

# 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, four load cases are used (two load groups and 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.

Load case 1



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





Figure 3: When four 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.

Continuation is used in the SIMP exponent as well as the projection slope. This is achieved with a **Parametric Sweep**, where the option **Reuse solution from previous step** has been enabled.

This model demonstrates how to export all domains together. This simplifies the import, but the curved boundaries will be described 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 **Density Model** feature. A Solid Mechanics interface represents the structural properties of the bracket, while the objective and the constraints are defined directly in the

**Topology Optimization** study step. 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 GMRES solver is used rather than the MUMPS solver, because it is faster.

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 Helmholtztype 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

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).
- 3 Click Add.
- 4 Click  $\bigcirc$  Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces> Optimization>Topology Optimization, Stationary.
- 6 Click **M** Done.

# GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

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

- I In the Geometry toolbar, click Insert Sequence and choose 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 4 **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 I 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.

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, type Geometrical Parameters in the Label text field.

# Parameters 2

- I 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 I (an I)

- I 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 table, enter the following settings:

Argument	Unit
F	Ра
рх	m
РУ	m

6 In the Function text field, type Pa.

## ADD MATERIAL

- I 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 Right-click and choose Add to Global Materials.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

# MATERIALS

Topology Link I (topInkI)

- I In the Settings window for Topology Link, locate the Link Settings section.
- 2 From the Material list, choose Structural steel (matl).

#### SOLID MECHANICS (SOLID)

#### Fixed Constraint I

- I In the Model Builder window, under Component I (compl) 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 I.
- **4** In the **Physics** toolbar, click **() Constraint Group** and choose **New Constraint Group**.

#### Fixed Constraint 2

- I 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

- I 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 (**o**) Constraint Group and choose New Constraint Group.

#### Fixed Constraint 4

- I 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

- I 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 I**.
- 4 Locate the Coordinate System Selection section. From the Coordinate system list, choose Boundary System I (sys1).
- ${\bf 5}\;$  Locate the Force section. Specify the  ${\bf F}_A$  vector as

0 tl

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

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

# MESH I

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

Free Triangular 1

- I In the Mesh toolbar, click  $\bigwedge$  Boundary and choose Free Triangular.
- **2** Select Boundaries 1, 4, 33, 37, 43, and 47 only.

# Size

- I 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

- I In the Mesh toolbar, click A 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 I

- I In the Mesh toolbar, click \land 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 I

- I In the Model Builder window, right-click Mesh I and choose Copying Operations> Copy Domain.
- **2** Select Domains 1–8 only.

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

**4** Click to select the **E Activate Selection** toggle button.

**5** Select Domains 9–16 only.

Compute a characteristic elastic strain energy.

## STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: 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	lgl	Weight	cgl	cg2	cg3	cg4
Load case 1	$\checkmark$	1.0	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$

# GLOBAL DEFINITIONS

Parameters 2

- I 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	I	SIMP exponent
proj_beta	1	I	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.

#### TOPOLOGY OPTIMIZATION

Density Model I (dtopol)

- I In the Model Builder window, under Component I (compl)>Topology Optimization click Density Model I (dtopol).
- 2 In the Settings window for Density Model, locate the Geometric Entity Selection section.
- 3 From the Selection list, choose Design Space Domains.
- 4 Locate the Filtering section. From the  $R_{\min}$  list, choose User defined.
- 5 In the text field, type meshsz.

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

- 6 Locate the Projection section. From the Projection type list, choose Hyperbolic tangent projection.
- 7 In the  $\beta$  text field, type proj\_beta.
- 8 Locate the Interpolation section. From the  $p_{\text{SIMP}}$  list, choose User defined.
- 9 In the text field, type simpP.
- 10 Locate the Control Variable Discretization section. From the Element order list, choose Constant.
- II Locate the **Control Variable Initial Value** section. In the  $\theta_0$  text field, type Vfrac.

#### COMPONENT I (COMPI)

Prescribed Material I

- I In the Definitions toolbar, click 😥 Optimization and choose Topology Optimization> 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.

