Created in COMSOL Multiphysics 5.6

Exporting and Importing a Topology-Optimized Hook

Introduction

Topology optimization in a structural mechanics context can answer the question: Given that you know the loads on the structure, which distribution of the available material maximizes stiffness? Or, conversely, how much material is necessary to obtain a predefined stiffness, and how should it be distributed? Such investigations typically occur during the concept design stages.

The conflicting goals of stiffness maximization and mass minimization lead to a continuum of possible optimal solutions, depending on how you balance the goals against each other. This topology optimization example demonstrates how to use a penalization method (SIMP) to obtain an optimal distribution of a fixed amount of material such that stiffness is maximized. Changing the amount of material available leads to a different solution, which is also Pareto optimal, representing a different balance between the conflicting objectives.

Model Definition

The model studies optimal material distribution in a hook, which is symmetric about the plane $z = 0$ and consists of a linear elastic material, structural steel. It is subjected to two distributed load cases: One at the tip of the hook and one along its lower inner curve. The geometry is imported as a geometry sequence (Figure 1).

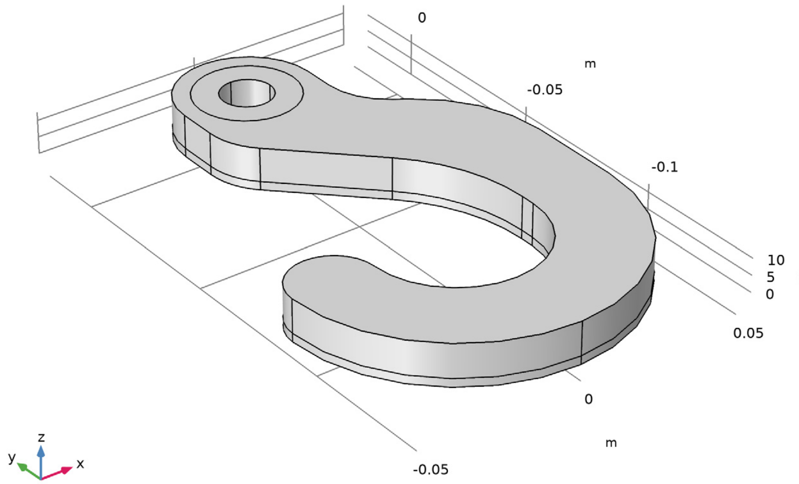


Figure 1: The computational domain of the hook is reduced by exploiting symmetry.

The optimality criterion is defined by the objective function, which is chosen to be the total strain energy in this example. Note that the strain energy exactly balances the work done by the applied load, so minimizing the strain energy minimizes the displacement induced at the points where loads are applied, effectively minimizing the compliance of the structure — maximizing its stiffness. The other, conflicting, objective is minimization of total mass, which is implemented as an upper bound on the mass of the optimized structure.

To regularize the problem we introduce a minimum length scale via a filter radius in a Helmholtz filter. See the Topology Optimization of an MBB Beam example for illustrations of this process. Ideally, the topology of the resulting designs should be mesh independent and for designs with moderate complexity this can be achieved, but if the optimal design is very complex, other designs with slightly different topologies perform only marginally worse, so the optimization problem has many local minima, and it is likely that a slightly suboptimal design is identified. The result of the optimization is shown in Figure 2.

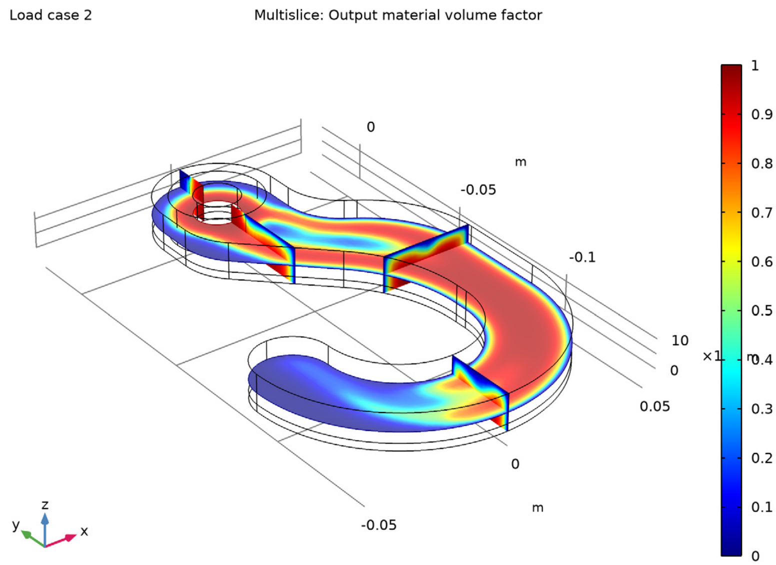


Figure 2: A multislice plot colored with the displacement magnitude shows the topology optimized design before it is exported.

