Optimization Module

Application Library Manual

5.2a

COMSOL

Optimization Module Application Library Manual

Version: COMSOL 5.2a

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM021703

Created in COMSOL Multiphysics 5.2a

Mooney-Rivlin Curve Fit

Introduction

This tutorial example demonstrates how to use the Optimization Module to estimate unknown function parameters based on measured data. The two-parameter Mooney-Rivlin solid material model is used as an example, but the procedure is generally applicable when you need to fit a parametrized analytic function to measured data.

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

Model Definition

The two-parameter incompressible Mooney-Rivlin material model describes the local behavior of rubber-like materials. The model assumes that the local strain energy density in an incompressible solid is a simple function of local strain invariants.

In a standard tensile test, a rotationally symmetric test specimen is pulled in such a way that it extends in one direction and contracts symmetrically in the other two. For this case of uniaxial extension, the relationship between applied force, F, and resulting extension, ΔL, of a true Mooney-Rivlin material is

$$\frac{F}{A_0} = 2\left(C_{10} + C_{01}\frac{L_0}{L_0 + \Delta L}\right)\left(\frac{L_0 + \Delta L}{L_0} - \left(\frac{L_0}{L_0 + \Delta L}\right)^2\right) \tag{1}$$

where A_0 is the original cross-section area of the test specimen and L_0 is its reference length. The constants C_{10} and C_{01} are material parameters which must be determined by fitting Equation 1 to the experimental data from the tensile test.

In practice, tensile test data is delivered in a form which is independent of the geometry of the test specimen used. There are multiple possible formats. The one used here contains corresponding measured values of engineering stress, P_i, representing force per unit reference area

$$P = \frac{F}{A_0}$$

and stretch, λ_i, representing relative elongation

$$\lambda = \frac{L_0 + \Delta L}{L_0}$$

The expected relationship between these variables for a Mooney-Rivlin material is

$$P(\lambda) = 2\left(C_{10} + \frac{C_{01}}{\lambda}\right)\left(\lambda - \frac{1}{\lambda^2}\right)$$

Given N pairs of measurements (λ_i, P_i), $i = 1..N$, the values of C_{10} and C_{01} which best fit the measured data are considered to be those which minimize the total squared error

$$e = \sum_{i=1}^{N} (P(\lambda_i) - P_i)^2$$

The curve fitting problem is therefore identical to an optimization problem.

Results

The following plot shows measured stress and stress computed using the material properties that have been fitted to the measured data.

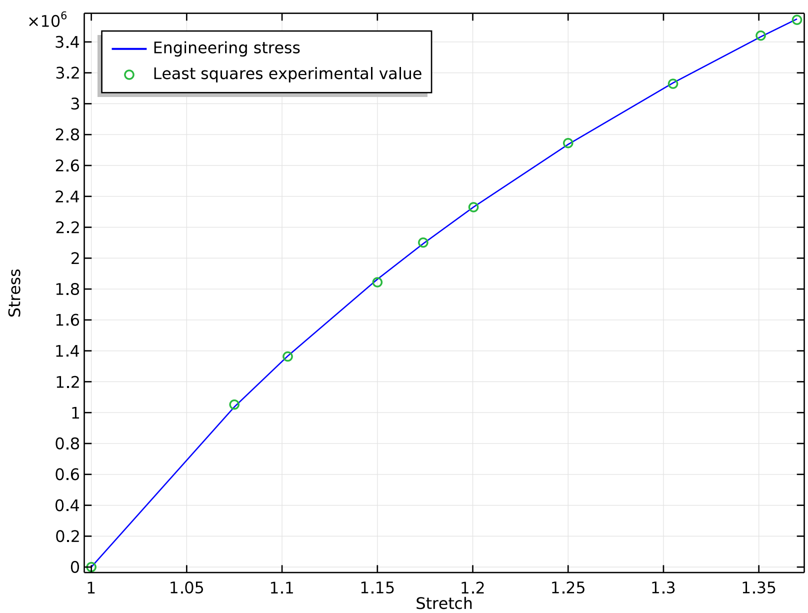


Figure 1: Measured (green circles) and computed (blue line) stresses.

Application Library path: `Optimization_Module/Parameter_Estimation/curve_fit_mooney_rivlin`

Modeling Instructions

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

NEW

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

Since no geometry is involved in this pure curve fitting problem, choose to create a **0D** component. The **Optimization** interface is required when using least-squares objective functions while the **Stationary** study step is necessary as a container for the control variables to be optimized.

MODEL WIZARD

1 In the **Model Wizard** window, click **0D**.

2 In the **Select Physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Add the stretch parameter `lambda` which has been varied in the measured data. Also add the unknown material parameters as global model parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
lambda	1	1	Stretch
C10	1[MPa]	1E6 Pa	Mooney-Rivlin parameter
C01	1[MPa]	1E6 Pa	Mooney-Rivlin parameter

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Global Variables**.

Set up the assumed relationship between stretch and engineering stress as a variable expression in terms of lambda and the yet unknown C10 and C01.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
P	2*(C10+C01/lambda)*(lambda-1/lambda^2)	Pa	Engineering stress

OPTIMIZATION (OPT)

Global Least-Squares Objective 1

1 On the **Optimization** toolbar, click **Global Least-Squares Objective**.

You can import tensile test data from file in the form of comma-separated values or, as demonstrated here, enter the data manually into a local table.