#### Prescribed Material Boundary I

- I In the Definitions toolbar, click 💮 Optimization and choose Topology Optimization> Prescribed Material Boundary.
- 2 In the Settings window for Prescribed Material Boundary, locate the Geometric Entity Selection section.
- **3** From the Selection list, choose Material Boundaries.

#### STUDY I

Topology Optimization

- I In the Model Builder window, under Study I 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 I.
- 6 In the Settings window for Study, type One Load Case in the Label text field.
- 7 In the Study toolbar, click  $\underset{t=0}{\bigcup}$  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.
- **IO** Select the **Plot** check box.
- II 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.
- **I3** In the Settings window for Study, locate the Study Settings section.
- **I4** Clear the **Generate default plots** check box.

#### Parametric Sweep

- I 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

- I 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.

Stress (solid)

- I In the Model Builder window, expand the One Load Case node, then click Stress (solid).
- 2 Drag and drop above One Load Case Output material volume factor.

#### Threshold

- I In the Model Builder window, click Threshold.
- 2 In the Threshold toolbar, click 💽 Plot.
- **3** Click the  $\longleftrightarrow$  **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 four load cases. Run the optimization again to see what effect this has on the optimal design.

# SOLID MECHANICS (SOLID)

Boundary Load 2

- I In the Model Builder window, under Component I (compl)>Solid Mechanics (solid) rightclick Boundary Load I 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

- I In the Home toolbar, click  $\stackrel{\sim}{\sim}$  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  $\stackrel{\text{res}}{\longrightarrow}$  Add Study to close the Add Study window.

# STUDY 2

# Step 1: Stationary

Add the four 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.

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

2 Select the Define load cases check box.

**3** Click + **Add** four times.

Load case	lgl	Weight	lg2	Weight	cgl	cg2	cg3	cg4
Load case 1	$\checkmark$	1.0		1.0		$\checkmark$	$\checkmark$	$\checkmark$
Load case 2		1.0	$\checkmark$	1.0	$\checkmark$		$\checkmark$	$\checkmark$
Load case 3	$\checkmark$	1.0		1.0	$\checkmark$	$\checkmark$		$\checkmark$
Load case 4		1.0	$\checkmark$	1.0	$\checkmark$	$\checkmark$	$\checkmark$	

**4** In the table, enter the following settings:

5 In the Model Builder window, click Study 2.

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

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

# ONE LOAD CASE

Parametric Sweep, Topology Optimization

- I In the Model Builder window, under One Load Case, Ctrl-click to select Parametric Sweep and Topology Optimization.
- 2 Right-click and choose Copy.

# FOUR LOAD CASES

## Parametric Sweep

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

#### Topology Optimization

- I 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/4	

- 3 Locate the Output While Solving section. From the Plot group list, choose Threshold I.
- 4 In the Model Builder window, click Four Load Cases.
- 5 In the Settings window for Study, locate the Study Settings section.
- 6 Clear the Generate default plots check box.

The default direct solver works, but the computational time can be decreased by using the iterative 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 load cases.

#### Solution 7 (sol7)

- I 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 Four Load Cases>Solver Configurations> Solution 7 (sol7)>Optimization Solver I>Stationary I>Segregated I node, then click Optimization.
- 4 In the Settings window for Segregated Step, locate the General section.
- 5 From the Linear solver list, choose Suggested Iterative Solver (GMRES with AMG) (opt).
- 6 In the Model Builder window, click Solid Mechanics.
- 7 In the Settings window for Segregated Step, locate the General section.
- 8 From the Linear solver list, choose Suggested Iterative Solver (GMRES with SA AMG) (solid).
- 9 In the Model Builder window, click Segregated I.
- 10 In the Settings window for Segregated, locate the General section.
- **II** From the **Termination technique** list, choose **Iterations** to reduce the computational time.
- 12 In the Model Builder window, click Advanced.
- 13 In the Settings window for Advanced, click to expand the Assembly Settings section.
- **I4** Clear the **Reuse sparsity pattern** check box.

**I5** 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

## Four Load Cases

- I In the Model Builder window, under Results click Topology Optimization.
- 2 In the Settings window for Group, type Four Load Cases in the Label text field.

