** window for **Box Selection**, type `Load Boundary` 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-L1/1.999`.

5 In the **y minimum** text field, type `b*0.999`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Roller Design

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

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

3 Locate the **Geometric Entity Level** section. 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 **Moving Boundaries** and **Load Boundary**.

6 Click **OK**.

Roller Design Edges

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

2 In the **Settings** window for **Adjacent Selection**, type `Roller Design Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Add**.

5 In the **Add** dialog box, select **Roller Design** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

8 From the **Geometric entity level** list, choose **Adjacent edges**.

Moving Boundaries Lower Edges

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

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

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

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

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Symmetry z Edges** and **Moving Boundaries Edges**.

6 Click **OK**.

Lower Points

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

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

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

4 Locate the **Box Limits** section. In the **y maximum** text field, type `0.001*b`.

Upper Points

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

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

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

4 Locate the **Box Limits** section. In the **y minimum** text field, type `b*0.999`.

Upper and Lower Points

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

2 In the **Settings** window for **Union Selection**, type `Upper and Lower Points` in the **Label** text field.

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

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

5 In the **Add** dialog box, in the **Selections to add** list, choose **Lower Points** and **Upper Points**.

6 Click **OK**.

Left Points

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

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

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

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

Right Points

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

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

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

4 Locate the **Box Limits** section. In the **x minimum** text field, type `a*0.999`.

Left and Right Points

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

2 In the **Settings** window for **Union Selection**, type `Left and Right Points` in the **Label** text field.

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

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

5 In the **Add** dialog box, in the **Selections to add** list, choose **Left Points** and **Right Points**.

6 Click **OK**.

COMPONENT 2 (COMP2)

Add the physics necessary for performing the shape optimization.

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 **Topology Optimization**.

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

6 In the tree, select **Mathematics>PDE Interfaces>Lower Dimensions> Coefficient Form Edge PDE (ce)**.

7 Click to expand the **Dependent Variables** section. In the table, clear the **Solve** check box for **Topology Optimization**.

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

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

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

COMPONENT 2 (COMP2)

Add control variable fields and Helmholtz filters on the edges shared by the z symmetry plane and the boundaries to be optimized.

DEFINITIONS (COMP2)

Control Variable Field 1 (p1)

1 In the **Definitions** toolbar, click **Control Variables** and choose **Control Variable Field**.

2 In the **Settings** window for **Control Variable Field**, type `movex` in the **Name** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Edge**.

4 From the **Selection** list, choose **Moving Boundaries Lower Edges**.

5 Locate the **Discretization** section. From the **Element order** list, choose **Linear**.

6 Locate the **Bounds** section. In the **Lower bound** text field, type `-1`.

7 In the **Upper bound** text field, type `1`.

Control Variable Field 2 (p2)

1 Right-click **Control Variable Field 1 (movex)** and choose **Duplicate**.

2 In the **Settings** window for **Control Variable Field**, type `movey` in the **Name** text field.

TOPOLOGY OPTIMIZATION

Topology Optimization

1 In the **Model Builder** window, under **Topology Optimization** click **Topology Optimization**.

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

3 In the table, clear the **Solve for** check boxes for **Control Variable Field 1 (movex)** and **Control Variable Field 2 (movey)**.

COEFFICIENT FORM EDGE PDE (CE)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Coefficient Form Edge PDE (ce)**.

2 In the **Settings** window for **Coefficient Form Edge PDE**, locate the **Edge Selection** section.

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

4 Locate the **Units** section. In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

5 Click to expand the **Discretization** section. From the **Frame** list, choose **Geometry**.

6 Click to expand the **Dependent Variables** section. In the **Field name (1)** text field, type `move`.

7 Click **Add Dependent Variable**.

8 In the **Dependent variables (1)** table, enter the following settings:

dXg
dYg

GLOBAL DEFINITIONS

Parameters 1

Add the shape optimization parameters.

