Model created in COMSOL Multiphysics 6.4

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 setup the topology optimization with an extrusion manufacturing constraint.

The shape optimization combines a **Free Shape Shell** feature with a **General Extrusion** operator, so that the deformation is only occurs in the xy-plane and does not vary in the z-coordinate.

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.

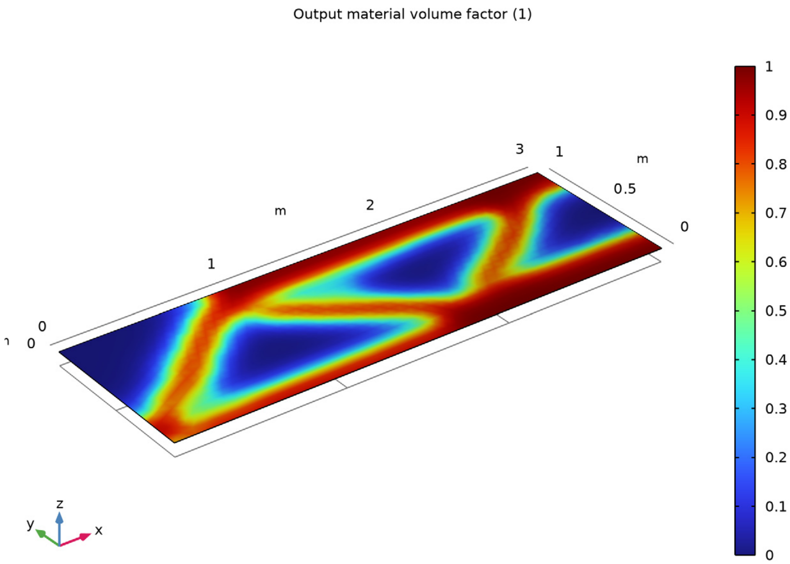


Figure 1: The filtered material volume factor is plotted on the z symmetry plane associated with the Density Model. An extrusion operator is used to transfer the variable to the volume.

The **General Extrusion** operator used for the shape optimization 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.

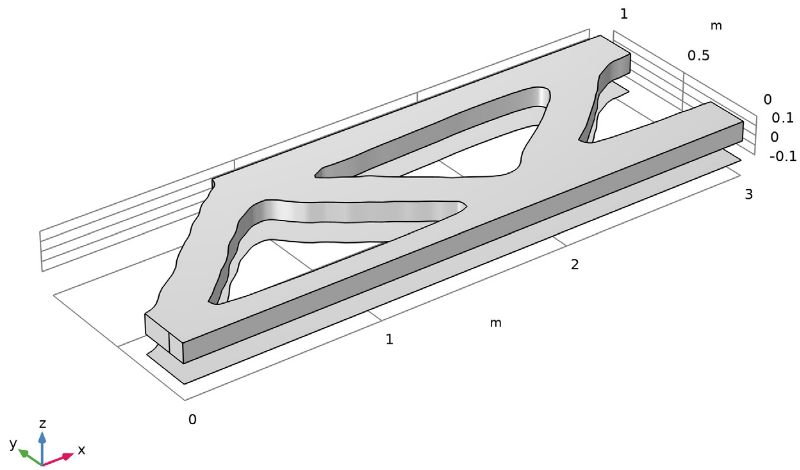


Figure 2: The topology optimized design has been transferred to a second component using an extrusion operator and a filter dataset pointing to a cut plane dataset. Note the shell representation in the background which is used to setup the shape optimization.

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).

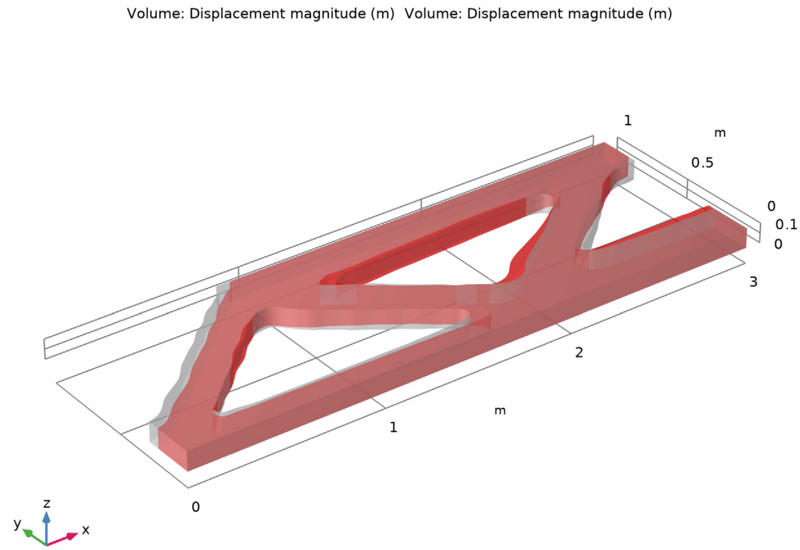


Figure 3: The plot shows the initial and optimized geometries in gray and red, respectively.

