Model created in COMSOL Multiphysics 6.4

Design Optimization of a Beam

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

When applying design optimization, one has to choose between different techniques with different strengths and weaknesses regarding robustness, learning curve, design freedom, and associated potential improvements. This example illustrates how to apply parameter, shape, and topology optimization to the problem of minimizing the weight of a beam subject to a displacement constraint. In this case, the parameter and shape optimization happen to give similar designs with similar performance, while the topology optimization gives a design with significantly better performance. Although it is not unusual that the superior design freedom of topology optimization results in superior performance, this is in no way guaranteed. The optimization can get stuck at a local minimum and it is often possible to improve on the result by using it as an initial design for shape or parameter optimization, as shown in the model Shape Optimization of a Tesla Microvalve.

Model Definition

The model geometry (Figure 1) consists of two regions: A fixed domain on which a distributed load is applied and a domain to be designed for optimal performance. For the parameter-optimization case, the y-coordinates of two points on the lower boundary are changed, while the entire lower boundary is deformed for the shape-optimization case. Finally, the topology optimization is able to distribute material freely in the design domain.

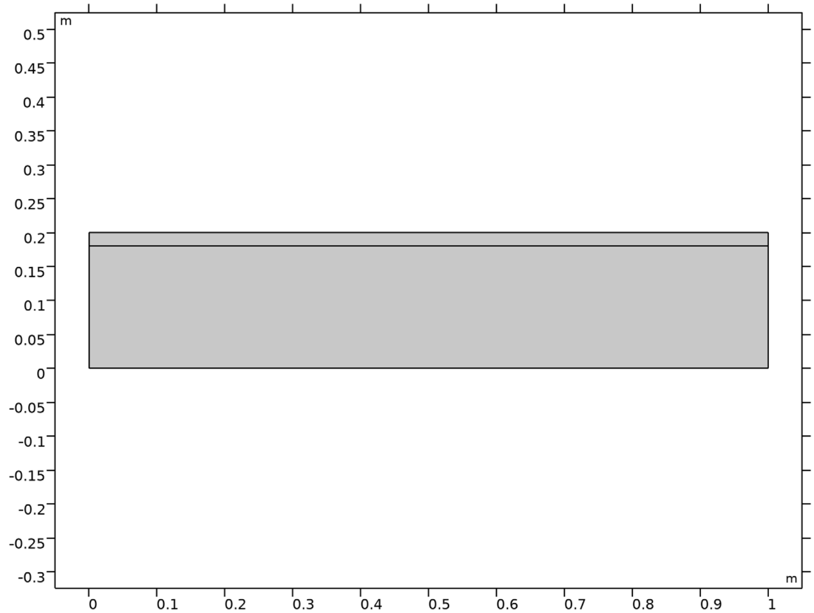

Figure 1: The model geometry.

The beam is made of aluminum, and the displacement field is calculated under the assumption of linear elasticity. The objective is to minimize the mass while keeping the displacement of the upper-right corner below 0.2 mm.

The parameter optimization moves the y-coordinates of two points in the polygonal geometry and constructs a new mesh in every iteration. This means that the gradient cannot be computed analytically using the adjoint method, so the use of a derivative-free optimization solver (such as Nelder–Mead) is appropriate.

The shape optimization uses two second-order Bernstein polynomials to move the bottom boundary. This is implemented using the **Free Shape Domain** and **Polynomial Boundary** features. The maximum displacement is set such that the lower boundary cannot move to the point where the topology changes. The lower-left point is fixed, but the right point is allowed to slide along a **Roller** boundary.

For a detailed introduction to the use of structural topology optimization and how to use a Helmholtz filter for regularization, see the model Topology Optimization of an MBB Beam. The main point is that the Young's modulus varies spatially to approximate an implicit geometry description. Moreover, it is not possible to specify zero void stiffness, because this causes the displacement field to become undefined in the void regions.

