Created in COMSOL Multiphysics 6.0

# Design Optimization of a Beam

## *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, it is in no way guaranteed. The optimization can get stuck in 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 it is the lower right corner that is moved, while the shape optimization changes the bottom right boundary. Finally, topology optimization is able to distribute material freely in the design domain.

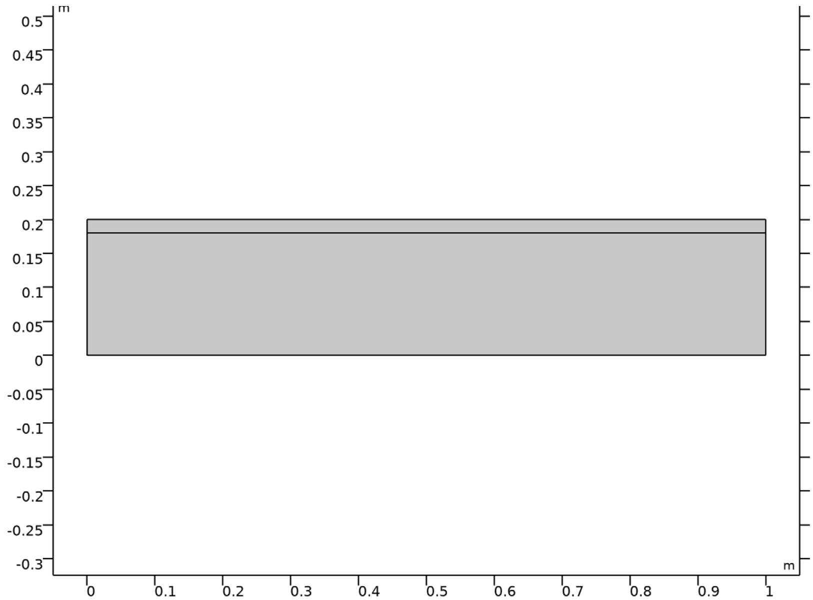


*Figure 1: The model geometry.*

The beam is made of aluminum, and the displacement field is calculated under the assumption of linear elasticity. The displacement of the upper right corner is constrained to be less than 0.2 mm.

The parameter optimization moves the lower right point 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 to 2nd order Bernstein polynomials to move the bottom right 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 points are that Young's modulus varies spatially to reflect the material distribution. It is not possible to set zero void stiffness, because this causes the void displacement field to become undefined.

## *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 end of the beam,

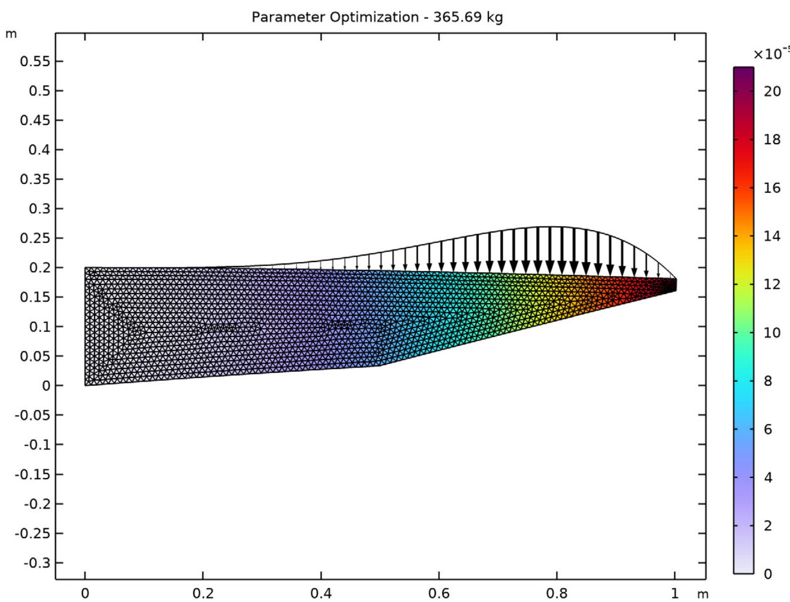


*Figure 2: The lower right corner is moved to the center of the beam, when its position is used as a control variable. The color scale show reveals that the displacement at the upper right corner satisfies the 0.5 mm constraint.*

The shape optimization results in a somewhat similar design and the mass is only 1% lower, see Figure 3. However, the study is faster than the parameter optimization 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 have affect the computational time adversely.