Notes About the COMSOL Implementation

This model combines the Topology Optimization, Shape 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 **Materials** 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 the **Add to Global Materials** button in the window toolbar.

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

SOLID MECHANICS (SOLID)

Roller 1

1 In the **Physics** toolbar, click **Boundaries** 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 **More Generators** 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**, click to expand the **Manufacturing Constraints** section.

3 From the **Manufacturing constraints** list, choose **Extrusion**.

4 Click to expand the **Extruded Boundary** section. From the **Selection** list, choose **Symmetry z Boundary**.

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

MATERIALS

Topology Link 1 (toplnk1)

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

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

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

4 Locate the **Link Settings** section. From the **Topology source** list, choose **Density Model 1 (dtopo1)**.

STUDY 1: TOPOLOGY OPTIMIZATION

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

2 In the **Settings** window for **Study**, type `Study 1: 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, under **Study 1: Topology Optimization** 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 Click to expand the **Output** section. Select the **Plot** checkbox.

6 In the table, enter the following settings:

Plot group	Plot window
Output material volume factor	Graphics

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

RESULTS

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**.

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 **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **Cut Plane**.

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

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

5 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 `dtopo1.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** checkbox.

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 **Simplify and Repair** section.

2 Clear the **Form solids from surface objects** checkbox.

3 Click to expand the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

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 From the **Input object handling** list, choose **Keep**.

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

Distances (m)
c

6 Select the **Reverse direction** checkbox.

7 Click **Build Selected**.

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 In the **Settings** window for **Move**, locate the **Input** section.

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

4 Locate the **Displacement** section. In the **z** text field, type `-c`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

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

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`.

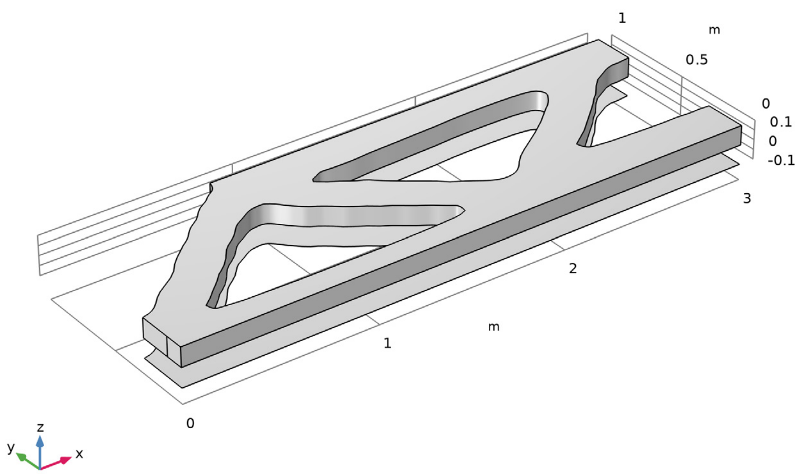
6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

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.

Line Segment 3 (ls3)

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

5 Locate the **Starting Point** section. In the **x** text field, type `a-L1/2`.

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

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

8 Locate the **Endpoint** section. In the **x** text field, type `a`.

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

10 In the **z** text field, type `-c`.

11 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Line Segment 4 (ls4)

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

5 Locate the **Starting Point** section. In the **x** text field, type `0`.

6 In the **y** text field, type `0`.

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

8 Locate the **Endpoint** section. In the **x** text field, type `0`.

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

10 In the **z** text field, type `-c`.

11 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Form Union (fin)

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

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

Symmetry x

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

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

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

Symmetry x edge 1

1 Right-click **Symmetry x** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Symmetry x edge 1` in the **Label** text field.

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

4 Click **Build Selected**.

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 `0.001*a`.

5 In the **y maximum** text field, type `1.01*L1`.

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

Symmetry x edge 2

1 Right-click **Roller support** and choose **Duplicate**.

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

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

4 In the **Label** text field, type `Symmetry x edge 2`.

5 Locate the **Box Limits** section. In the **y minimum** text field, type `0.99*L1`.

6 In the **y maximum** text field, type `Inf`.

7 Click **Build Selected**.

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-0.51*L1`.