This example demonstrates a strategy based on using MMA with a limited number of iterations. The filtered variable is solved on a finer mesh to get a smooth design for postprocessing. You can use this to export a STL file, but to transfer the geometry to a 2nd component for verification purposes, it is best to use a **Filter** dataset. This approach enables the recycling of selections from the other component, but the parameters for the geometry **Import** feature can still require some tweaking to get a valid mesh for computation of the displacements in the imported geometry.

Results

The following plot shows the displacement field for the optimized solution after the geometry has been transferred and re-meshed.

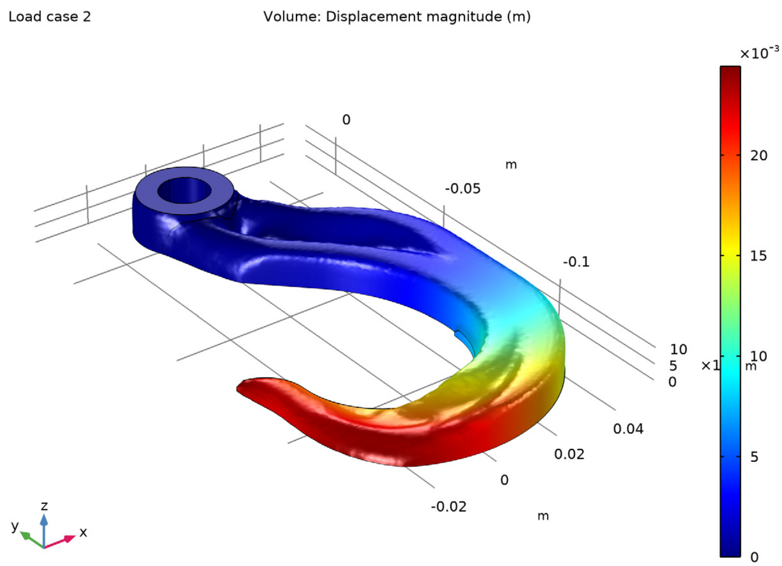


Figure 3: A plot of the displacement field for the optimized design after it has been imported as an STL file.

Notes About the COMSOL Implementation

The control variable field, Helmholtz filter and SIMP penalization are defined through the use of a topology optimization feature. A Solid Mechanics interface represents the structural properties of the beam, while an Optimization interface allows adding the objective and the constraints for the optimization problem. The elastic strain energy density is a predefined variable, `solid.Ws`, available to use as the objective function for the optimization problem.

Only half of the hook geometry must be modeled, because of the assumed symmetry, which is implemented as a Symmetry condition on the symmetry plane. Plots for postprocessing and inspection of the solution while solving are set up using a Mirror data set, to show the complete solution. The geometry is parameterized, making it easy to experiment with different hook sizes, but keep in mind that the mesh size is taken as the filter radius.

The optimization solver is selected and controlled from an Optimization study step. For topology optimization, either the MMA or the SNOPT solver can be used. They have different merits and weaknesses. MMA tends to be braver in the beginning, proceeding quickly toward an approximate optimum, while SNOPT is more cautious but also converges more efficiently close to the final solution thanks to its second-order approximation of the objective.

Finally, it is worth noting that the problem of finding a stiff design can be quite different from finding a design that does not break and it is generally advised to use shape or parameter optimization to ensure that the design is free of excessive stress concentrations.

References

1. B.S. Lazarov and O. Sigmund, "Filters in topology optimization based on Helmholtz-type differential equations," *International Journal for Numerical Methods in Engineering*, vol 86, pp. 765–781, 2011.

2. F. Wang, B.S. Lazarov and O. Sigmund, "On projection methods, convergence and robust formulations in topology optimization," *Structural and Multidisciplinary Optimization*, vol 43, pp. 767–784, 2011.

3. M.P. Bendsøe, "Optimal shape design as a material distribution problem," *Structural Optimization*, vol. 1, pp. 193–202, 1989.

