



Topology Optimization of a Beam with Milling Constraints

Introduction

This model is based on the [Design Optimization of a Beam](#) model. The model uses the **Density Model** feature to solve a structural topology optimization problem with milling constraints.

Model Definition

The model geometry ([Figure 1](#)) consists of two regions: A fixed domain on which a distributed load is applied and a design domain.

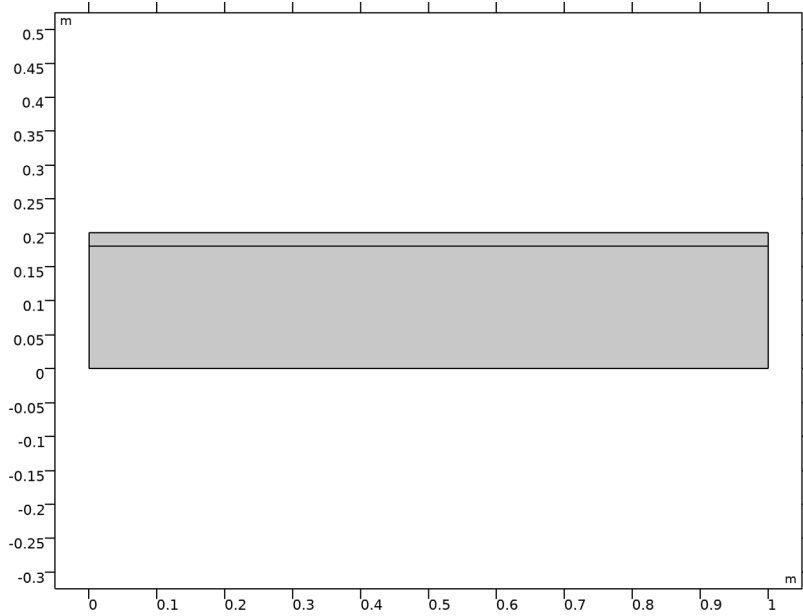


Figure 1: The model geometry with the Prescribed Material domain at the top.

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

For a detailed introduction to the use of structural topology optimization and how to use a Helmholtz filter for regularization, see the model [Topology Optimization of an MBB Beam](#). The main points are that Young's modulus varies spatially to reflect the material distribution. It is not possible to set zero void stiffness, as this causes the void displacement

field to become undefined. This example considers milling constraints for the x and y directions, and this is implemented with two PDEs ($n = 2$):

$$\begin{aligned}
0 &\leq \theta_c \leq 1 \\
\theta_f &= \theta_c + R_{\min}^2 \nabla^2 \theta_f \\
0 &= \hat{\mathbf{m}}_{\text{mil}}^i \cdot \nabla \theta_m^i \quad , \quad |\hat{\mathbf{m}}_{\text{mil}}^i| = 1 \\
\theta_m &= \left[\sum_{i=1}^n ((\theta_m^i)^{-p_{\text{mil}}})/n \right]^{-1/p_{\text{mil}}} \\
\theta &= \frac{(\tanh(\beta(\theta_f - \theta_\beta)) + \tanh(\beta\theta_\beta))}{(\tanh(\beta(1 - \theta_\beta)) + \tanh(\beta\theta_\beta))} \\
\theta_p &= \theta_{\min} + (1 - \theta_{\min})\theta^p \\
E &= E_0\theta_p
\end{aligned}$$

The approach is inspired by [Ref. 1](#), which uses a finite volume discretization, but in this case a stabilized finite element method is used to solve the convective equations.

Results and Discussion

Figure 2 displays the result of optimization together with the distributed load and the mesh. The displacement field is shown in colors and the maximum value is located near the end of the beam.