2 In the **Settings** window for Global Least-Squares Objective, locate the **Experimental Data** section.

3 From the **Data source** list, choose **Local table**.

4 Click **Add Column**.

5 In the table, enter the following settings:

1	2
1	0
1.075	1.052e6
1.103	1.3633e6
1.15	1.844e6
1.174	2.101e6
1.2004	2.330e6

1	2
1.25	2.745e6
1.305	3.129e6
1.351	3.441e6
1.37	3.543e6

6 From the **Parameter type** list, choose **Parameter**.

7 In the **Parameter name** text field, type `lambda`.

8 From the **Parameter column** list, choose **1**.

9 In the table, enter the following settings:

Data column	Unit	Model expression	Weight
2	Pa	P	1

STUDY 1

Use an Optimization study step to set up control variables and select an optimization solver. The Levenberg-Marquardt solver is particularly efficient for least-squares problems such as this one.

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **Levenberg-Marquardt**.

4 Locate the **Control Variables and Parameters** section. Click **Add**.

5 In the table, enter the following settings:

Parameter name	Initial value	Scale
C10	1[MPa]	1[MPa]

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Initial value	Scale
C01	1[MPa]	1[MPa]

8 On the **Study** toolbar, click **Compute**.

RESULTS

Add a plot of the least-squares fitted stress-strain curve together with the measured data.

1D Plot Group 1

On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Global 1

On the **1D Plot Group 1** toolbar, click **Global**.

1D Plot Group 1

1 In the **Settings** window for Global, locate the **y-axis data** section.
2 Click **P - Engineering stress** in the upper-right corner of the section. In the **Model Builder** window, click **1D Plot Group 1**.

Global 2

On the **1D Plot Group 1** toolbar, click **Global**.

1D Plot Group 1

1 In the **Settings** window for Global, locate the **y-axis data** section.
2 Click **comp1.opt.glsobj1.col1 - Least squares experimental value** in the upper-right corner of the section. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
3 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
4 From the **Positioning** list, choose **In data points**.
5 On the **1D Plot Group 1** toolbar, click **Plot**.

   Finish the plot by adjusting the title, axis labels, and legend positioning.
6 In the **Model Builder** window, click **1D Plot Group 1**.
7 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
8 From the **Title type** list, choose **None**.
9 Locate the **Plot Settings** section. Select the **x-axis label** check box.
10 In the associated text field, type `Stretch`.
11 Select the **y-axis label** check box.
12 In the associated text field, type `Stress`.
13 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Derived Values

Use the predefined **Objective value** node to evaluate the estimated values of material parameters C01 and C10.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.
2 In the **Settings** window for Global Evaluation, locate the **Data** section.
3 From the **Parameter selection (lambda)** list, choose **Last**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Solver>Control parameters>C10 - Mooney-Rivlin parameter**.
5 Click **Evaluate**.
6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Solver>Control parameters>C01 - Mooney-Rivlin parameter**.
7 Click **Evaluate**.

1D Plot Group 1

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

Created in COMSOL Multiphysics 5.2a

Optimizing a Flywheel Profile

Introduction

The radial stress component in an axially symmetric and homogeneous flywheel of constant thickness exhibits a sharp peak near the inner radius. From there, it decreases monotonously until it reaches zero at the flywheel's outer rim; see Figure 1. The uneven stress distribution—apparent also for the azimuthal component—reveals a design that does not make optimal use of the material available.

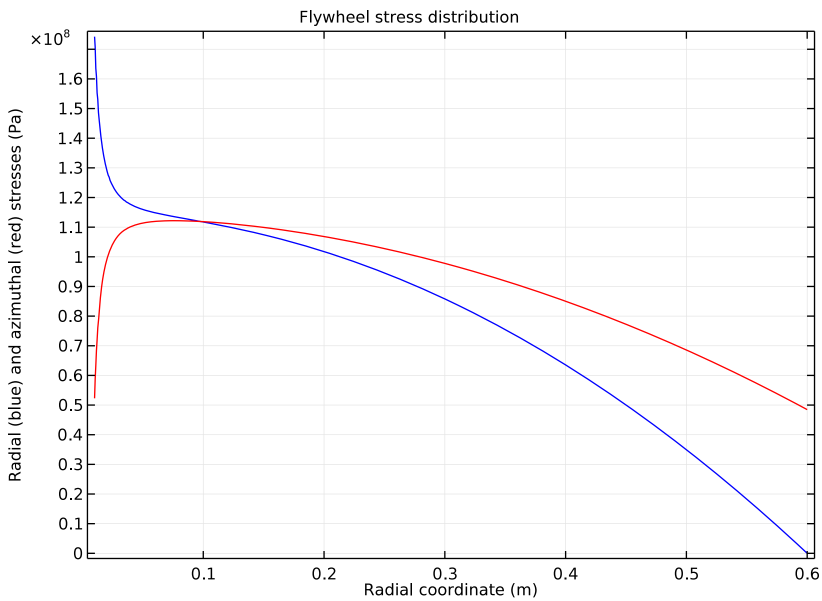


Figure 1: Radial (blue) and azimuthal (red) stress components in a homogeneous flywheel of constant thickness.