#### Stress (solid) 1

- I In the Model Builder window, expand the Four Load Cases node, then click Stress (solid) I.
- 2 Drag and drop above Four Load Cases Output material volume factor 1.

#### Threshold I

- I In the Model Builder window, click Threshold I.
- 2 In the Threshold I toolbar, click **I** Plot.

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

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

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

### MESH I

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

# MESH 2

# Size

- I 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

- I In the Home toolbar, click 2 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  $\stackrel{\text{res}}{\longrightarrow}$  Add Study to close the Add Study window.

#### SMOOTH DESIGN (MESH2)

- I 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

- I In the Model Builder window, under Smooth Design (mesh2) click Step I: 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 Four Load Cases, Stationary.
- 6 In the Study toolbar, click **=** Compute.

## RESULTS

#### Filter 2

- I In the Model Builder window, expand the Results>Datasets node.
- 2 Right-click Results>Datasets>Filter 2 and choose Create Mesh in New Component.

#### MESH 3

#### Import I

- I In the Settings window for Import, locate the Import section.
- 2 From the Boundary partitioning list, choose Minimal.

# 3 Click Import.

# Free Triangular 1

- I In the Mesh toolbar, click  $\triangle$  Boundary and choose Free Triangular.
- 2 In the Settings window for Free Triangular, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.

## Size

- I In the Model Builder window, expand the Free Triangular I node, then click Size.
- 2 In the Settings window for Size, click to expand the Element Size Parameters section.
- 3 In the Maximum element size text field, type meshsz.
- 4 In the Minimum element size text field, type meshsz/2.
- 5 Click 🖷 Build Selected.

# Free Tetrahedral I

In the Mesh toolbar, click \land Free Tetrahedral.

# Size

- I In the Model Builder window, expand the Free Tetrahedral I 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 I (matlnk I)

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.

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

#### SOLID MECHANICS (SOLID2)

I 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 Settings window for Solid Mechanics, click to expand the Discretization section.
- 4 From the Displacement field list, choose Quadratic serendipity.

#### Fixed Constraint I

- I In the Model Builder window, expand the Solid Mechanics (solid2) node, then click Fixed Constraint I.
- 2 In the Settings window for Fixed Constraint, locate the Boundary Selection section.
- **3** From the Selection list, choose Constraint I.

#### Fixed Constraint 2

- I 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

- I 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

- I 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

- I In the Model Builder window, click Boundary Load I.
- 2 In the Settings window for Boundary Load, locate the Boundary Selection section.
- 3 From the Selection list, choose Load I.
- **4** Locate the **Force** section. Specify the  $\mathbf{F}_{\mathbf{A}}$  vector as

0	х
0	у
<pre>comp1.load(-P0,Y-YC,Z)</pre>	z

Boundary Load 2

- I 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	у
<pre>comp1.load(-P0,Y-YC,Z)</pre>	z

#### ADD STUDY

- I In the Home toolbar, click  $\stackrel{\text{res}}{\longrightarrow}$  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  $\sim 2$  Add Study to close the Add Study window.

#### VERIFICATION

- I 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

- I In the Model Builder window, under One Load Case click Step I: 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

I 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).

# FOUR LOAD CASES

# Step 1: Stationary

- I In the Model Builder window, under Four Load Cases click Step I: 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 Four Load Cases>Step 1: Stationary and choose Copy.

# VERIFICATION

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

# Step 1: Stationary

- I In the Settings window for Stationary, locate the Physics and Variables Selection section.
- **2** In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid2)	$\checkmark$	Automatic (Stationary)
Topology Optimization (Component I)		Automatic

**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)

- I 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 I

- I In the Model Builder window, right-click Surface I and choose Deformation.
- **2** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.
- 3 In the Displacement (solid2) toolbar, click **O** Plot.

# Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

# GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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.

# ADD COMPONENT

In the Home toolbar, click 《 Add Component and choose 3D.

# GEOMETRY I

Create the geometry. To simplify this step, import a prepared geometry.