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Lmin	0.2[m]	0.2 m	Shape optimization filter radius
Lmax	0.1[m]	0.1 m	Shape optimization maximum displacement

COEFFICIENT FORM EDGE PDE (CE)

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 2 (comp2)>** **Coefficient Form Edge PDE (ce)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the c text-field array, type `Lmin^2` in the first column of the first row.

4 In the c text-field array, type `Lmin^2` in the second column of the second row.

5 Locate the **Absorption Coefficient** section. In the a text-field array, type `1` in the first column of the first row.

6 In the a text-field array, type `1` in the second column of the second row.

7 Locate the **Source Term** section. In the f text-field array, type `movex` on the first row.

8 In the f text-field array, type `movey` on the second row.

This completes the setup of the Helmholtz filter on an edge with a filter radius of `Lmin`. Note that the use of a **Frequency Domain** study step would require removal of the damping terms. Next specify boundary conditions to prevent the shape from moving outside the box of the topology optimization.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Points** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Upper and Lower Points**.

4 Locate the **Dirichlet Boundary Condition** section. Clear the **Prescribed value of dXg** check box.

Dirichlet Boundary Condition 2

1 In the **Physics** toolbar, click **Points** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Left and Right Points**.

4 Locate the **Dirichlet Boundary Condition** section. Clear the **Prescribed value of dYg** check box.

DEFINITIONS (COMP2)

Add a nonlocal integration coupling to enforce the volume constraint.

Integration 1 (intop1)

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

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

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

Once again use an extrusion operator to transfer the filtered field from the edges to the boundaries.

General Extrusion 2 (genext2)

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 **Edge**.

4 From the **Selection** list, choose **Moving Boundaries Lower Edges**.

5 Locate the **Destination Map** section. In the **x-expression** text field, type `Xg`.

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

7 In the **z-expression** text field, type `0`.

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

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

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

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

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

Name	Expression	Unit	Description
dXg_bnd	genext2(dXg)*Lmax	m	X boundary displacement
dYg_bnd	genext2(dYg)*Lmax	m	Y boundary displacement

COMPONENT 2 (COMP2)

Deforming Domain 1

In the **Physics** toolbar, click **Deformed Geometry** and choose **Free Deformation**.

Prescribed Normal Mesh Displacement 1

1 In the **Deformed Geometry** toolbar, click **Prescribed Normal Mesh Displacement**.

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

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

Prescribed Mesh Displacement 1

1 In the **Deformed Geometry** toolbar, click **Prescribed Mesh Displacement**.

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

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

4 Locate the **Prescribed Mesh Displacement** section. Specify the *dx* vector as

dXg_bnd	X
dYg_bnd	Y
0	Z

SOLID MECHANICS 2 (SOLID2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Solid Mechanics 2 (solid2)**.

Roller 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Roller**.
2 In the **Settings** window for **Roller**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry x Boundary**.

Roller 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Roller**.
2 In the **Settings** window for **Roller**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **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 **Roller Support**.
4 Locate the **Prescribed Displacement** section. From the **Displacement in y direction** list, choose **Prescribed**.

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**.
4 Locate the **Force** section. From the **Load type** list, choose **Total force**.
5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

0	x
-100[kN]	y
0	z

MESH 2

Once again create a swept mesh along the extrusion direction of the geometry.

Free Triangular 1

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click to expand the **Element Size Parameters** section. In the **Curvature factor** text field, type `2`.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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

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

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.

TOPOLOGY OPTIMIZATION

Step 1: Stationary

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

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

SHAPE OPTIMIZATION

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

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

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 In the **Maximum number of iterations** text field, type `20`.

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

5 Click **Replace 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**.

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

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

8 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 2 (comp2)>Definitions>Nonlocal couplings> comp2.intop1(expr) - Integration 1**.

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

Expression	Lower bound	Upper bound
comp2.intop1(1)/a/b/c		volfrac

Step 1: Stationary

1 In the **Model Builder** window, 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 **Topology Optimization (Component 1)**.

Initialize the study to create a plot for use while solving.

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

The default solver groups the filtered boundary displacements separate from the volume material displacements and this breaks the sensitivity analysis. This issue can be solved by grouping them together or by using a **Fully Coupled** solver.