This model solves the problem of finding the thickness profile that results in a radial stress distribution that is as even as possible for given values of the flywheel's mass and moment of inertia. The model was inspired by Ref. 1.

Model Definition

Before describing the optimization problem, this section derives the dynamical equations, which you implement using a General Form PDE interface in 1D.

STRESSES IN A ROTATING FLYWHEEL

In a rotating flywheel, stresses due to the flywheel's weight are typically very small compared to dynamically induced stresses; therefore this model neglects gravitational stress. Expressed in terms of $U \equiv u/r$, where u is the local radial displacement (m) and r is the radial coordinate, the stress components along the radial and azimuthal directions in a rotationally symmetric disk made of a homogeneous, isotropic, and elastic material with Young's modulus E (N/m^2) and Poisson's ratio ν read:

$$\begin{aligned} \sigma_r &= \frac{E}{1-\nu^2}\left[r\frac{dU}{dr} + (1+\nu)U\right] \\ \sigma_\varphi &= \frac{E}{1-\nu^2}\left[\nu r\frac{dU}{dr} + (1+\nu)U\right] \end{aligned} \tag{1}$$

Inserting these expressions in the equation of motion for an infinitesimal mass element, results in the second-order ordinary differential equation (ODE)

$$-r^2\frac{d^2U}{dr^2} - (3+\Phi)r\frac{dU}{dr} + (1-(1+\nu)\Phi)U = \frac{1-\nu^2}{E}\rho\omega^2 r^2 \qquad r_0 < r < r_1 \tag{2}$$

valid for a centrally bored flywheel with inner radius r_0 (m) and outer radius r_1 (m) rotating with the angular velocity ω (rad/s). In this equation, the flywheel's thickness, H, which can be a function of r, enters through the dimensionless function

$$\Phi \equiv \frac{r}{H}\frac{dH}{dr} = r\frac{d}{dr}\log\left(\frac{H}{H_0}\right)$$

At the inner radius the displacement is zero and at the outer radius the radial stress component vanishes, which corresponds to the following boundary conditions:

$$U\big|_{r=r_0} = 0, \qquad r\frac{dU}{dr} + (1+\nu)U\big|_{r=r_1} = 0 \tag{3}$$

Given the function Φ, Equation 2 combined with Equation 3 forms a well-posed ODE problem. With the solution $U = U(r)$ at hand, you can determine the stress components through the Equation 1.

THE OPTIMIZATION PROBLEM

For the special case of constant flywheel thickness, $H(r) = H_0$, the function Φ is identically zero. As Figure 1 shows and you verify later, this shape results in an uneven stress distribution, with a maximum for σ_r at $r = r_0$.

This model concerns optimizing the flywheel's profile to obtain a radial stress distribution that is as even as possible under the design requirements of specified flywheel mass and moment of inertia. To formulate the task in mathematical terms, tentatively introduce the objective function

$$Q_{\text{stress}}[H] = \int_{r_0}^{r_1} \frac{(\sigma_r - \sigma_{r,\,\text{mean}})^2}{\sigma_0^2} dr \tag{4}$$

where $\sigma_{r,\text{mean}}$ denotes the average radial stress value along the flywheel's radial extension, and σ_0 is a normalization constant. The latter is introduced to make the integrand dimensionless and its value is chosen to be roughly an order of magnitude smaller than σ_r to give Q_{stress} a suitable magnitude. The optimization problem is then to find the shape $H = H(r)$ that minimizes Q_{stress} under the additional constraints

$$m \equiv 2\pi\rho \int_{r_0}^{r_1} H r dr = m_0$$

and

$$I \equiv \pi\rho \int_{r_0}^{r_1} H r^3 dr = I_0$$

where m_0 and I_0 are the desired mass and moment of inertia, respectively.

However, Equation 4 alone does not give a reasonable result; suppressing profiles where dH/dr is not smooth requires a second term in the objective function:

$$Q_{\text{smoothness}}[H] = A \int_{r_0}^{r_1} \left(\frac{dH}{dr}\right)^2 dr$$

Here A is a normalization constant to be chosen such that $Q_{\text{smoothness}}$ and Q_{stress} are comparable in magnitude; as long as this condition is satisfied, the model is fairly insensitive to the value of A.

MODEL DATA

Table 1 gives the input data for the model. As the initial design, take a flywheel of constant thickness H_0. The material properties correspond to those of steel.

TABLE 1: MODEL DATA

PROPERTY	VALUE	DESCRIPTION
r_0	0.01 m	Inner flywheel radius
r_1	0.60 m	Outer flywheel radius
H_0	0.03 m	Initial flywheel thickness
E	$2.1 \cdot 10^{11}$ N/m^2	Young's modulus
ν	0.3	Poisson's parameter
ρ	7800 kg/m^3	Density
ω	$2\pi \cdot 50$ rad/s	Angular velocity
σ_0	10^7 Pa	Normalization constant, stress term
A	1	Normalization constant, smoothness term

Results and Discussion

Figure 2 shows the optimized flywheel profile (black lines) with that of the original flat flywheel of the same mass and moment of inertia included for comparison (green lines).