#### Import I (imp1)

- I In the **Home** toolbar, click **Import**.
- 2 In the Settings window for Import, locate the Import section.
- 3 From the Source list, choose COMSOL Multiphysics file.
- 4 Click 📂 Browse.
- **5** Browse to the model's Application Libraries folder and double-click the file bracket.mphbin.
- 6 Click ा Import.

### Cylinder I (cyl1)

I In the Geometry toolbar, click 🔲 Cylinder.

- 2 In the Settings window for Cylinder, locate the Size and Shape section.
- 3 In the Radius text field, type D2/2+Da/2.
- 4 Locate the **Position** section. In the **x** text field, type -W/2.
- **5** In the **y** text field, type YC.
- 6 Locate the Axis section. From the Axis type list, choose x-axis.
- 7 Locate the Selections of Resulting Entities section. Select the Resulting objects selection check box.

Cylinder 2 (cyl2)

- I In the **Geometry** toolbar, click **Cylinder**.
- 2 In the Settings window for Cylinder, locate the Size and Shape section.
- 3 In the Radius text field, type D1/2+Da/2.
- 4 Locate the Position section. In the x text field, type W/2-X1.
- **5** In the **y** text field, type Y1.
- **6** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Move I (movI)

- I In the Geometry toolbar, click 💭 Transforms and choose Move.
- 2 In the Settings window for Move, locate the Input section.
- 3 From the Input objects list, choose Cylinder 2.
- 4 Locate the **Displacement** section. In the y text field, type -DY1.
- 5 Locate the Input section. Select the Keep input objects check box.

Block I (blk1)

- I In the **Geometry** toolbar, click 🗍 Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type rfillet+thickness.
- 4 In the **Depth** text field, type L1+rfillet.
- 5 In the Height text field, type rfillet+thickness.
- 6 Locate the Position section. In the x text field, type W/2-rfillet-thickness.
- 7 In the y text field, type -L1-rfillet.
- 8 In the z text field, type H/2-rfillet-thickness.

Mirror I (mir I)

- I In the Geometry toolbar, click 💭 Transforms and choose Mirror.
- 2 Select the object **blk1** only.
- 3 In the Settings window for Mirror, locate the Input section.
- 4 Select the Keep input objects check box.

#### Mirror 2 (mir2)

- I In the Geometry toolbar, click 💭 Transforms and choose Mirror.
- 2 Select the objects blk1, cyl2, mir1, and mov1 only.
- 3 In the Settings window for Mirror, locate the Normal Vector to Plane of Reflection section.
- 4 In the x text field, type 1.
- **5** In the **z** text field, type **0**.
- 6 Locate the Input section. Select the Keep input objects check box.

Add a big block to introduce a symmetry boundary in the geometry. The geometry can still be viewed by hiding the boundaries of the big block.

#### Block 2 (blk2)

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, locate the Position section.
- **3** In the **y** text field, type -1.
- 4 In the z text field, type -H/2.

# Partition Objects 1 (parl)

- I In the Geometry toolbar, click 🔲 Booleans and Partitions and choose Partition Objects.
- **2** Select the object **imp1** only.
- 3 In the Settings window for Partition Objects, locate the Partition Objects section.
- **4** Find the **Tool objects** subsection. Click to select the **Delta Activate Selection** toggle button.
- 5 Select the objects blk1, blk2, cyl1, cyl2, mir1, mir2(1), mir2(2), mir2(3), mir2(4), and mov1 only.
- **6** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

## Constraint I

- I In the Geometry toolbar, click 😼 Selections and choose Cylinder Selection.
- 2 In the Settings window for Cylinder Selection, type Constraint 1 in the Label text field.

- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Size and Shape section. In the Outer radius text field, type D1/2\*1.01.
- 5 Locate the Position section. In the x text field, type W/2-X1.
- **6** In the **y** text field, type Y1.
- 7 Locate the Output Entities section. From the Include entity if list, choose Entity inside cylinder.

# Constraint 2

- I Right-click Constraint I and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Constraint 2 in the Label text field.
- **3** Locate the **Position** section. In the **y** text field, type -Y1-DY1.

