Created in COMSOL Multiphysics 5.6

Bracket — Topology Optimization

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.

The various examples based on a bracket geometry form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module. This example also requires the Optimization Module.

Model Definition

The model shows how to determine the optimal material distribution in a bracket geometry. The bracket is symmetric about the plane $x = 0$ and is made of a linear elastic material, structural steel.

It is optimized in two load configurations: First a single load case, which gives rise to an asymmetric design with two poorly connected halves. In a second analysis, eight load cases are used (two load groups times four constraint groups). This results in a symmetric design with a stronger connection between the two halves.

The original geometry, which can be considered as the design space, is imported as a geometry sequence (Figure 1).

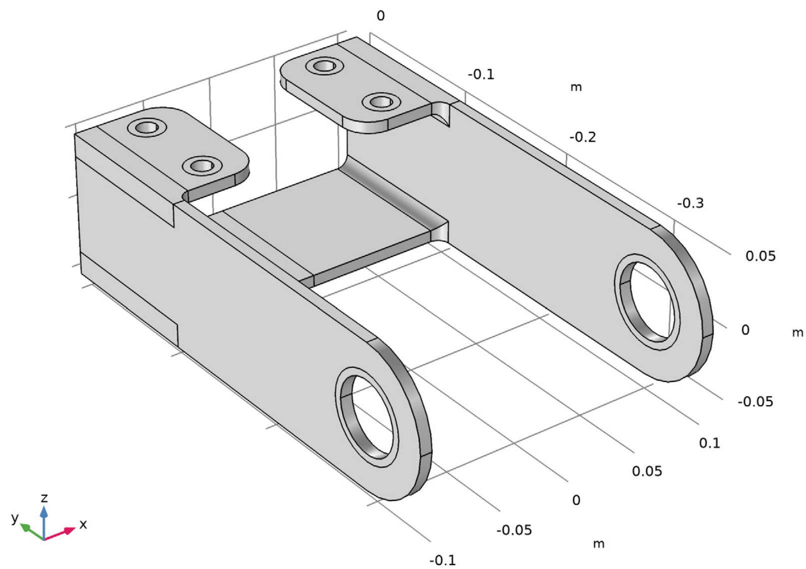


Figure 1: The computational domain of the bracket is partitioned, so that the circles around the holes can be excluded from the design space.

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. 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. To reduce the probability of this happening, the model is solved using continuation in the SIMP exponent and the projection slope, that is, a sequence of optimization problems are solved. One optimization result is thus used as the initial guess of the next. This is achieved by combining a **Parametric Sweep** and an **Optimization** study step.

Results

The result of the optimization with one load case is shown in Figure 2.

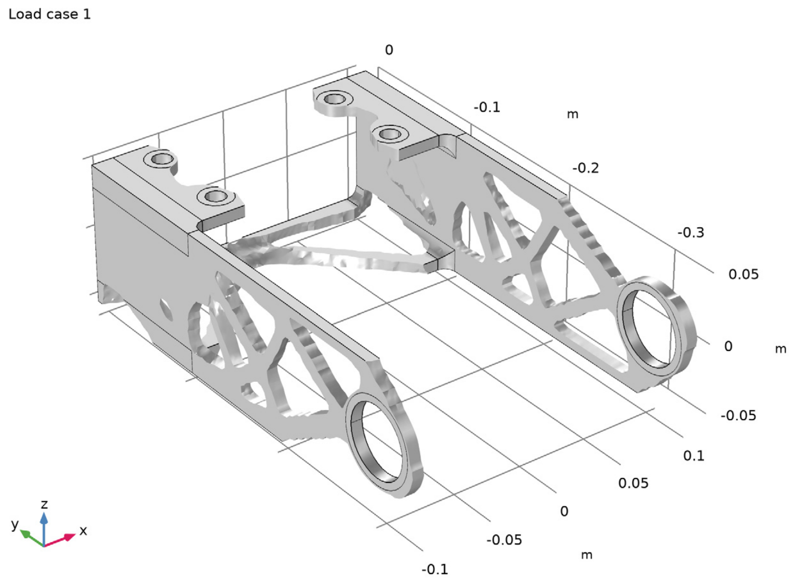


Figure 2: A filtered volume plot, colored with the displacement magnitude. Only a single load case is considered, so the design becomes asymmetric and the two halves are poorly connected.