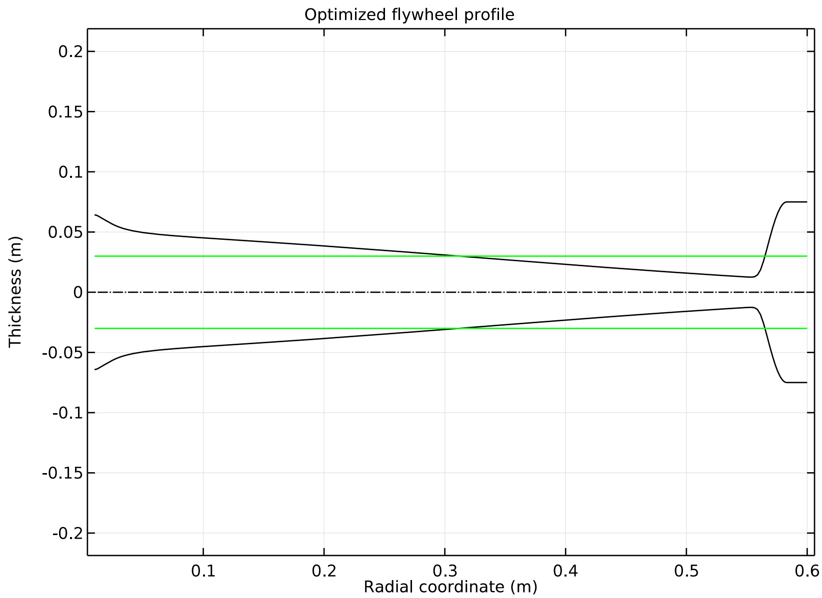


Figure 2: Optimized thickness profile.

Figure 3 displays the radial and azimuthal stress components for both the initial and the optimized flywheel profiles. In the optimized flywheel, the stress components are almost equal and nearly constant for most of the radial cross-section. The maximal stress, which occurs in the radial direction at the inner radius, is roughly 102 MPa for the optimized profile compared to 174 MPa for the flat flywheel—a reduction by more than 40%.

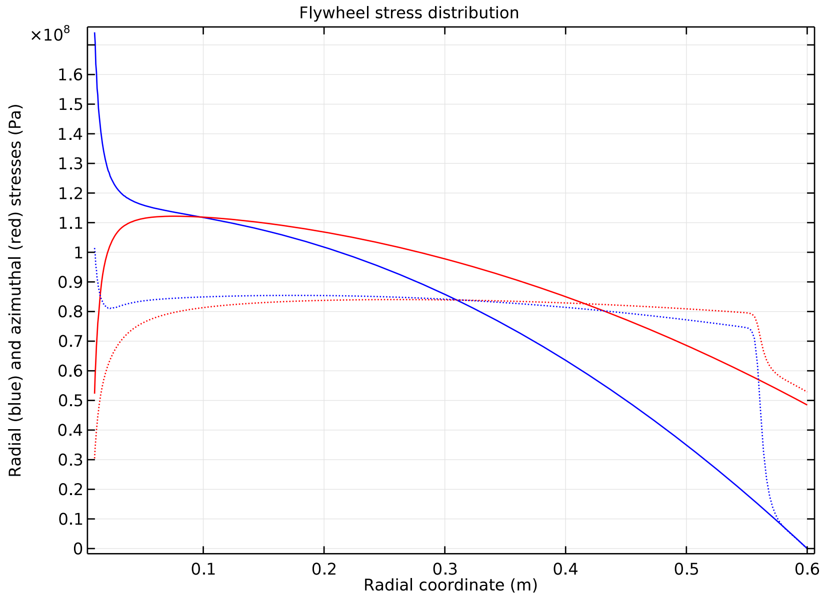


Figure 3: Radial (blue) and azimuthal (red) stress components for initial (solid) and optimized (dotted) flywheel profiles.

Reference

1. G.R. Kress, "Shape Optimization of a Flywheel," *Struct. Multidisc. Optim.*, vol. 19, pp. 74–81, 2000.

Application Library path: `Optimization_Module/Shape_Optimization/flywheel_profile`

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 **1D**.
2 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.
3 Click **Add**.
4 In the **Dependent variables** table, enter the following settings:

U

5 In the **Select Physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.
6 Click **Add**.
7 Click **Study**.
8 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
9 Click **Done**.

ROOT

1 In the **Model Builder** window, click the root node.
2 In the root nodes'**Settings** window, locate the **Unit System** section.
3 From the **Unit system** list, choose **None**.

