Created in COMSOL Multiphysics 5.6

Topology Optimization of an MBB Beam

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 geometry and load pattern is similar to the Messerschmitt-Bölkow-Blohm beam (MBB) used as a validation test for topology optimization.

Note: This application is also used in *Introduction to the Optimization Module*.

Model Definition

The model studies optimal material distribution in the beam, which consists of a linear elastic material, structural steel. The dimensions of the beam region — 6 meters by 1 meter by 0.5 meters — means an original total weight of 23,550 kg. The beam is symmetric about the plane $x = 3$. Both its ends rest on rollers while an edge load acts on the top middle part (Figure 1).

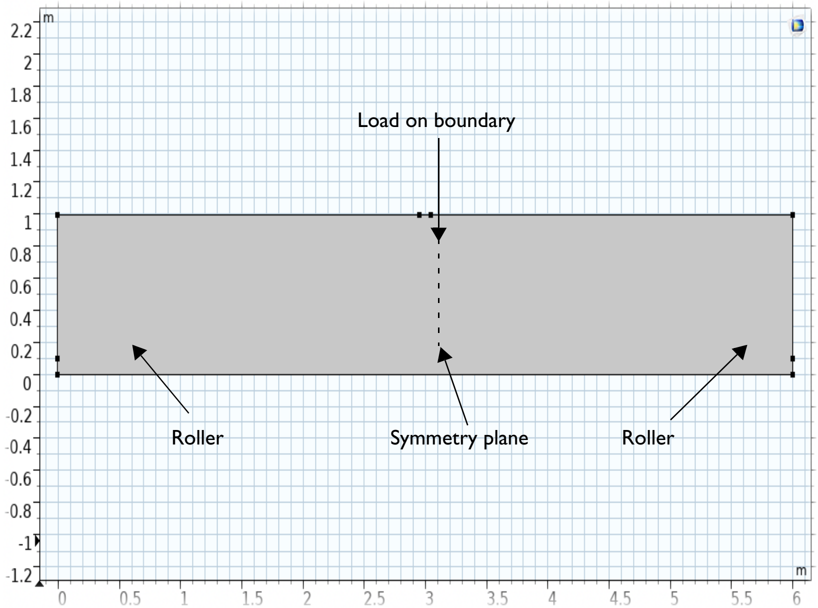

Figure 1: Geometry of the beam with loads and constraints.

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.

The design can be described with a artificial material volume factor, θ_c which acts as control variable in the optimization problem. In structural mechanics, convention dictates that it is 0 in void domains and 1 in solid domains. In order to make the problem formulation well-posed one has to limit the complexity of the design; otherwise the optimization will just tell us that the optimal structure is a composite with infinitesimal features, which cannot be resolved with any mesh. The detail of the design can be limited using a filter that produces another field, θ_f which is guaranteed to be free of features smaller than some set value, $R_{\min}$ because it is computed as a solution to a Helmholtz equation (see Ref. 1):

$$\theta_f = R_{\min}^2 \nabla^2 \theta_f + \theta_c$$

The equation essentially just smears out the design as illustrated below with and θ_c and θ_f to the left and right, respectively.

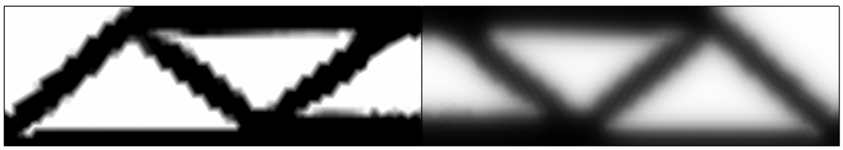

Figure 2: The material volume factor (right) has a lot of gray scale.

This filtered design variable can then be projected to reduce the extent of the gray scale region. COMSOL supports projection based on the hyperbolic tangent function, see Ref. 2:

$$\theta = \frac{(\tanh(\beta(\theta_f - \theta_\beta)) + \tanh(\beta\theta_\beta))}{(\tanh(\beta(1 - \theta_\beta)) + \tanh(\beta\theta_\beta))}$$

This is illustrated for $\theta_\beta = 0.5$ and $\beta = 8$ below with θ_f and θ to the left and right.