When eight load cases are considered, the optimization instead results in the design displayed in Figure 3.

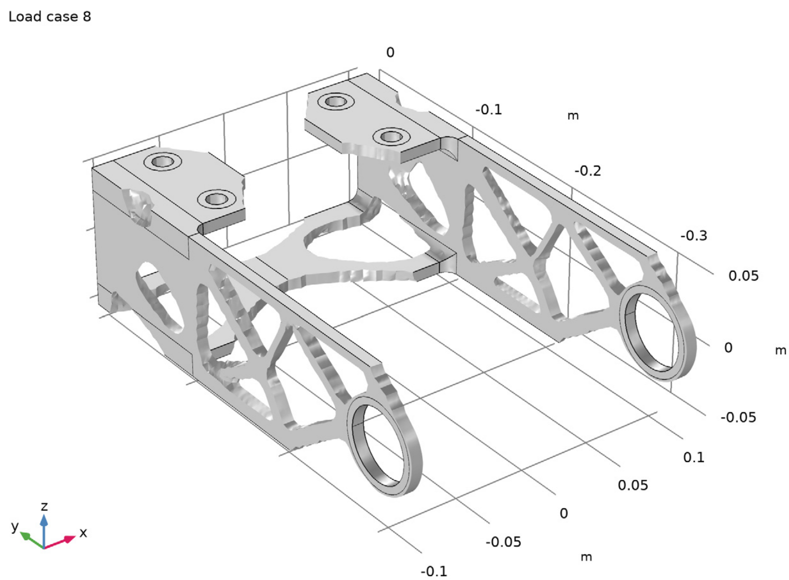


Figure 3: When eight load cases are considered, the design becomes symmetric, and the connection between the two halves is stronger when compared to the single load case.

The following plot shows the displacement field for the optimized solution after the geometry has been transferred to another component via a **Filter dataset**.

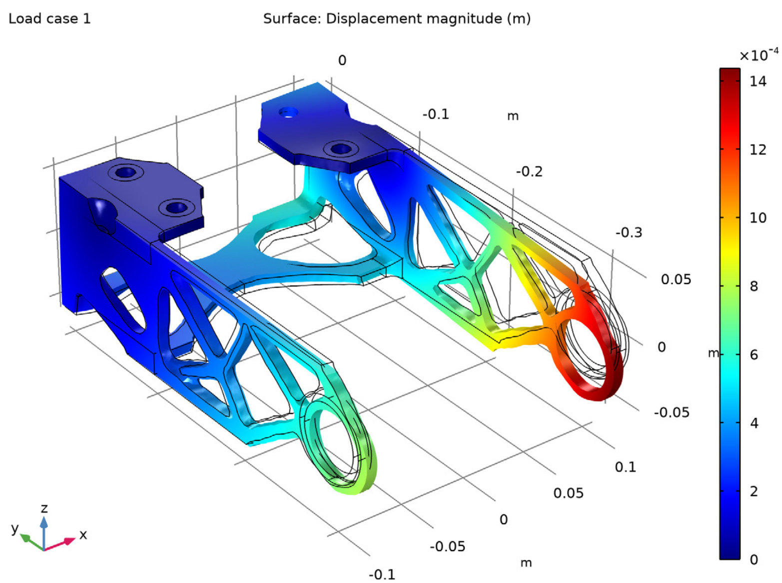


Figure 4: A plot of the displacement field for the optimized design after it has been transferred to another component using a Filter dataset, a mesh part and a Import geometry feature. The hole furthest away from the camera is unconstrained for Load case 1.

Notes About the COMSOL Implementation

This example demonstrates a strategy based on using the MMA solver with a limited number of iterations. The filtered variable is solved on a finer mesh to get a smooth design for postprocessing.

The example makes use of box selections for imposing the boundary conditions on the imported geometry. This can be a significant time saver in the context of topology optimization, because many designs with similar boundary conditions can quickly be generated and tested against each other.