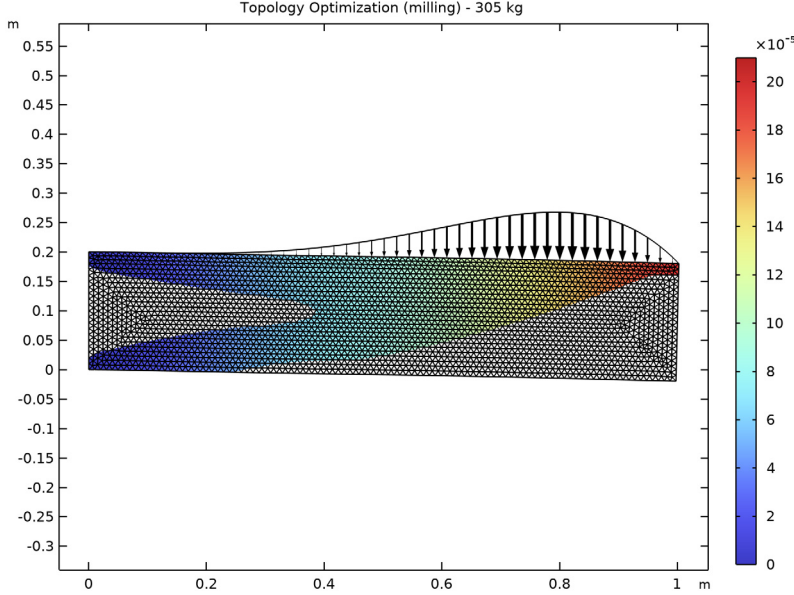


Figure 2: The optimization removes material from both milling directions to reduce the mass as much as possible without violating the displacement constraint in the upper right corner.

Notes About the COMSOL Implementation

This model combines the **Topology Optimization** and **Solid Mechanics** interfaces. In this case, the default values of the **Density Model** work well, but for more complicated 3D problems it might be beneficial to apply a continuation strategy in p , β , and p_{mil} . The model opts for smooth results in postprocessing by using a linear discretization for the milling variables, θ_m^i , but you can also use a constant discretization.

The model is nonlinear but only in the sense that it consists of a series of linear coupled problems. Therefore a **Segregated** solver can compute the solution in a single iteration.

Finally, the **Optimization** study step is recycled from the [Design Optimization of a Beam](#) model, but one could equally well have used a **Topology Optimization** study step in which case the move limit could be defined in that step. Using MMA instead of (the default)

GCMMA would still require changing a setting on the **Optimization Solver** node. However, this is not strictly necessary, so the model can be solved without editing the solver sequence if a longer computational time is acceptable.

Reference

1. L. Høghøj and E.A. Träff, “An advection–diffusion based filter for machinable designs in topology optimization,” *Comp. Meth. App. Mech. & Eng.*, vol. 391, p. 114488, 2022.

Application Library path: Optimization_Module/Topology_Optimization/
beam_optimization_milling

Modeling Instructions

APPLICATION LIBRARIES

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **Optimization Module>Design Optimization>beam_optimization** in the tree.
- 3 Click  **Open**.

PARAMETER OPTIMIZATION, SHAPE OPTIMIZATION

- 1 In the **Model Builder** window, Ctrl-click to select **Parameter Optimization** and **Shape Optimization**.
- 2 Right-click and choose **Delete**.

COMPONENT 1 (COMPI)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

SHAPE OPTIMIZATION

In the **Model Builder** window, expand the **Component 1 (comp1)>Shape Optimization** node.

Free Shape Domain 1, Polynomial Boundary 1, Symmetry/Roller 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Shape Optimization**, Ctrl-click to select **Free Shape Domain 1**, **Polynomial Boundary 1**, and **Symmetry/Roller 1**.
- 2 Right-click and choose **Delete**.

