Created in COMSOL Multiphysics 6.2

Topology Optimization of a Beam with Milling Constraints

This model is licensed under the COMSOL Software License Agreement 6.2.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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.

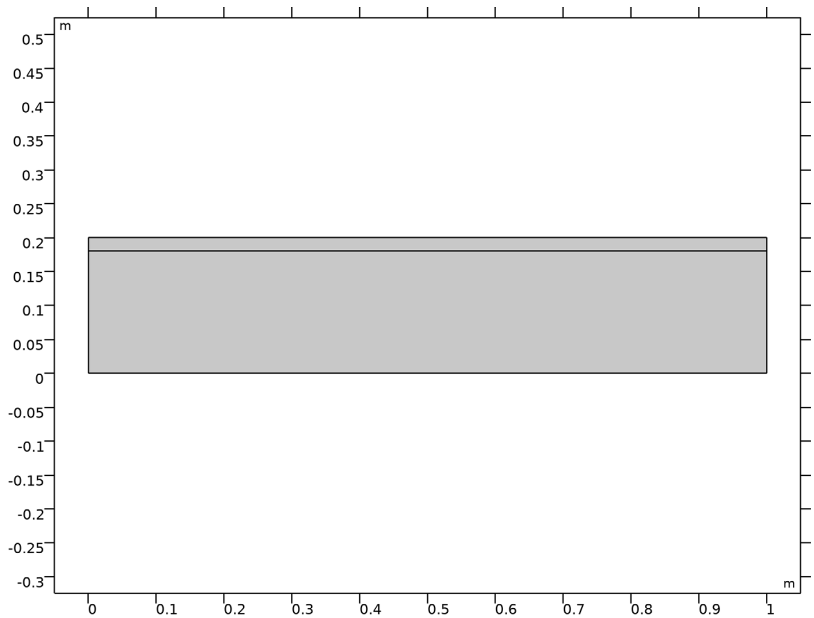

Figure 1: The model geometry with the Prescribed Material domain at the top.

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 ($n = 2$):

$$0 \le \theta_c \le 1$$

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

$$0 = \hat{\mathbf{m}}_{\mathrm{mil}}^i \cdot \nabla \theta_m^i \quad , \quad \left|\hat{\mathbf{m}}_{\mathrm{mil}}^i\right| = 1$$

$$\theta_m = \left[\sum_{i=1}^{n} ((\theta_m^i)^{-p_{\mathrm{mil}}})/n\right]^{-1/p_{\mathrm{mil}}}$$

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

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

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.

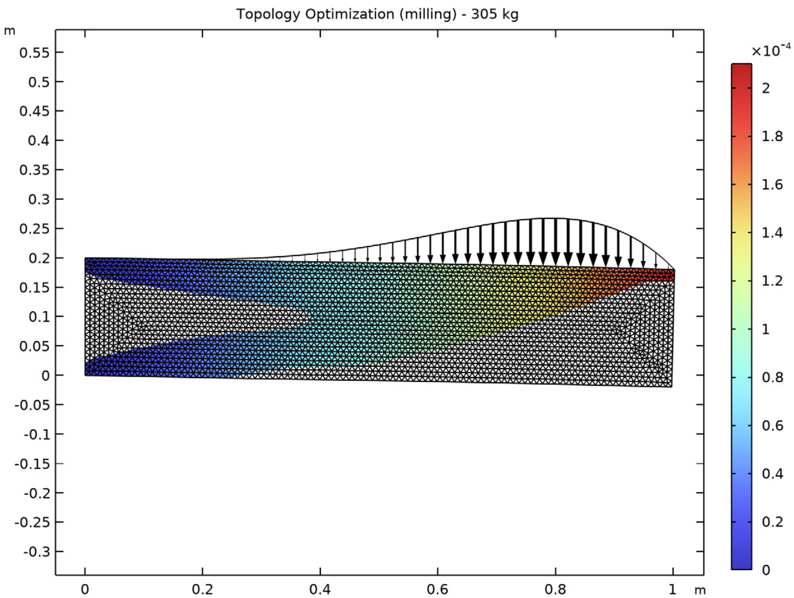

Figure 2: The optimization removes material from both milling directions to reduce the mass as much as possible without violating the displacement constraint in the upper right corner.

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 (COMP1)

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 (dtopo1)

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

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

8 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

Topology Optimization

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

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 Clear the **Globally Convergent MMA** check box.

5 In the **Model Builder** window, expand the **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver 1>Stationary 1** node.

6 Right-click **Stationary 1** and choose **Segregated**.

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

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

9 Right-click **Segregated 1** and choose **Segregated Step** twice.

10 In the **Settings** window for **Segregated Step**, type `Solid Mechanics` in the **Label** text field.

11 Locate the **General** section. Under **Variables**, click + **Add**.

12 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopo1.theta_c)** and **Displacement field (comp1.u)**.

13 Click **OK**.

14 In the **Model Builder** window, under **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver 1>Stationary 1>Segregated 1** click **Segregated Step 1**.

15 In the **Settings** window for **Segregated Step**, type `Milling` in the **Label** text field.

16 Locate the **General** section. Under **Variables**, click **Add**.

17 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopo1.theta_c)**, **Milling material volume factor (comp1.dtopo1.theta_m1)**, and **Milling material volume factor (comp1.dtopo1.theta_m2)**.

18 Click **OK**.

19 In the **Model Builder** window, under **Topology Optimization>Solver Configurations>Solution 5 (sol5)>Optimization Solver 1>Stationary 1>Segregated 1** click **Segregated Step**.

20 In the **Settings** window for **Segregated Step**, type `Optimization` in the **Label** text field.

21 Locate the **General** section. In the **Variables** list, choose **Milling material volume factor (comp1.dtopo1.theta_m1)**, **Displacement field (comp1.u)**, and **Milling material volume factor (comp1.dtopo1.theta_m2)**.

22 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

Topology Optimization

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

RESULTS

Topology Optimization

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