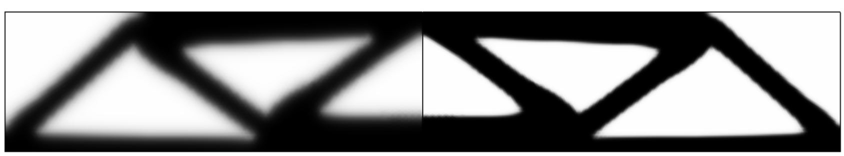

Figure 3: The (projected) material volume factor shown to the right has no visible gray scale.

High values of β are associated with strong projection and less gray scale, but also slows down the optimization process, so initially we will not use it and thus set $\theta = \theta_f$.

To prevent a completely solid design, we impose an overall limit V_{frac} on the fraction θ_{avg} of the domain area occupied by solid material

$$0 \le \theta_{\text{avg}} = \int_{\Omega} \theta(\mathbf{x}) d\Omega \le V_{\text{frac}}$$

To guarantee a binary solution we use the SIMP penalization method (see Ref. 3) to define the penalized field

$$\theta_p = \theta_{\min} + (1 - \theta_{\min})\theta(\mathbf{x})^p$$
$$E(\mathbf{x}) = E_0 \theta_p$$

Since the stiffness must for numerical reasons not vanish completely anywhere in the model, the relative minimum Young's modulus, $\theta_{\min}$, has been introduced. The exponent $p \ge 1$ is a penalty factor which makes intermediate densities provide less stiffness compared to what they cost in weight. Increasing p therefore forces θ_c toward either of its bounds leading to a sharper solution.

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 sub-optimal design is identified.

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.

Results

The following plot shows the stiffness distribution for the optimized solution. The resulting design is an approximation of a truss structure, which is expected for this beam size. Note that the optimal design for a longer beam is quite different.

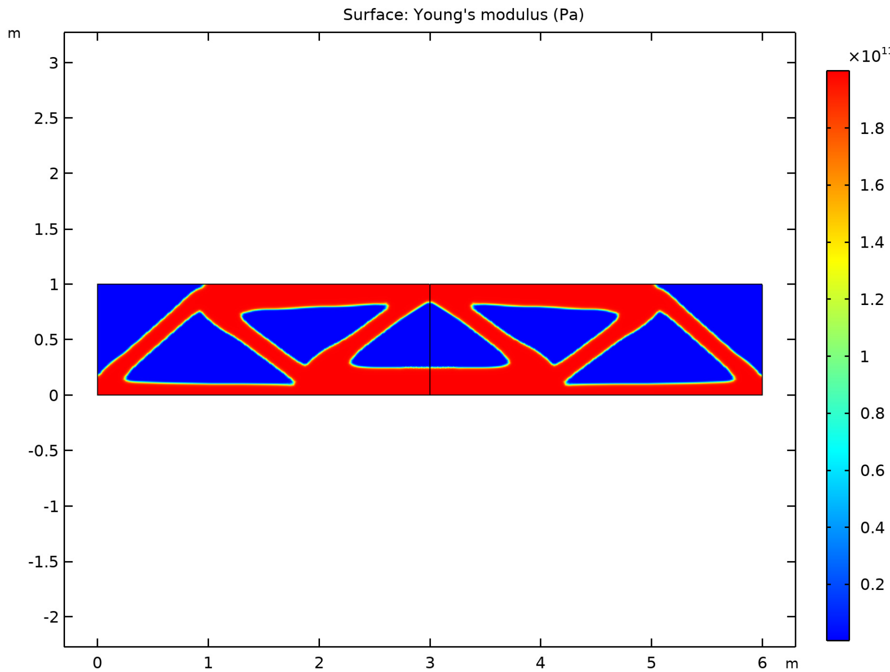

Figure 4: A plot of the penalized stiffness for the optimized design.

Notes About the COMSOL Implementation

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 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 the left half of the beam geometry must be modeled, because of the assumed symmetry, which is implemented as a Roller 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 beam sizes, but keep in mind that the mesh size is taken as the default 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.

This example demonstrates a strategy based on using MMA with a limited number of iterations on successively finer meshes. It quickly and reliably produces trustworthy topologies showing good improvement of the objective function value. These solutions are not necessarily globally optimal, which may, however, be of less importance in practice.