Results and Discussion

Figure 2 displays the result of the parameter optimization together with the distributed load and the mesh. The displacement field is shown in colors and the maximum value is located near the right end of the beam,

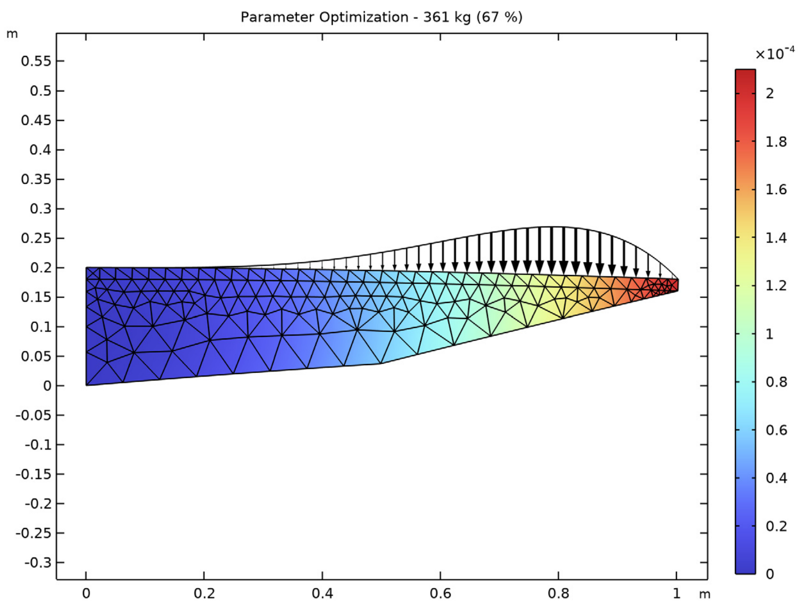

Figure 2: The y-coordinates of two points in the geometry are changed to remove material from the lower-right part of the beam. The color scale reveals that the displacement at the upper-right corner satisfies the 0.2 mm constraint.

The shape optimization results in a somewhat similar design and the mass is only 1% lower; see Figure 3. The computation time is similar to that for the parameter-optimization case despite having more control variables. This is due to the fact that the mesh topology is fixed, which allows for the use of analytic computation of the gradient via the adjoint method. It is thus possible to use IPOPT, a quadratic gradient-based optimizer. Moreover, the addition of more control variables is not expected to affect the computational time adversely.

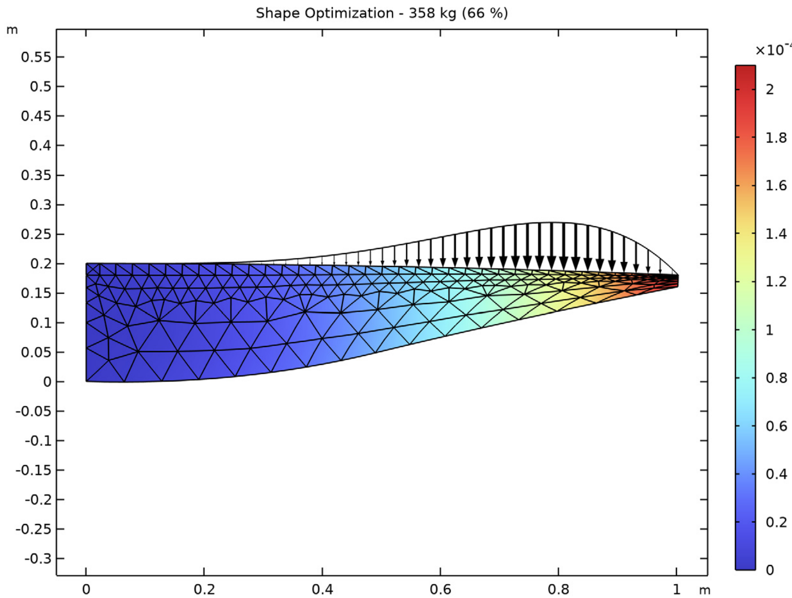

Figure 3: The shape of the beam is optimized by moving the lower boundary vertically. The movement is large toward the right end of the beam, which causes a concentration of elements in this area.

Figure 4 plots the design found with topology optimization. Four triangles have been cut out and the right end of the beam is left unsupported. This allows for a seesaw effect because most of the load is to the left of the unsupported region. It can thus be expected that the performance of the design is sensitive to variations in the load distribution. The model Bracket — Topology Optimization shows how multiple load cases can be used to find designs that are robust to load variations.