This model demonstrates how to export all domains together. This simplifies the import, but the Mesh part describes curved boundaries with flat triangles, so the hole geometry will be affected in this process, which might affect a contact analysis.

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 bracket, while an Optimization study step 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.

In this example the solver settings are changed slightly from their default values, in order to speed up the solution:

- Since the solid mechanics problem is linear, it is enough to use one pass through the segregated solver, so it is set to terminate after one iteration, rather than on a tolerance.
- When the constraints are changed, the structure of the stiffness matrix will change. For this reason, the caching of sparsity patterns is switched off.
- The PARDISO solver is used rather than the MUMPS solver. The former is in general faster, but can be less accurate for ill-conditioned problems. For a case like this, there is no need for the additional potential accuracy.

Finally, it is worth noting that the problem of finding a stiff design can be quite different from finding a design that does not fail, 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/bracket_topology_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**.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `bracket_topology_optimization_stl_parameters.txt`.

GEOMETRY 1

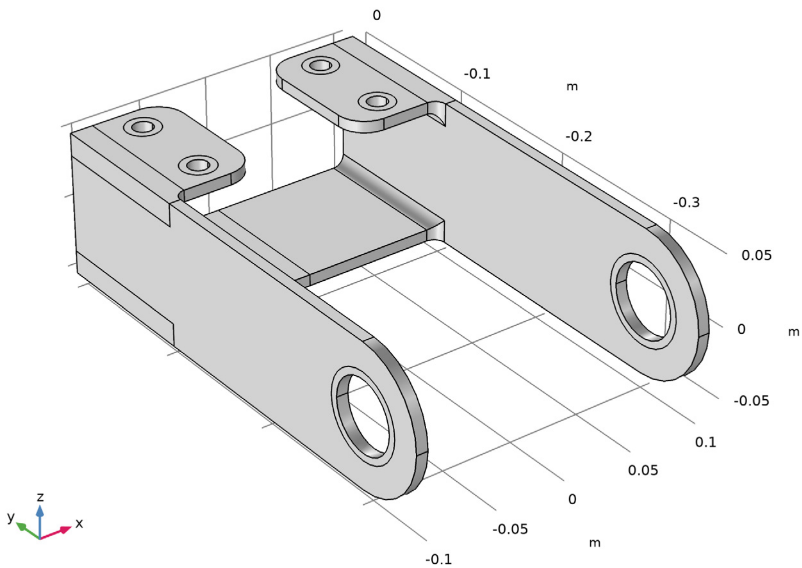
Create the geometry. To simplify this step, insert a prepared geometry sequence.

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

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

5 In the **Model Builder** window, collapse the **Geometry 1** node.

GLOBAL DEFINITIONS

Geometrical Parameters

Add a new parameter group and a function for imposing the forces on the two large holes. The forces correspond to a torque on the bracket.

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

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

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
`Lmin`	`5 [mm]`	0.005 m	`Filter radius`
`meshsz`	`Lmin`	0.005 m	`Mesh size`
`meshsz2`	`Lmin/2`	0.0025 m	`Fine mesh size)`
`P0`	`2.5 [MPa]`	2.5E6 Pa	`Peak load intensity`

DEFINITIONS

Analytic 1 (an1)

1. In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.
2. In the **Settings** window for **Analytic**, type `load` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.
4. In the **Arguments** text field, type `F,px,py`.
5. Locate the **Units** section. In the **Arguments** text field, type `Pa, m, m`.
6. In the **Function** text field, type `Pa`.

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**.
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)**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **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 **Constraint 1**.

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

Fixed Constraint 2

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

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

3 From the **Selection** list, choose **Constraint 2**.

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

Fixed Constraint 3

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

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

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

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

Fixed Constraint 4

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

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

3 From the **Selection** list, choose **Constraint 4**.

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

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 1**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
load(-P0,Y-YC,Z)	n

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

MESH 1

Mesh one side of the domain using swept and tetrahedral meshes.

Free Triangular 1

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

2 Select Boundaries 1, 4, 33, 37, 43, and 47 only.

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/2`.

Swept 1

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

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

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

4 Select Domains 1, 2, and 5–8 only.

Free Tetrahedral 1

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

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

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

4 Select Domains 3 and 4 only.

Copy the mesh from one side to the other.

Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Domain**.

2 Select Domains 1–8 only.

3 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

4 Select the **Activate Selection** toggle button.

5 Select Domains 9–16 only.

Compute a characteristic elastic strain energy.

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 **Study Extensions** section.

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

4 Click **Add**.

5 In the table, enter the following settings:

Load case	lg1	Weight	cg1	cg2	cg3	cg4
Load case 1	√	1.0	√	√	√	√

GLOBAL DEFINITIONS

Parameters 2

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 2**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Vfrac	0.5	0.5	Volume fraction
simpP	1	1	SIMP exponent
proj_beta	1	1	Projection slope

The characteristic strain energy is taken as double the computed value to compensate for the fact that half of the material will be removed.

Set up the density topology feature and let it control Young's modulus.

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 **Filtering** section.

3 From the $R_{\min}$ list, choose **User defined**.

4 In the text field, type `meshsz`.

The filter radius must be larger or equal to the mesh size.

5 Locate the **Projection** section. From the **Projection type** list, choose **Hyperbolic tangent projection**.

6 In the β text field, type `proj_beta`.

7 Locate the **Interpolation** section. From the p_{SIMP} list, choose **User defined**.

8 In the text field, type `simpP`.

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

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

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 **Domains Outside Design Space**.

STUDY 1

Topology Optimization

1 In the **Model Builder** window, under **Study 1** 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 `50`.

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

Expression	Lower bound	Upper bound
comp1.dtopo1.theta_avg		Vfrac

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

6 In the **Settings** window for **Study**, type `One Load Case` in the **Label** text field.

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

Use the plot to show the design during the optimization.

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

9 In the **Settings** window for **Topology Optimization**, locate the **Output While Solving** section.

10 Select the **Plot** check box.

11 From the **Plot group** list, choose **Threshold**.

In general it is best to scale the objective function with the initial value, but in this case the scale is close enough to 1 that it is unnecessary.

12 In the **Model Builder** window, click **One Load Case**.

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

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add** twice.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
simpP (SIMP exponent)	1 2 3 4	
proj_beta (Projection slope)	2 4 6 8	

5 Click to expand the **Advanced Settings** section. Select the **Reuse solution from previous step** check box.

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

RESULTS

One Load Case

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

2 In the **Settings** window for **Group**, type `One Load Case` in the **Label** text field.

Threshold

1 In the **Model Builder** window, expand the **One Load Case** node, then click **Threshold**.

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

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

The resulting design is asymmetric due to the load and the two halves being poorly connected. If one of the bolt breaks, significant stiffness will be lost. Add a second load case together with a study using two load groups and four constraints giving a total of eight load cases. Run the optimization again to see what effect this has on the optimal design.

SOLID MECHANICS (SOLID)

Boundary Load 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** 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 **Load 2**.

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

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

Add the eight load cases with the load groups changing most frequently. The purpose is to avoid unnecessary updates of the stiffness matrix by first changing the right hand side.

1 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

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

3 Click **Add** eight times.

4 In the table, enter the following settings:

Load case	lg1	Weight	lg2	Weight	cg1	cg2	cg3	cg4
Load case 1	√	1.0		1.0		√	√	√
Load case 2		1.0	√	1.0		√	√	√
Load case 3	√	1.0		1.0	√		√	√
Load case 4		1.0	√	1.0	√		√	√
Load case 5	√	1.0		1.0	√	√		√

Load case	lg1	Weight	lg2	Weight	cg1	cg2	cg3	cg4
Load case 6		1.0	√	1.0	√	√		√
Load case 7	√	1.0		1.0	√	√	√	
Load case 8		1.0	√	1.0	√	√	√	

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

6 In the **Settings** window for **Study**, type `Eight Load Cases` in the **Label** text field.

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

ONE LOAD CASE

Parametric Sweep, Topology Optimization

1 In the **Model Builder** window, under **One Load Case**, Ctrl-click to select **Parametric Sweep** and **Topology Optimization**.

2 Right-click and choose **Copy**.

EIGHT LOAD CASES

Parametric Sweep

In the **Model Builder** window, right-click **Eight Load Cases** and choose **Paste Multiple Items**.