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/mbb_beam_optimization`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Optimization>Topology Optimization, Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Global Definitions>Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
a	3 [m]	3 m	Beam half width
b	1 [m]	1 m	Beam height
L1	0.1 [m]	0.1 m	Support width
meshsz	6 [cm]	0.06 m	Mesh size
volfrac	0.5	0.5	Maximum volume fraction
beta	1	1	Projection slope

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type a.

4 In the **Height** text field, type b.

Point 1 (pt1)

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

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

3 In the **y** text field, type `L1`.

Point 2 (pt2)

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

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

3 In the **x** text field, type `a-L1/2`.

4 In the **y** text field, type `b`.

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

Load Boundary

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `a-L1/1.999`.

5 In the **y minimum** text field, type `b*0.999`.

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

Symmetry y

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x maximum** text field, type `a*0.001`.

5 In the **y maximum** text field, type `L1*1.001`.

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

Symmetry x

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `a*0.999`.

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

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

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)**.

SOLID MECHANICS (SOLID)

Roller 1

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

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

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

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

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

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

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in y direction** check box.

This is effectively a roller condition along the x-axis, but it is applied on a vertical boundary to avoid bending stiffness.

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 Boundary**.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

Since this is a linear problem, the magnitude of the applied force does not affect the optimal topology.

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

`0`	x
`-100 [kN]`	y

MESH 1

Size

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

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 Click **Build All**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
`volfrac`	`0.5`	0.5	`Maximum volume fraction`
`beta`	`1`	1	`Projection slope`

Here we have also defined the maximum volume fraction.

Add a density topology feature, which can be used to distinguish between void and solid regions. This variable is coupled back to the **Solid Mechanics** interface via the Young's modulus. The filter radius should not be smaller than the mesh element size, so the default will work, but a fixed value has to be chosen to make the result of the optimization mesh independent.

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 **Projection** section.

3 From the **Projection type** list, choose **Hyperbolic tangent projection**.

4 In the β text field, type `beta`.

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

OPTIMIZATION

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

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

3 Locate the **Study Settings** section. 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
beta (Projection slope)	1 8	
meshsz (Mesh size)	6 3	cm

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

b equal to one corresponds to no projection, so this effectively initializes an optimization with projection on a fine mesh with the solution to the optimization problem on a coarse without projection.

Initialize the solution object with control variable and its filtered version, so that a suitable plot can be set up.

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

RESULTS

In the **Model Builder** window, expand the **Results** node.

Set up a **Mirror** dataset which can be used for plotting the whole geometry rather than just the left half.

Mirror 2D 1

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

2 Right-click **Results>Datasets** and choose **More 2D Datasets>Mirror 2D**.

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

4 In row **Point 1**, set **X** to `a`.

5 In row **Point 2**, set **X** to `a`.

6 Locate the **Data** section. From the **Dataset** list, choose **Optimization/Parametric Solutions 1 (sol2)**.

7 Click to expand the **Advanced** section. Select the **Define variables** check box.

Mirror 2D 2

1 Right-click **Mirror 2D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Optimization/Solution 1 (sol1)**.

Topology

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 In the **Label** text field, type `Topology`.

Surface 1

1 Right-click **Topology** and choose **Surface**.

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

3 In the **Expression** text field, type `if(mir1side,dtopo1.theta_c,dtopo1.theta_f)`.

4 Select the **Description** check box.

5 In the associated text field, type `theta_c (left) and theta_f (right)`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

7 Select the **Reverse color table** check box.

Topology 2

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

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

3 From the **Dataset** list, choose **Mirror 2D 2**.

4 In the **Label** text field, type `Topology 2`.

Surface 1

1 In the **Model Builder** window, expand the **Topology 2** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `if(mir1side,dtopo1.theta_f,dtopo1.theta)`.

4 In the **Description** text field, type `theta_f (left) and theta (right)`.

Add the **Topology Optimization** study step, select a solver, limit the number of iterations and switch on plotting while solving.

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 `50`.

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 **Topology 2**.

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.

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

RESULTS

Topology 2

Plotting the projected field gives a sharper and fairer picture of the topology, as seen by the **Solid Mechanics** interface.