3 Right-click **Shape Optimization** and choose **Delete**.

GLOBAL DEFINITIONS

Parameters 1

- 1 In the **Model Builder** window, expand the **Topology Optimization** 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
volfrac	0.4	0.4	Volume fraction
meshsz	1 [cm]	0.01 m	Mesh size

TOPOLOGY OPTIMIZATION

Density Model 1 (dtopol)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Topology Optimization** click **Density Model 1 (dtopol)**.
- 2 In the **Settings** window for **Density Model**, click to expand the **Milling** section.
- 3 From the **Milling constraints** list, choose **Enabled**.
- 4 Click  **Add**.
- 5 In the table, enter the following settings:

X	Y
0	1
1	0

- 6 Locate the **Filtering** section. From the $R_{\min}$ list, choose **User defined**.
- 7 In the text field, type meshsz.
- 8 Locate the **Projection** section. From the **Projection type** list, choose **Hyperbolic tangent projection**.
- 9 Locate the **Control Variable Initial Value** section. In the θ_0 text field, type 0.1.

MESH 1

Free Triangular 2

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

Size

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

TOPOLOGY OPTIMIZATION

Optimization

- 1 In the **Model Builder** window, expand the **Topology Optimization** node, then click **Optimization**.
- 2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
- 3 Find the **Solver settings** subsection. In the **Maximum number of model evaluations** text field, type 1000.
- 4 Click **Replace Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>Density Model 1>Global>comp1.dtopo1.theta_avg - Average material volume factor**.
- 5 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
comp1.dtopo1.theta_avg/ volfrac	Average material volume factor	Stationary

Solver Configurations

In the **Model Builder** window, expand the **Topology Optimization>Solver Configurations** node.

Solution 5 (sol5)

- 1 In the **Model Builder** window, expand the **Topology Optimization>Solver Configurations>Solution 5 (sol5)** node.
- 2 Right-click **Dependent Variables 1** and choose **Update Variables**.
- 3 In the **Settings** window for **Optimization Solver**, locate the **Optimization Solver** section.
- 4 Select the **Maximum number of outer iterations** check box. In the associated text field, type 50.
- 5 Clear the **Globally Convergent MMA** check box.
- 6 Select the **Move limits** check box.
- 7 In the **Model Builder** window, expand the **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver 1>Stationary 1** node.

- 8 Right-click **Stationary I** and choose **Segregated**.
- 9 In the **Settings** window for **Segregated**, locate the **General** section.
- 10 From the **Termination technique** list, choose **Iterations**.
- 11 Right-click **Segregated I** and choose **Segregated Steptwice**.
- 12 In the **Settings** window for **Segregated Step**, type Solid Mechanics in the **Label** text field.
- 13 Locate the **General** section. Under **Variables**, click  **Add**.
- 14 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopol.theta_c)** and **Displacement field (comp1.u)**.
- 15 Click **OK**.
- 16 In the **Model Builder** window, under **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver I>Stationary I>Segregated I** click **Segregated Step I**.
- 17 In the **Settings** window for **Segregated Step**, type Milling in the **Label** text field.
- 18 Locate the **General** section. Under **Variables**, click  **Add**.
- 19 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopol.theta_c)**, **Milling material volume factor (comp1.dtopol.theta_m1)**, and **Milling material volume factor (comp1.dtopol.theta_m2)**.
- 20 Click **OK**.
- 21 In the **Model Builder** window, under **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver I>Stationary I>Segregated I** click **Segregated Step**.
- 22 In the **Settings** window for **Segregated Step**, type Optimization in the **Label** text field.
- 23 Locate the **General** section. In the **Variables** list, choose **Milling material volume factor (comp1.dtopol.theta_m1)**, **Displacement field (comp1.u)**, and **Milling material volume factor (comp1.dtopol.theta_m2)**.
- 24 Under **Variables**, click  **Delete**.

RESULTS

Topology Optimization

- 1 In the **Model Builder** window, expand the **Results** node, then click **Topology Optimization**.
- 2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.
- 3 In the **Title** text area, type Topology Optimization (milling) -
eval(mass1.mass) kg.

4 From the **Number format** list, choose **Stopwatch**.

5 In the **Number of decimals** text field, type 0.

TOPOLOGY OPTIMIZATION

Optimization

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

RESULTS

Topology Optimization

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