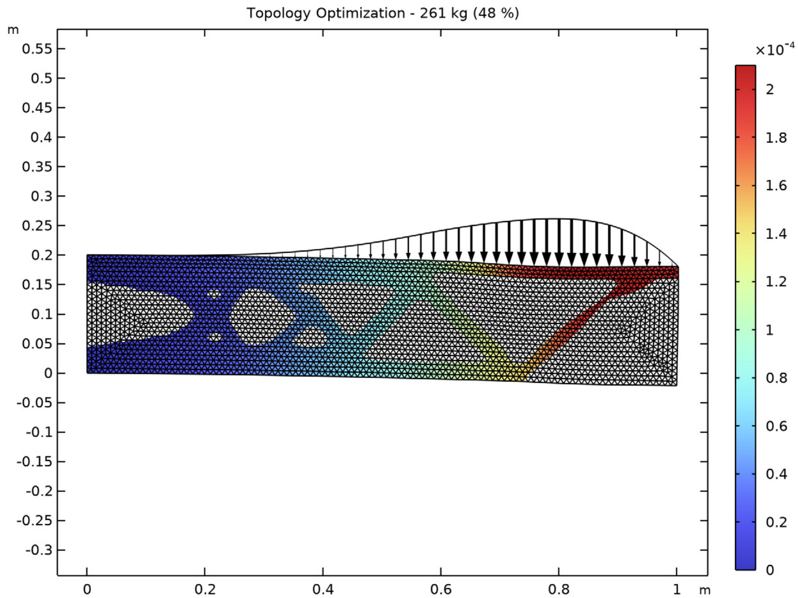

Figure 4: Topology optimization has significant design freedom, which in this case results in a design that is 27% lighter than that of the shape optimization.

Notes About the COMSOL Implementation

This model combines the Optimization and Solid Mechanics interfaces. First, you set up and solve the parameter optimization. Then you define the shape-optimization problem using the features available from the component node. Both the shape and parameter optimizations work with a coarser mesh, but the topology optimization needs a rather fine mesh if a structure with slender components is desired. This is particularly true if the constraint on the displacement is relaxed to a higher value.

The topology optimization is implemented by adding a Density Model feature to the component. This is used to define a custom Young's modulus for a modified linear elasticity model under the Solid Mechanics interface. The second-order discretization in the Solid Mechanics interface cannot be justified for the implicit geometry description associated with topology optimization, but the extra computational cost is manageable for this simple 2D problem.

Application Library path: `Optimization_Module/Design_Optimization/beam_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 **2D**.

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 **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Add parameters for the geometry, including the *y*-coordinates for two points to be controlled by the parameter optimization.

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
L0	1[m]	1 m	Beam length
T0	20[cm]	0.2 m	Beam height
M0	L0*T0*2700[kg/m^3]	540 kg/m	Beam weight
Yopt1	0[m]	0 m	Y position, point 1
Yopt2	0[m]	0 m	Y position, point 2
maxDisp	0.2[mm]	2E-4 m	Maximum displacement

GEOMETRY 1

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (m)	y (m)
0	0
L0/2	Yopt1
L0	Yopt2
L0	0.9*T0
L0	T0
L0	T0
0	T0

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

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

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

x (m)	y (m)
0	0.9*T0
L0	0.9*T0

Bottom Boundaries

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

2 In the **Settings** window for **Box Selection**, type `Bottom 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 **y maximum** text field, type `eps`.

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

Right Boundaries

1 Right-click **Bottom Boundaries** and choose **Duplicate**.

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

3 Locate the **Box Limits** section. In the **x minimum** text field, type `L0-eps`.

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

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

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

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** > **Aluminum**.

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.

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)

Fix the left boundary. Then impose a nonuniform load on the upper boundary.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 1 and 3 only.

Boundary Load 1

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

2 Select Boundary 5 only.

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

4 From the **Load type** list, choose **Force per reference length**.

5 Specify the $\mathbf{f}_L$ vector as

`0`	x
`-1e6[N/m]*((X/L0)^4*(1-X/L0))`	y

Enable **Variable Utilities** and use it do define a **Mass Properties** feature.

DEFINITIONS

Mass Properties 1 (mass1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Physics Utilities** > **Mass Properties**.
2 In the **Settings** window for **Mass Properties**, locate the **Density** section.
3 From the **Density source** list, choose **From physics interface**.

Tip Displacement

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.
2 In the **Settings** window for **Point Probe**, type `Tip Displacement` in the **Label** text field.
3 In the **Variable name** text field, type `pnt_disp`.
4 Select Point 7 only.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.
3 From the **Probes** list, choose **None**.