   This setting turns off all unit support in the model.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Description
r0	0.01	Inner flywheel radius
r1	0.60	Outer flywheel radius
H0	0.03	Initial flywheel thickness
E	2.1e11	Young's modulus
nu	0.3	Poisson's ratio
C	E/(1-nu^2)	Constant in ODE and stress equations
rho	7.8e3	Density

Name	Expression	Description
omega	2*pi*50	Angular velocity
sigma0	1e7	Normalization constant, stress term
A	1	Normalization constant, smoothness term

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

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

3 In the **Left endpoint** text field, type `r0`.

4 In the **Right endpoint** text field, type `r1`.

5 Right-click **Interval 1 (i1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for Variables, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **All domains**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Description
sigma_r	C*(x*Ux+(1+nu)*U)	Stress, r-component
sigma_phi	C*(nu*x*Ux+(1+nu)*U)	Stress, phi-component
Phi	(x/H)*Hx	Expression in ODE

Name	Expression	Description
q_stress	(sigma_r-sigma_r_mean)^2/sigma0^2	Objective function, stress term
q_smoothness	A*Hx^2	Objective function, smoothness term

Define an Average operator that you can use to define the mean radial stress.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

Variables 2

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
sigma_r_mean	aveop1(sigma_r)	Mean of radial stress component
m0	2*pi*H0*intop1(rho*x)	Initial flywheel mass
m	2*pi*intop1(H*rho*x)	Flywheel mass
I0	pi*H0*intop1(rho*x^3)	Initial flywheel moment of inertia
I	pi*intop1(H*rho*x^3)	Flywheel moment of inertia

GENERAL FORM PDE (G)

General Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **General Form PDE 1**.

2 In the **Settings** window for General Form PDE, locate the **Conservative Flux** section.

3 In the Γ text field, type `-x^2*Ux-(1+nu)*x*U`.

4 Locate the **Source Term** section. In the *f* text field, type `rho*omega^2*x^2/C-(nu-Phi)*x*Ux-(1+nu)*(1-Phi)*U`.

Dirichlet Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 Select Boundary 1 only.

OPTIMIZATION (OPT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Optimization (opt)**.

Control Variable Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Control Variable Field**.

2 In the **Settings** window for Control Variable Field, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Control Variable** section. In the **Control variable name** text field, type `H`.

5 In the **Initial value** text field, type `H0`.

6 Locate the **Discretization** section. From the **Element order** list, choose **Linear**.

Control Variable Bounds 1

1 In the **Model Builder** window, right-click **Control Variable Field 1** and choose **Control Variable Bounds**.

2 In the **Settings** window for Control Variable Bounds, locate the **Bounds** section.

3 In the **Lower bound** text field, type `0.25*H0`.

4 In the **Upper bound** text field, type `2.5*H0`.

Integral Objective 1

1 On the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Objective** section. In the **Objective expression** text field, type `q_stress+ q_smoothness`.

Global Inequality Constraint 1

1 On the **Physics** toolbar, click **Global** and choose **Global Inequality Constraint**.

2 In the **Settings** window for Global Inequality Constraint, locate the **Constraint** section.

3 In the **Constraint expression** text field, type `m-m0`.

Global Inequality Constraint 2

1 On the **Physics** toolbar, click **Global** and choose **Global Inequality Constraint**.

2 In the **Settings** window for Global Inequality Constraint, locate the **Constraint** section.

3 In the **Constraint expression** text field, type `I-I0`.

MESH 1

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

Edge 1

1. In the **Settings** window for Edge, locate the **Domain Selection** section.
2. From the **Geometric entity level** list, choose **Entire geometry**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `2.5e-3`.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Size**.

Size 1

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. Select Boundary 1 only.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `1e-3`.

Edge 1

Right-click **Edge 1** and choose **Size**.

Size 2

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. Select Boundary 2 only.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `5e-4`.
7. Click **Build All**.

STUDY 1

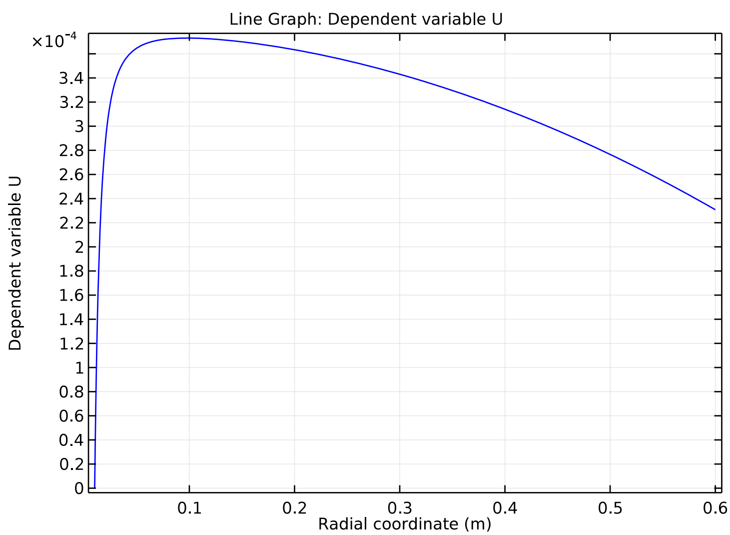
On the **Home** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

The first default plot shows the dependent variable U. Update the label for the horizontal axis.

1. In the **Model Builder** window, click **1D Plot Group 1**.
2. In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.
3. Select the **x-axis label** check box.
4. In the associated text field, type `Radial coordinate (m)`.
5. On the **1D Plot Group 1** toolbar, click **Plot**.



Local radial displacement, u, over radial coordinate, r, for constant flywheel thickness.

1D Plot Group 2

Although trivial for the current solution, the second plot group shows the flywheel profile, H. You can let COMSOL Multiphysics update this plot while it solves the optimization problem so that you can see the flywheel profile evolve.