Application Library path: `Optimization_Module/Topology_Optimization/hook_optimization_stl`

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**.

GEOMETRY 1

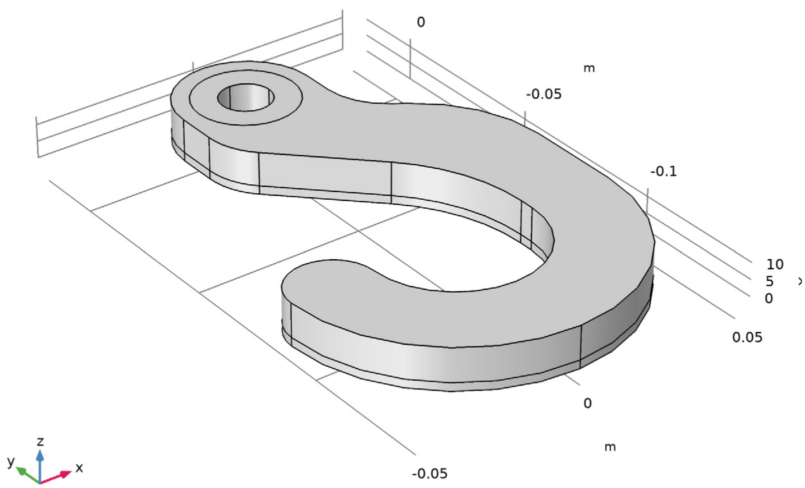
Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `hook_optimization_stl_geom_sequence.mph`.

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

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



The geometry should now look like that in Figure 1. Note that the inserted geometry is parameterized and that the parameters used are automatically added to the list of global parameters in the model.

GLOBAL DEFINITIONS

Geometric Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometric Parameters` in the **Label** text field.

Now there is a group for geometric parameters. Add a second group for parameters related to the optimization.

Parameters 2

1 In the **Home** toolbar, click **Pi** **Parameters** and choose **Add>Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
volfrac	0.5	0.5	Volume fraction
Lmin	2[mm]	0.002 m	Filter radius
meshsz	Lmin	0.002 m	Mesh size
meshsz2	Lmin/2	0.001 m	Fine mesh size

MESH 1

Swept 1

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

2 In the **Settings** window for **Swept**, click to expand the **Sweep Method** section.

3 From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz`.

5 In the **Minimum element size** text field, type `meshsz/4`.

6 Click **Build All**.

Use structural steel for the solid material.

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

Topology Link 1 (toplnk1)

1 In the **Settings** window for **Topology Link**, locate the **Link Settings** section.

2 From the **Material** list, choose **Structural steel (mat1)**.

The Young's modulus is now coupled to the material. Next, set up a mirror dataset for seeing the full design, while optimizing.

DEFINITIONS

Density Model 1 (dtopo1)

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

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

3 From the **Selection** list, choose **Optimized Domains**.

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

5 In the text field, type `Lmin`.

6 Locate the **Control Variable Discretization** section. From the **Element order** list, choose **Constant**.

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

Prescribed Material 1

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

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

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

Add a **Material Boundary Density** and **Void Boundary Density** feature to avoid 90 degree angles at domain boundaries. In this case, these features will also cause the optimization to find design without internal cavities.

Prescribed Material Boundary 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Prescribed Material Boundary**.
2 In the **Settings** window for **Prescribed Material Boundary**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Load Boundaries**.

Prescribed Void Boundary 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Prescribed Void Boundary**.
2 In the **Settings** window for **Prescribed Void Boundary**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Free Boundaries**.

Variables 1

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

In topology optimization the geometry is not sufficiently well defined to warrant 2nd order shape functions.

SOLID MECHANICS (SOLID)

1 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.
2 From the **Displacement field** list, choose **Linear**.

Fixed Constraint 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)** and choose **Fixed Constraint**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Hole**.

Symmetry 1

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

Take both loads into account using the load case functionality.

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 **1st 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

6 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

Boundary Load 2

1 In the **Model Builder** window, right-click **Boundary Load 1** and choose **Duplicate**.

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

3 From the **Selection** list, choose **2nd Load Boundary**.

4 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

OPTIMIZATION

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

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

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Define load cases** check box.

4 Click **Add**.

5 Click **Add**.

6 In the table, enter the following settings:

Load case	lg1	Weight	lg2	Weight
Load case 1	√	1.0		1.0
Load case 2		1.0	√	1.0

Initialize the study to generate plots to show while solving.