Topology Optimization

1 In the **Settings** window for **Topology Optimization**, locate the **Objective Function** section.

2 In the table, enter the following settings:

Expression	Description
comp1.solid.Ws_tot/8	

3 Locate the **Output While Solving** section. From the **Plot group** list, choose **Threshold 1**.

4 In the **Model Builder** window, click **Eight Load Cases**.

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

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

The default solver works, but the computational time can be decreased by using the PARDISO solver and avoiding reuse of the sparsity pattern. Finally, the log becomes cleaner if the solver is not allowed to change the order of the loadcases.

Solution 7 (sol7)

1 In the **Model Builder** window, right-click **Solver Configurations** and choose **Reset Solver to Default**.

2 Expand the **Solution 7 (sol7)** node.

3 In the **Model Builder** window, expand the **Eight Load Cases>Solver Configurations>Solution 7 (sol7)>Optimization Solver 1>Stationary 1** node, then click **Segregated 1**.

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

5 From the **Termination technique** list, choose **Iterations**.

6 In the **Model Builder** window, click **Direct**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

9 In the **Model Builder** window, click **Suggested Direct Solver (solid)**.

10 In the **Settings** window for **Direct**, locate the **General** section.

11 From the **Solver** list, choose **PARDISO**.

12 In the **Model Builder** window, click **Advanced**.

13 In the **Settings** window for **Advanced**, click to expand the **Assembly Settings** section.

14 Clear the **Reuse sparsity pattern** check box.

15 In the **Model Builder** window, click **Parametric**.

16 In the **Settings** window for **Parametric**, locate the **General** section.

17 From the **Parameter value run order** list, choose **As specified**.

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

RESULTS

Applied Loads (solid), Applied Loads (solid) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Applied Loads (solid)** and **Applied Loads (solid) 1**.

2 Right-click and choose **Delete**.

Eight Load Cases

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

2 In the **Settings** window for **Group**, type `Eight Load Cases` in the **Label** text field.

Threshold 1

1 In the **Model Builder** window, expand the **Eight Load Cases** node, then click **Threshold 1**.

2 In the **Threshold 1** toolbar, click **Plot**.

Now the design is symmetric and the two halves are better connected.

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

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

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 `meshsz/2`.

4 In the **Minimum element size** text field, type `meshsz/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 3**.

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

Solution 13 (sol13)

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

Step 1: Stationary

1 In the **Model Builder** window, click **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 **Eight Load Cases, Stationary**.

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

RESULTS

Filter 1

Create a new component from the filter dataset.

ADD COMPONENT

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

2 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 **Dataset** list, choose **Filter 2**.

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

6 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`.

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 **Extremely fine**.
4 Click **Build All**.

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 **Constraint 1**.

Fixed Constraint 2

1 In the **Model Builder** window, click **Fixed Constraint 2**.

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

3 From the **Selection** list, choose **Constraint 2**.

Fixed Constraint 3

1 In the **Model Builder** window, click **Fixed Constraint 3**.

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

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

Fixed Constraint 4

1 In the **Model Builder** window, click **Fixed Constraint 4**.

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

3 From the **Selection** list, choose **Constraint 4**.

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 **Load 1**.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
comp1.load(-P0,Y-YC,Z)	z

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 **Load 2**.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
comp1.load(-P0,Y-YC,Z)	z

ADD STUDY

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

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

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

The filtered material volume factor is always solved for, but this takes less time on the coarse mesh.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

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

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

VERIFICATION

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

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

ONE LOAD CASE

Step 1: Stationary

1 In the **Model Builder** window, under **One Load Case** 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)**.

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)**.

EIGHT LOAD CASES

Step 1: Stationary

1 In the **Model Builder** window, under **Eight Load Cases** 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)**.

4 Right-click **Eight Load Cases>Step 1: Stationary** and choose **Copy**.

VERIFICATION

In the **Model Builder** window, right-click **Verification** and choose **Paste Stationary**.

Step 1: Stationary

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

2 In the table, select the **Solve for** check box for **Solid Mechanics (solid2)**.

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

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

RESULTS

Topology Optimization

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

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 14 (7) (sol14)**.

Surface 1

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

Deformation 1

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

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

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