5 In the **y minimum** text field, type `0.99*b`.

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

Symmetry y edge 1

1 Right-click **Load boundary** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Symmetry y edge 1` in the **Label** text field.

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

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

Symmetry y edge 2

1 Right-click **Symmetry y edge 1** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Symmetry y edge 2` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `-Inf`.

4 In the **y maximum** text field, type `0.01*b`.

5 Click **Build Selected**.

Fixed edges

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

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

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

4 In the **Label** text field, type `Fixed edges`.

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

6 In the **Add** dialog, in the **Selections to add** list, choose **Line Segment 3** and **Line Segment 4**.

7 Click **OK**.

Symmetry y edge

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

2 In the **Settings** window for **Union Selection**, type `Symmetry y edge` in the **Label** text field.

3 In the **Model Builder** window, click **Symmetry y edge (unisel2)**.

4 In the **Label** text field, type `Symmetry y edge`.

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

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

7 In the **Add** dialog, in the **Selections to add** list, choose **Symmetry y edge 1** and **Symmetry y edge 2**.

8 Click **OK**.

Symmetry x edge

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

2 In the **Settings** window for **Union Selection**, type `Symmetry x edge` 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, in the **Selections to add** list, choose **Symmetry x edge 1** and **Symmetry x edge 2**.

6 Click **OK**.

COMPONENT 1: TOPOLOGY OPTIMIZATION

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 In the **Settings** window for **Component**, type `Component 1: Topology Optimization` in the **Label** text field.

COMPONENT 2: SHAPE OPTIMIZATION

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

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

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

COMPONENT 2: SHAPE OPTIMIZATION (COMP2)

Free Shape Shell 1

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

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

3 From the **Selection** list, choose **Move 1**.

4 Locate the **Control Variable Settings** section. From the d_{max} list, choose **User defined**.

5 In the table, enter the following settings:

	Lock	Lower bound (m)	Upper bound (m)
X		-Lmax	Lmax

	Lock	Lower bound (m)	Upper bound (m)
Y		-Lmax	Lmax
Z	√	-0.2	0.2

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

7 In the text field, type Lmin.

Symmetry/Roller 1

1 In the **Shape Optimization** toolbar, click **Symmetry/Roller**.

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

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

4 From the **Selection** list, choose **Symmetry y edge**.

5 Locate the **Prescribed Normal Vector** section. Specify the **n** vector as

0	X
1	Y
0	Z

Symmetry/Roller 2

1 Right-click **Symmetry/Roller 1** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Symmetry x edge**.

4 Locate the **Prescribed Normal Vector** section. Specify the **n** vector as

1	X
0	Y
0	Z

Fixed Edge 1

1 In the **Shape Optimization** toolbar, click **Fixed Edge**.

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

3 From the **Selection** list, choose **Fixed edges**.

ADD PHYSICS

1 In the **Shape Optimization** 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** checkbox for **Study 1: Topology Optimization**.

5 Click the **Add to Component 2: Shape Optimization** button in the window toolbar.

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

MATERIALS

Material Link 1 (matlnk1)

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

STUDY 1: TOPOLOGY OPTIMIZATION

Topology Optimization

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

2 In the table, clear the **Solve for** checkbox for **Free Shape Shell 1**.

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 boundary to the domain.

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 **Move 1**.

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 `-c`.

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

DEFORMED GEOMETRY

Prescribed Deformation 1

1 In the **Deformed Geometry** toolbar, click **Prescribed Deformation**.

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

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

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

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

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**.

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 $\mathbf{F}_{tot}$ 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 **More Generators** 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** checkbox for **Solid Mechanics (solid)**.

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

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

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

STUDY 1: TOPOLOGY OPTIMIZATION

Step 1: Stationary

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

2 In the **Solve for** column of the table, under **Component 2: Shape Optimization (comp2)**, clear the checkbox for **Deformed Geometry**.

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.

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** checkbox.

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

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

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

8 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 2: Shape Optimization (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 **Solve for** column of the table, under **Component 1: Topology Optimization (comp1)**, clear the checkbox for **Topology Optimization**.

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

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

Shape Optimization

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

RESULTS

Deformed Geometry, Topology Optimization 1

Right-click 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.

Study 1: 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: Shape Optimization (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 **Study 2: Shape Optimization/Solution 2 (3) (sol2)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

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 **Study 2: 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** checkbox. 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.