   Solve the model without minimization to generate a plot to update while optimizing.
4 In the **Model Builder** window, click **Study 1**.
5 In the **Settings** window for **Study**, type `Study 1: Parameter Optimization` in the **Label** text field.
6 In the **Study** toolbar, click **Compute**.

RESULTS

Parameter Optimization

1 In the **Settings** window for **2D Plot Group**, type `Parameter Optimization` in the **Label** text field.
2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
3 In the **Title** text area, type `Parameter Optimization - eval(mass1.mass) kg (eval(mass1.mass/M0*100,,%d) %)`.
4 From the **Number format** list, choose **Automatic**.
5 In the **Precision** text field, type `3`.
6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **Parameter Optimization** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.disp`.

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

5 In the **Minimum** text field, type `0`.

6 In the **Maximum** text field, type `1.05*maxDisp`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `100`.

Mesh 1

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

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

3 From the **Element color** list, choose **None**.

Deformation 1

1 Right-click **Mesh 1** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `100`.

Line 1

1 In the **Model Builder** window, right-click **Parameter Optimization** 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**.

Deformation 1

1 Right-click **Line 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, click **Add Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Load** > **solid.fax,solid.fay - Force per deformed area (spatial frame)**.

3 Locate the **Expression** section. In the **X-component** text field, type `u`.

4 In the **Y-component** text field, type `v-1e-8*solid.FperLengthy`.

5 Locate the **Scale** section.

6 Select the **Scale factor** checkbox. In the associated text field, type `100`.

Arrow Line 1

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

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

3 In the **X-component** text field, type `solid.FperLengthx`.

4 In the **Y-component** text field, type `solid.FperLengthy`.

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

6 Select the **Scale factor** checkbox. In the associated text field, type `1e-6`.

7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `160`.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Deformation 1

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

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

3 Select the **Scale factor** checkbox. In the associated text field, type `100`.

4 In the **Parameter Optimization** toolbar, click **Plot**.

STUDY 1: PARAMETER OPTIMIZATION

Set up and compute the solution to the parameter-optimization problem.

Parameter Optimization

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

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

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

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Mass Properties 1** > **comp1.mass1.mass - Mass - kg**.

5 Locate the **Control Parameters** section. Click **Add** twice.

6 In the table, enter the following settings:

Parameter	**Initial value**	**Lower bound**	**Upper bound**	**Unit**
Yopt1 (Y position, point 1)	0	0	`0.9*T0`	m
Yopt2 (Y position, point 2)	0	0	`0.9*T0`	m

7 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **comp1.pnt_disp - Tip Displacement - m**.

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

Expression	**Lower bound**	**Upper bound**	**Evaluate for**
`comp1.pnt_disp/maxDisp`		1	Stationary

9 Click to expand the **Output** section. Select the **Plot** checkbox.

10 From the **Probes** list, choose **None**.

11 In the **Model Builder** window, click **Study 1: Parameter Optimization**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Clear the **Generate default plots** checkbox.

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

RESULTS

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

Next, solve the same problem using shape optimization with a **Free Shape Domain** and a **Polynomial Boundary**.

COMPONENT 1 (COMP1)

Free Shape Domain 1

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

Polynomial Boundary 1

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

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

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

4 Click to expand the **Continuity** section. From the **Preserve continuity of normals over symmetry boundaries** list, choose **Disabled**.

5 From the **Preserve continuity of normals between polynomial boundaries** list, choose **Enabled**.

6 Locate the **Control Variable Settings** section. In the text field, type `0.9*T0`.

Symmetry/Roller 1

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

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

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

ROOT

Add a study for the shape optimization and run it before adding the optimization, so that a plot can be generated for visualizing the results while optimizing.

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

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Results While Solving** section.

2 From the **Probes** list, choose **None**.

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

4 In the **Settings** window for **Study**, locate the **Study Settings** section.

5 Clear the **Generate default plots** checkbox.

6 In the **Label** text field, type `Study 2: Shape Optimization`.

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

RESULTS

Shape Optimization

1 In the **Model Builder** window, right-click **Parameter Optimization** and choose **Duplicate**.

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

3 Locate the **Title** section. In the **Title** text area, type `Shape Optimization - eval(mass1.mass) kg (eval(mass1.mass/M0*100,,%d) %)`.