1. In the **Model Builder** window, under **Results** click **1D Plot Group 2**.
2. In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Flywheel profile`.

Derived Values

Now compute the maximum value of the radial stress component, which appears at the inner radius.

Point Evaluation 1

On the **Results** toolbar, click **Point Evaluation**.

Derived Values

1 Select Boundary 1 only.

2 In the **Settings** window for Point Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1> Definitions>Variables>sigma_r - Stress, r-component**.

3 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result—roughly `1.74e8`, that is, 174 MPa—appears in the **Table** window below the **Graphics** window. Keep this table so that you can compare this value to that for the optimized flywheel profile.

RESULTS

Reproduce the plot in Figure 1 of the radial and azimuthal stress components with the following steps:

1D Plot Group 3

On the **Results** toolbar, click **1D Plot Group**.

Line Graph 1

On the **1D Plot Group 3** toolbar, click **Line Graph**.

1D Plot Group 3

1 In the **Settings** window for Line Graph, locate the **Selection** section.

2 From the **Selection** list, choose **All domains**.

3 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Definitions>Variables>sigma_r - Stress, r-component**.

4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>x - x-coordinate**.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

6 On the **1D Plot Group 3** toolbar, click **Plot**.

7 Right-click **Line Graph 1** and choose **Duplicate**.

8 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Definitions> Variables>sigma_phi - Stress, phi-component**.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

10 In the **Model Builder** window, click **1D Plot Group 3**.

11 In the **Settings** window for 1D Plot Group, locate the **Title** section.

12 From the **Title type** list, choose **Manual**.

13 In the **Title** text area, type `Flywheel stress distribution`.

14 Locate the **Plot Settings** section. Select the **x-axis label** check box.

15 In the associated text field, type `Radial coordinate (m)`.

16 Select the **y-axis label** check box.

17 In the associated text field, type `Radial (blue) and azimuthal (red) stresses (Pa)`.

18 On the **1D Plot Group 3** toolbar, click **Plot**.

The plot in the **Graphics** window should resemble that in Figure 1.

Now turn to the optimization problem.

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 **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

4 From the **Plot group** list, choose **1D Plot Group 2**.

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **SNOPT**.

Adjust the optimality tolerance.

4 In the **Optimality tolerance** text field, type `1e-4`.

To keep the solution for the initial profile, disable the corresponding solver sequence and then generate a new one for the optimization.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Solution 1 (sol1)** and choose **Disable**.

Solution 2 (sol2)

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

Also, change the solution data set for **1D Plot Group 2** to the one corresponding to the new sequence.

RESULTS

1D Plot Group 2

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 2 (sol2)**.

STUDY 1

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

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

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

4 On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 3

The third plot group shows the stress distribution components for the original flat profile. Add the results for the optimized profile to the plot.

1 In the **Model Builder** window, under **Results>1D Plot Group 3** right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

5 On the **1D Plot Group 3** toolbar, click **Plot**.
6 In the **Model Builder** window, under **Results>1D Plot Group 3** right-click **Line Graph 2** and choose **Duplicate**.
7 In the **Settings** window for Line Graph, locate the **Data** section.
8 From the **Data set** list, choose **Study 1/Solution 2 (sol2)**.
9 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
10 On the **1D Plot Group 3** toolbar, click **Plot**.

The plot in the **Graphics** window should resemble that in Figure 3.

Derived Values

Proceed to compute the radial stress value for the optimized flywheel profile.

1 In the **Model Builder** window, under **Results>Derived Values** click **Point Evaluation 1**.
2 In the **Settings** window for Point Evaluation, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Solution 2 (sol2)**.
4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The value should be close to 102 MPa, which is roughly 42 per cent lower than that for the flat flywheel profile.

RESULTS

1D Plot Group 2

Finally, reproduce the plot in Figure 2 as follows.

1 In the **Model Builder** window, expand the **1D Plot Group 2** node, then click **Line Graph 1**.
2 In the **Settings** window for Line Graph, click to expand the **Coloring and style** section.
3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.
4 Right-click **Results>1D Plot Group 2>Line Graph 1** and choose **Duplicate**.
5 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
6 In the **Expression** text field, type `-H`.
7 Right-click **Results>1D Plot Group 2>Line Graph 2** and choose **Duplicate**.
8 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

9 In the **Expression** text field, type `H0`.

10 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

11 Right-click **Results>1D Plot Group 2>Line Graph 3** and choose **Duplicate**.

12 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

13 In the **Expression** text field, type `-H0`.

Add a line for the symmetry axis ($z = 0$).