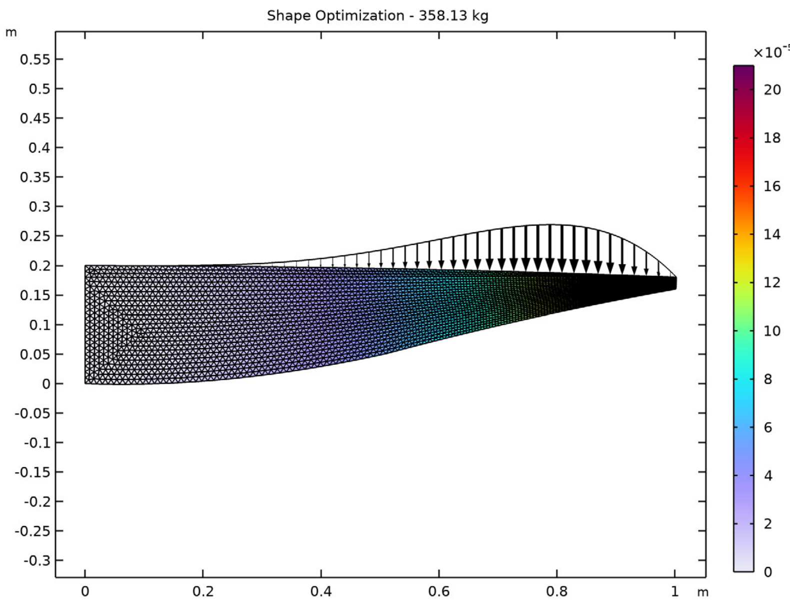


*Figure 3: The shape of the beam is optimized by moving the lower boundary vertically. The movement is large toward the right of the beam, which causes a concentration of elements in this area.*

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

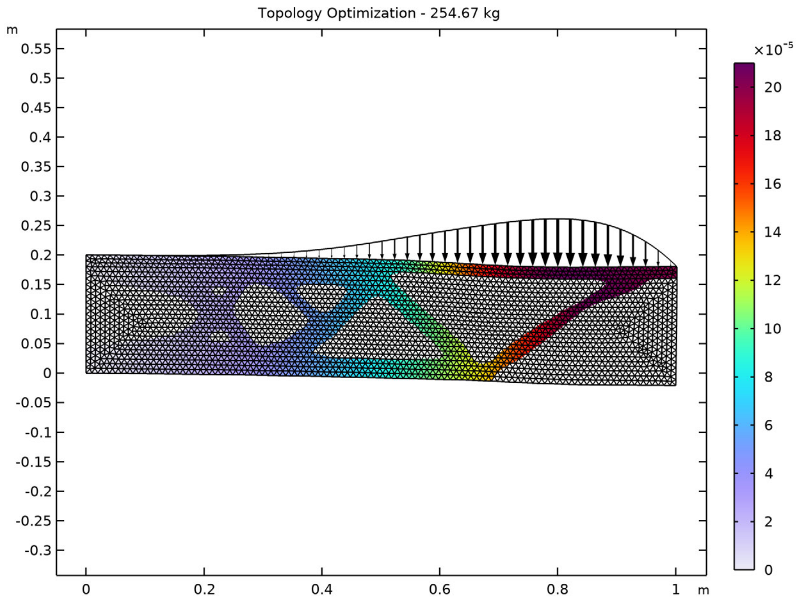


*Figure 4: Topology optimization has significant design freedom, so it is able to find a design, which is 41% lighter than the result of the shape optimization.*

## *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 under the component's **Definitions** node. Both the shape and parameter optimization 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 under component definitions. This is used to define a custom Young's modulus for a new Linear Elasticity model under the Solid Mechanics interface. This new Linear Elasticity model is disabled in the studies for the shape and parameter optimization.

**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 | 0 | Y position, point 1 |
| Yopt2 | 0 | 0 | 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 **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>Aluminum**.

**4** Click **Add to Component** in the window toolbar.

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

## SOLID MECHANICS (SOLID)

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

### *Fixed Constraint 1*

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

**5** Specify the $\mathbf{F}_\mathrm{L}$ vector as

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

## MESH 1

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

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

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

**4** Click **Build All**.

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 `Parameter Optimization` in the **Label** text field.
6 In the **Home** 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`.
4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

*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** check box.
5 In the **Minimum** text field, type `0`.
6 In the **Maximum** text field, type `1.05*maxDisp`.

*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** check box.
4 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** check box.
4 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.F_Ax,solid.F_Ay - Load (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. Select the **Scale factor** check box.

6 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** check box.

7 In the associated text field, type `1e-6`.

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

9 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** check box.

4 In the associated text field, type `100`.

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

## PARAMETER OPTIMIZATION

Setup and compute the solution to the parameter optimization problem.

*Optimization*

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

2 In the **Settings** window for **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 **Objective Function** section. From the **Objective scaling** list, choose **Initial solution based**.

6 Locate the **Control Variables and Parameters** section. Click + **Add** twice.

7 In the table, enter the following settings:

| Parameter name | Initial value | Scale | Lower bound | Upper bound |
|---|---|---|---|---|
| Yopt1 (Y position, point 1) | 0 | 1 | 0 | 0.9*T0 |
| Yopt2 (Y position, point 2) | 0 | 1 | 0 | 0.9*T0 |

8 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**.