4 From the **Number format** list, choose **Automatic**.

5 In the **Precision** text field, type `3`.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Shape Optimization/ Solution 4 (sol4)**.

STUDY 2: SHAPE OPTIMIZATION

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

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

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

3 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1) > Definitions > Mass Properties 1 > comp1.mass1.mass - Mass - kg**.

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

5 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1) > Definitions > comp1.pnt_disp - Tip Displacement - m**.

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

Expression	Lower bound	Upper bound
comp1.pnt_disp/maxDisp		1

7 Click to expand the **Output** section. From the **Probes** list, choose **None**.

8 Select the **Plot** checkbox.

9 In the table, enter the following settings:

Plot group	Plot window
Shape Optimization	Graphics

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

RESULTS

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

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

COMPONENT 1 (COMP1)

Finally, set up a topology-optimization problem using the density method. Start by adding the **Density Model** feature and use it to define a penalized Young's modulus in an alternative **Linear Elastic Material**.

Density Model 1 (dtopo1)

1. In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Topology Optimization**.
2. Select Domain 1 only.
3. In the **Settings** window for **Density Model**, locate the **Control Variable Discretization** section.
4. From the **Element order** list, choose **Constant**.
5. Locate the **Control Variable Initial Value** section. In the θ_0 text field, type 1.

Prescribed Material 1

1. In the **Topology Optimization** toolbar, click **Prescribed Material**.
2. Select Domain 2 only.

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)**.

MESH 1

Topology optimization uses an implicit geometry description, which requires a finer mesh

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

MESH 2

1. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
2. From the **Element size** list, choose **Extremely fine**.

3 Click **Build All**.

The quadratic discretization used in the **Solid Mechanics** interface cannot be justified for the implicit geometry description associated with topology optimization, but the extra computational cost is manageable for this simple 2D problem.

ROOT

Add a study for the optimization. Follow the (now) usual procedure for creating a plot to be used in the optimization.

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

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

STUDY 3: TOPOLOGY OPTIMIZATION

1 In the **Settings** window for **Study**, type `Study 3: Topology Optimization` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3: Topology Optimization** 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 (comp1)**, clear the checkbox for **Deformed Geometry**.

4 Locate the **Results While Solving** section. From the **Probes** list, choose **None**.

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

RESULTS

Topology Optimization

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

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

3 Locate the **Title** section. In the **Title** text area, type `Topology Optimization - eval(mass1.mass) kg (eval(mass1.mass/M0*100,,%d) %)`.

4 From the **Number format** list, choose **Automatic**.

5 In the **Precision** text field, type `3`.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Topology Optimization/ Solution 5 (sol5)**.

Filter 1

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

2 Right-click **Surface 1** and choose **Filter**.

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

4 In the **Logical expression for inclusion** text field, type `0.5<dtopo1.theta`.

STUDY 3: TOPOLOGY OPTIMIZATION

Topology Optimization

1 In the **Study** toolbar, click **Optimization** and choose **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 `50`.

Generally, there is no point-converging topology optimization in a strict sense, because the topology stops changing much earlier.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Mass Properties 1** > **comp1.mass1.mass - Mass - kg**.

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

6 Locate the **Control Variables** section. In the table, clear the **Solve for** checkbox for **Polynomial Boundary 1**.

7 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **comp1.pnt_disp - Tip Displacement - m**.

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

Expression	Lower bound	Upper bound
`comp1.pnt_disp/maxDisp`		1

9 Click to expand the **Output** section. From the **Probes** list, choose **None**.

10 Select the **Plot** checkbox.

11 In the table, enter the following settings:

Plot group	Plot window
Topology Optimization	Graphics

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

RESULTS

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

STUDY 2: SHAPE OPTIMIZATION

Shape Optimization

Disable the **Density Module** control variables in the **Shape Optimization** study.

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

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

3 In the table, clear the **Solve for** checkbox for **Density Model 1 (dtopo1)**.

STUDY 1: PARAMETER OPTIMIZATION

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Parameter Optimization** click **Step 1: Stationary**.

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1) > Topology Optimization**.

5 Click **Disable in Solvers**.

6 In the tree, select **Component 1 (comp1) > Deformed Geometry, Controls material frame**.

7 Click **Disable in Solvers**.

STUDY 2: SHAPE OPTIMIZATION

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

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1) > Topology Optimization**.

5 Click **Disable in Solvers**.