RESULTS

Coefficient Form Edge PDE

1 In the **Model Builder** window, expand the **Results>Coefficient Form Edge PDE** node, then click **Coefficient Form Edge PDE**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

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

4 From the **Frame** list, choose **Geometry (Xg, Yg, Zg)**.

5 In the **Model Builder** window, collapse the **Coefficient Form Edge PDE** node.

Line 1

1 In the **Model Builder** window, expand the **Coefficient Form Edge PDE** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

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

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

Arrow Line 1

1 In the **Model Builder** window, right-click **Coefficient Form Edge PDE** and choose **Arrow Line**.

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

3 In the **X-component** text field, type `dXg*Lmax`.

4 In the **Y-component** text field, type `dYg*Lmax`.

5 In the **Z-component** text field, type `0`.

6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

7 Select the **Scale factor** check box.

8 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

Selection 1

1 Right-click **Arrow Line 1** and choose **Selection**.

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

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

Color Expression 1

1 In the **Model Builder** window, right-click **Arrow Line 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `max(abs(dXg),abs(dYg))`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `1`.

Shape Optimization

1 In the **Model Builder** window, under **Results** click **Coefficient Form Edge PDE**.

2 In the **Settings** window for **3D Plot Group**, type `Shape Optimization` in the **Label** text field.

SHAPE OPTIMIZATION

Shape Optimization

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

Solver Configurations

In the **Model Builder** window, expand the **Shape Optimization>Solver Configurations** node.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Shape Optimization>Solver Configurations> Solution 2 (sol2)>Optimization Solver 1>Stationary 1>Segregated 1** node.

2 Right-click **Material Frame Variables** and choose **Disable**.

3 In the **Model Builder** window, click **Coefficient Form Edge PDE**.

4 In the **Settings** window for **Segregated Step**, locate the **General** section.

5 Under **Variables**, click **Add**.

6 In the **Add** dialog box, select **Material mesh displacement (comp2.material.disp)** in the **Variables** list.

7 Click **OK**.

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

RESULTS

Topology Optimization 1

In the **Model Builder** window, under **Results** right-click **Topology Optimization 1** and choose **Delete**.

Create a dataset in the geometry frame, so that the initial and optimized volumes can be plotted on top of each other. The plot illustrates the shape change in an alternative way, but it only makes sense with transparency enabled.

Topology Optimization/Solution 1 (4) (sol1)

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

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

3 From the **Solution** list, choose **Solution 2 (sol2)**.

4 From the **Component** list, choose **Component 2 (comp2)**.

5 From the **Frame** list, choose **Geometry (Xg, Yg, Zg)**.

Volumetric (for transparent view)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Shape Optimization/Solution 2 (3) (sol2)**.

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

5 In the **Label** text field, type `Volumetric (for transparent view)`.

Volume 1

1 Right-click **Volumetric (for transparent view)** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

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

Volume 2

1 In the **Model Builder** window, right-click **Volumetric (for transparent view)** and choose **Volume**.

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

3 From the **Dataset** list, choose **Shape Optimization/Solution 2 (4) (sol2)**.

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

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

6 Click the **Transparency** button in the **Graphics** toolbar.

7 In the **Volumetric (for transparent view)** toolbar, click **Plot**.

There are some z-fighting artifacts on the **Symmetry/Roller** boundaries, but this can be avoided by shrinking one of the plots slightly.

Deformation 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Deformation**.

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

3 In the **X-component** text field, type `-1e-3*(Xg/a-0.5)`.

4 In the **Y-component** text field, type `-1e-3*(Yg/b-0.5)`.

5 In the **Z-component** text field, type `-1e-3*(Zg/c-0.5)`.

6 Locate the **Scale** section.

7 Select the **Scale factor** check box. In the associated text field, type `1`.

8 In the **Volumetric (for transparent view)** toolbar, click **Plot**.

Stress (solid) Topology Optimization

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, type `Stress (solid) Topology Optimization` in the **Label** text field.