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

| Expression | Lower bound | Upper bound | Evaluate for |
|---|---|---|---|
| comp1.pnt_disp/maxDisp | | 1 | Stationary |

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

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

12 Locate the **Optimization Solver** section. Find the **Solver settings** subsection. From the **Keep solutions** list, choose **Only last**.

13 In the **Model Builder** window, click **Parameter Optimization**.

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

15 Clear the **Generate default plots** check box.

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

## RESULTS

### *Parameter Optimization*

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)

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

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

### SHAPE OPTIMIZATION

*Free Shape Domain 1*

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

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

*Polynomial Boundary 1*

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

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

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

4 Locate the **Control Variable Settings** section. In the $d_{max}$ text field, type `0.9*T0`.

*Symmetry/Roller 1*

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization>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 **Add Study** 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** check box.
6. In the **Label** text field, type `Shape Optimization`.
7. In the **Home** 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`.
4. Locate the **Data** section. From the **Dataset** list, choose **Shape Optimization/ Solution 4 (sol4)**.

## PARAMETER OPTIMIZATION

*Optimization*

In the **Model Builder** window, under **Parameter Optimization** right-click **Optimization** and choose **Copy**.

## SHAPE OPTIMIZATION

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

*Optimization*

1. In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
2. From the **Method** list, choose **IPOPT**.
3. Locate the **Control Variables and Parameters** section. Ctrl-click to select table rows 1 and 2.
4. Click **Delete**.

5 Locate the **Output While Solving** section. From the **Plot group** list, choose **Shape Optimization**.

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

## RESULTS

### *Shape Optimization*

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

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

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

## COMPONENT 1 (COMP1)

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

1 In the **Definitions** toolbar, click **Optimization** and choose **Topology Optimization>Density Model**.

## TOPOLOGY OPTIMIZATION

### *Density Model 1 (dtopo1)*

1 Select Domain 1 only.

2 In the **Settings** window for **Density Model**, locate the **Control Variable Discretization** section.

3 From the **Element order** list, choose **Constant**.

### *Prescribed Material 1*

1 In the **Definitions** toolbar, click **Optimization** and choose **Topology Optimization>Prescribed Material**.

2 Select Domain 2 only.

## SOLID MECHANICS (SOLID)

### *Linear Elastic Material 2*

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

4 From the $E$ list, choose **User defined**. In the associated text field, type `dtopo1.theta_p*mat1.Enu.E`.

It is sufficient to interpolate the Young's modulus, if the `theta_avg` variable of the **Density Model** feature is used as objective function, but we also have to interpolate the density, if we want to continue using the mass computed by the **Mass Properties** features as objective function. Otherwise the **Density Model** feature will be unable to change the mass/objective function.

5 From the ρ list, choose **User defined**. In the associated text field, type `mat1.def.rho*dtopo1.theta`.

### 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 **Add Study** in the window toolbar.

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

### TOPOLOGY OPTIMIZATION

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

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

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

*Step 1: Stationary*

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

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

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

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

5 In the **Home** 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`.
4. Locate the **Data** section. From the **Dataset** list, choose **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`.

## SHAPE OPTIMIZATION

### *Optimization*

In the **Model Builder** window, under **Shape Optimization** right-click **Optimization** and choose **Copy**.

## TOPOLOGY OPTIMIZATION

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

### *Optimization*

1. In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
2. From the **Method** list, choose **MMA**.
3. Find the **Solver settings** subsection. In the **Maximum number of model evaluations** text field, type `100`.

   Generally there is no point converging topology optimization in a strict sense, because the topology stops changing much earlier.
4. Locate the **Control Variables and Parameters** section. In the table, clear the **Solve for** check box for **Polynomial Boundary 1**.
5. Locate the **Output While Solving** section. From the **Plot group** list, choose **Topology Optimization**.

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

### RESULTS

*Topology Optimization*

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

### PARAMETER OPTIMIZATION

*Optimization*

Disable the **Density Module** control variables in the **Parameter Optimization** and **Shape Optimization** studies.

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

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

**3** In the table, clear the **Solve for** check boxes for **Density Model 1 (dtopo1)** and **Polynomial Boundary 1**.

### SHAPE OPTIMIZATION

*Optimization*

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

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

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

### PARAMETER OPTIMIZATION

*Step 1: Stationary*

**1** In the **Model Builder** window, under **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** check box.

**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**.

**7** Click **Disable in Solvers**.

**8** In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Linear Elastic Material 2**.

9 Click **Disable**.

#### SHAPE OPTIMIZATION

*Step 1: Stationary*

1 In the **Model Builder** window, under **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** check box.

4 In the tree, select **Component 1 (Comp1)>Topology Optimization**.

5 Click **Disable in Solvers**.

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Linear Elastic Material 2**.

7 Click **Disable**.