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

RESULTS

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Filter**.

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

Threshold

1 In the **Model Builder** window, expand the **Results>Topology Optimization** node, then click **Threshold**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

OPTIMIZATION

Topology Optimization

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

The control variable will not converge to the tolerance in so few iterations, but the actual design will not change noticeably by using more iterations.

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 **Threshold**.

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

The optimization has removed material near the tip of the hook.

RESULTS

Output material volume factor

1 In the **Model Builder** window, under **Results>Topology Optimization** click **Output material volume factor**.

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

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

Add a finer mesh and solve for the filtered material volume factor on this.

MESH 1

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

MESH 2

Size

1 In the **Model Builder** window, expand the **Mesh 2** node, then click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.
3 In the **Maximum element size** text field, type `meshsz2`.
4 In the **Minimum element size** text field, type `meshsz2/4`.
5 Click **Build All**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.
5 Click **Add Study** in the window toolbar.
6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

SMOOTH DESIGN (MESH2)

1 In the **Model Builder** window, click **Study 2**.
2 In the **Settings** window for **Study**, type `Smooth Design (mesh2)` in the **Label** text field.

Solution 2 (sol2)

In the **Study** toolbar, click **Show Default Solver**.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Smooth Design (mesh2)>Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
3 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Optimization, Stationary**.

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

RESULTS

Filter 1

Create a new component from the filter dataset.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

MESH 3

Import 1

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

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

3 From the **Source** list, choose **Dataset**.

4 From the **Boundary partitioning** list, choose **Minimal**.

5 Click **Import**.

Adapt 1

1 In the **Mesh** toolbar, click **Modify** and choose **Elements>Adapt**.

2 In the **Settings** window for **Adapt**, locate the **Adaptation** section.

3 From the **Solution** list, choose **None**.

4 From the **Type of expression** list, choose **Absolute size**.

5 In the **Size expression** text field, type `meshsz/2`.

6 In the **Maximum number of refinements** text field, type `0`.

7 In the **Maximum coarsening factor** text field, type `Inf`.

8 Click **Build Selected**.

Free Tetrahedral 1

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

Size

1 In the **Model Builder** window, expand the **Free Tetrahedral 1** node, then click **Size**.

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

3 From the **Predefined** list, choose **Finer**.

4 Click **Build All**.

Use structural steel for the solid material.

MATERIALS

Material Link 1 (matlnk1)

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

SOLID MECHANICS (SOLID)

Copy/paste the physics from the first component and fix the selections.

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

SOLID MECHANICS (SOLID2)

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

2 In the **Messages from Paste** dialog box, click **OK**.

We can use 2nd order displacements now that we have an explicit geometry representation.

3 In the **Model Builder** window, click **Solid Mechanics (solid2)**.

4 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

5 From the **Displacement field** list, choose **Quadratic serendipity**.

Fixed Constraint 1

1 In the **Model Builder** window, expand the **Solid Mechanics (solid2)** node, then click **Fixed Constraint 1**.

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

3 From the **Selection** list, choose **Hole**.

Symmetry 1

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

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

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

Boundary Load 1

1 In the **Model Builder** window, click **Boundary Load 1**.

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

3 From the **Selection** list, choose **1st Load Boundary**.

Boundary Load 2

1 In the **Model Builder** window, click **Boundary Load 2**.

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

3 From the **Selection** list, choose **2nd Load Boundary**.

ADD STUDY

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

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

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

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

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

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

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

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 **Solid Mechanics (solid2)**.

SMOOTH DESIGN (MESH2)

Step 1: Stationary

1 In the **Model Builder** window, under **Smooth Design (mesh2)** 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 **Solid Mechanics (solid2)**.

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 1

4 Locate the **Study Extensions** section. Select the **Define load cases** check box.

5 Click **Add**.

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lg1	Weight	lg2	Weight
`Load case 1`	√	`1.0`		`1.0`
`Load case 2`		`1.0`	√	`1.0`

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

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

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

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

Add a new 3D plot group to plot the displacement.

RESULTS

Displacement (solid2)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement (solid2)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Verification/Solution 3 (4) (sol3)**.

Volume 1

Right-click **Displacement (solid2)** and choose **Volume**.

Displacement (solid2)

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

2 In the **Displacement (solid2)** toolbar, click **Plot**.

Topology Optimization 1

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