# Constraint 3

- I Right-click Constraint 2 and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Constraint 3 in the Label text field.
- **3** Locate the **Position** section. In the **x** text field, type -W/2+X1.

#### Constraint 4

- I Right-click Constraint 3 and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Constraint 4 in the Label text field.
- **3** Locate the **Position** section. In the **y** text field, type -Y1.

#### Load Ia

- I In the Geometry toolbar, click 🔓 Selections and choose Cylinder Selection.
- 2 In the Settings window for Cylinder Selection, type Load 1a in the Label text field.
- **3** Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the Size and Shape section. In the Outer radius text field, type D2/2\*1.01.
- **5** In the **Bottom distance** text field, type **0**.
- 6 In the Start angle text field, type 90 .
- 7 In the End angle text field, type 270.
- 8 Locate the **Position** section. In the **y** text field, type YC.
- 9 Locate the Axis section. From the Axis type list, choose x-axis.
- **IO** Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

## Load Ib

- I Right-click Load Ia and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Load 1b in the Label text field.
- 3 Locate the Size and Shape section. In the Start angle text field, type -90.
- 4 In the End angle text field, type 90.

## Load 2a

- I Right-click Load Ib and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Load 2a in the Label text field.
- **3** Locate the Size and Shape section. In the Top distance text field, type 0.
- 4 In the **Bottom distance** text field, type Inf.

# Load 2b

- I Right-click Load 2a and choose Duplicate.
- 2 In the Settings window for Cylinder Selection, type Load 2b in the Label text field.
- 3 Locate the Size and Shape section. In the Start angle text field, type 90.
- 4 In the End angle text field, type 270.

#### Load I

- I In the Geometry toolbar, click 💁 Selections and choose Union Selection.
- 2 In the Settings window for Union Selection, type Load 1 in the Label text field.
- **3** Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the Input Entities section. Click + Add.
- 5 In the Add dialog box, in the Selections to add list, choose Load Ia and Load 2a.
- 6 Click OK.

## Load 2

- I In the Geometry toolbar, click 🔓 Selections and choose Union Selection.
- 2 In the Settings window for Union Selection, type Load 2 in the Label text field.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Input Entities section. Click + Add.
- 5 In the Add dialog box, in the Selections to add list, choose Load 1b and Load 2b.

6 Click OK.

# Domains Outside Design Space

I In the Geometry toolbar, click 🐚 Selections and choose Adjacent Selection.

- **2** In the **Settings** window for **Adjacent Selection**, type Domains Outside Design Space in the **Label** text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent domains**.
- 5 Locate the Input Entities section. Click + Add.
- 6 In the Add dialog box, in the Input selections list, choose Constraint 1, Constraint 2, Constraint 3, and Constraint 4.
- 7 Click OK.

Design Space Domains

- I In the Geometry toolbar, click 🝖 Selections and choose Difference Selection.
- 2 In the Settings window for Difference Selection, type Design Space Domains in the Label text field.
- **3** Locate the **Input Entities** section. Click + Add.
- 4 In the Add dialog box, select Partition Objects I in the Selections to add list.
- 5 Click OK.
- 6 In the Settings window for Difference Selection, locate the Input Entities section.
- 7 Click + Add.
- 8 In the Add dialog box, in the Selections to subtract list, choose Cylinder I and Cylinder 2.
- 9 Click OK.

Material Boundaries

- I In the Geometry toolbar, click 🝖 Selections and choose Cylinder Selection.
- **2** In the **Settings** window for **Cylinder Selection**, type Material Boundaries in the **Label** text field.
- **3** Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Size and Shape section. In the Outer radius text field, type Da+D2/2\*1.01.
- 5 Locate the **Position** section. In the **y** text field, type YC.
- 6 Locate the Axis section. From the Axis type list, choose x-axis.
- 7 Locate the Output Entities section. From the Include entity if list, choose Entity inside cylinder.
- 8 In the Geometry toolbar, click 🟢 Build All.

9 Click the 🕂 Zoom Extents button in the Graphics toolbar.

The model geometry is now complete.

32 | BRACKET - TOPOLOGY OPTIMIZATION