14 Right-click **Results>1D Plot Group 2>Line Graph 4** and choose **Duplicate**.

15 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

16 In the **Expression** text field, type `0`.

17 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

18 From the **Color** list, choose **Black**.

19 In the **Model Builder** window, click **1D Plot Group 2**.

20 In the **Settings** window for 1D Plot Group, locate the **Title** section.

21 In the **Title** text area, type `Optimized flywheel profile`.

22 Locate the **Plot Settings** section. Select the **x-axis label** check box.

23 In the associated text field, type `Radial coordinate (m)`.

24 Select the **y-axis label** check box.

25 In the associated text field, type `Thickness (m)`.

26 Click to expand the **Axis** section. Select the **Preserve aspect ratio** check box.

27 On the **1D Plot Group 2** toolbar, click **Plot**.

Topology Optimization of a Loaded Knee Structure

Introduction

Imagine that you are designing a bracket that must carry a specified load offset down and to the side of its anchoring. As a first attempt, you try a fully solid construction like the one below

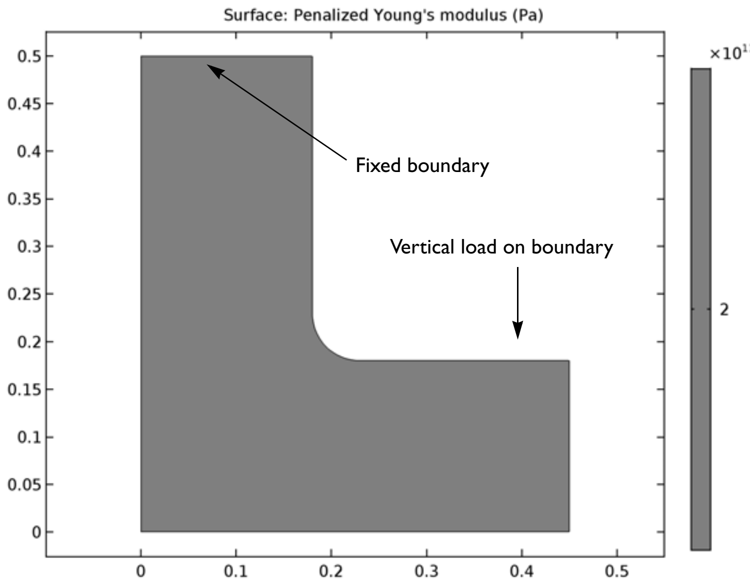


Figure 1: Geometry of the structure with loads and constraints.

This design is clearly over-designed and does not make efficient use of material. As a direct simulation reveals, stresses are unevenly distributed, meaning that some areas carry significantly less load than others. Provided that the stiffness of the part can be reduced to, for example, 40% of the value computed from the fully solid design without violating the specification, it is tempting to remove some material—saving both weight and costs. But what is the minimum amount of material necessary, and how should it be distributed?

This model demonstrates how you can apply the *SIMP model* (Solid Isotropic Material with Penalization) for structural topology optimization with COMSOL Multiphysics. Note that, in contrast to the way SIMP is usually applied, this example minimizes weight for a specified minimum stiffness, instead of maximizing stiffness for a specified maximum weight.

Model Definition

The area which may contain solid material is bounded by the initial L-shaped design., see Figure 1. The structure's uppermost boundary is fixed while the load is carried evenly distributed over the outermost 5 cm of the boundary indicated in the figure. The material from which the structure is made is a structural steel with Young's modulus, $E_0 = 200$ GPa, and Poisson's ratio, $\nu = 0.33$. The part is to be cut from a sheet of 2 cm thickness.

In the SIMP model (see Ref. 1), the stress tensor is considered to be a function of the true Young's modulus E_0 and the artificial density, ρ_{design}, which acts as control variable in the optimization problem:

$$
\begin{aligned}
E(\mathbf{x}) &= \rho_{\mathrm{design}}(\mathbf{x})^p E_0 \\
\mathbf{x} &\in \Omega,
\end{aligned}
$$

Since the stiffness must for numerical reasons not vanish completely anywhere in the model, the density parameter is constrained such that $10^{-9} \le \rho^p \le 1$. The exponent $p \ge 1$ is a penalty factor which makes intermediate densities provide less stiffness compared to what they cost in weight.

When there is a lower limit on the stiffness of any acceptable design, expressed equivalently as an upper bound on the total strain energy

$$
0 \le \int_\Omega W_s(\mathbf{x})d\Omega \le W_s^{\max} \tag{1}
$$

the SIMP penalization forces ρ_{design} towards either of its bounds. Increasing p therefore leads to a sharper solution. The maximum strain energy $W_s{}^{\max}$ is conveniently expressed as a factor times the lowest possible strain energy, as computed for the fully solid design in Figure 1.

Conceptually, the optimization problem is simply minimizing the amount of material used, measured as the solid fraction of the initial design domain

$$
\frac{1}{A}\int_\Omega \rho_{\mathrm{design}}(\mathbf{x})d\Omega \tag{2}
$$

without violating the stiffness constraint in Equation 1. Here, A is the area of the design domain.

However, the solution must not contain too much fine detail, since that would compromise the reliability of the final design. Also it is desirable to make the solution as independent of the mesh resolution as possible. To accomplish this, some kind of regularization is needed. One of the easiest regularizations to implement is based on a combination of successive mesh refinements and a penalty applied to the gradient of the design variable.

The integral of the squared norm of the gradient of the design variable ρ_{design} measures, in some sense, the total variation of the design variable. For a sharp topology solution represented by linear shape functions, the magnitude of the gradient is inversely proportional to the mesh element size. The zone of intermediate densities is at the same time one element thick, wherefore its area is proportional to the mesh element size. A reasonably mesh- and problem-independent penalty term is therefore of the form

$$\frac{h_0 h_{\mathrm{max}}}{A}\int_{\Omega}\left|\nabla\rho_{\mathrm{design}}(\mathbf{x})\right|^2 d\Omega$$

where h_0 is an initial mesh size, governing the size of details in the solution and h_{max} is the current mesh size at a given level in the process. This penalty term is dimensionless and, for the worst possible solution, on the order of 1.

The already dimensionless objective, Equation 2, and the penalty term must be balanced against each other, for example as a linear combination controlled by a parameter, q, to create a final composite objective function:

$$f = \frac{(1-q)}{A}\int_{\Omega}\rho_{\mathrm{design}}(\mathbf{x})d\Omega + q\frac{h_0 h_{\mathrm{max}}}{A}\int_{\Omega}\left|\nabla\rho_{\mathrm{design}}(\mathbf{x})\right|^2 d\Omega$$

THE FINITE ELEMENT MESH

Without regularization, problems of this type are intrinsically mesh-size dependent. For this model, generate an initial free triangular mesh using a maximum element size of 15 mm. This mesh, shown in Figure 2, is rather coarse and effectively limits the smallest possible size of details in the topology solution. When the mesh is later refined and the optimization solver restarted on the finer mesh, proper scaling of the regularization term ensures that the updated solution retains the same topology, only sharpening the details.

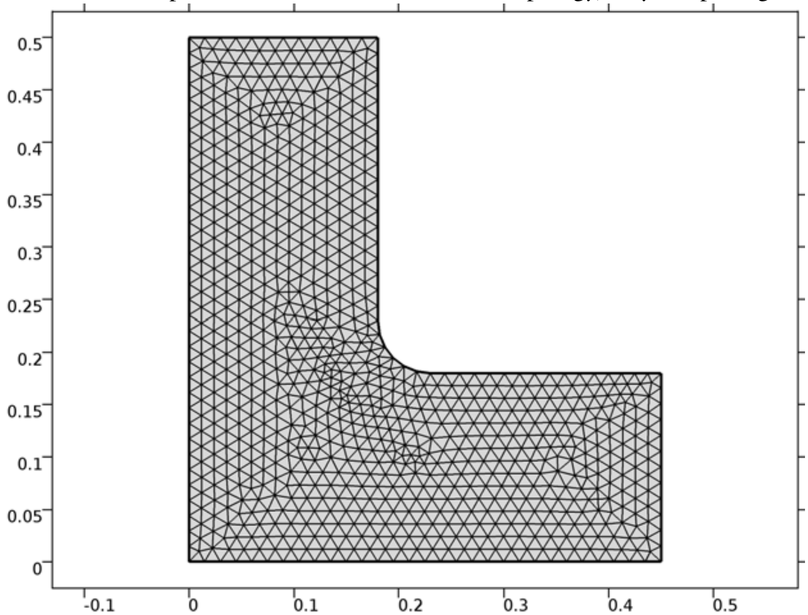


Figure 2: Finite element mesh of the structure.

Results and Discussion

Figure 3 shows one possible material distribution, optimal for the chosen regularization strategy. Black areas represent material and white areas represent void. Unless composite materials are considered, gray areas are unphysical so a useful optimized design should be essentially black and white. It is possible to obtain an even sharper result by reducing the regularization weight q, for example to $q = 0.1$. At the same time, however, artifacts caused by irregularities in the mesh may become more visible.

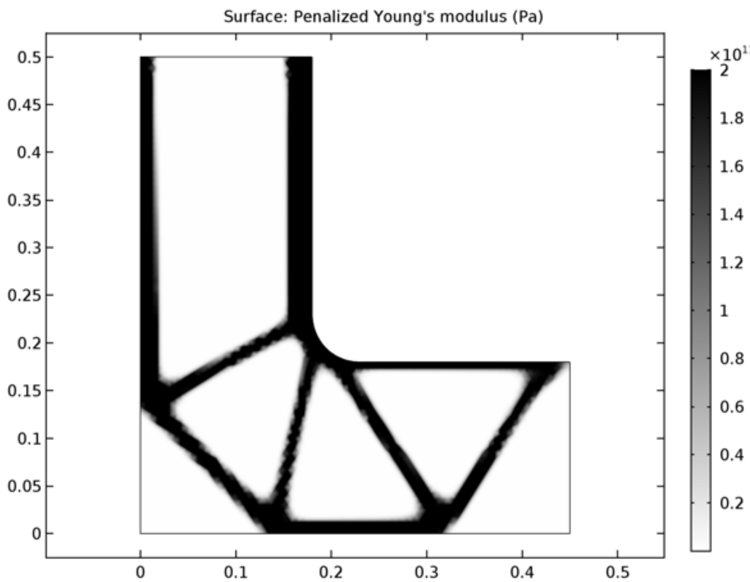


Figure 3: Distribution of the control variable after optimization.

As can be seen in Figure 4, below, the stress is relatively evenly distributed over the solid material. Each individual member in the truss-like structure is subjected to roughly the same maximum stress, indicating that the design makes full use of the material's stiffness. At the same time, the peak stress is not excessive: only about 3 times the mean stress in the solid areas.

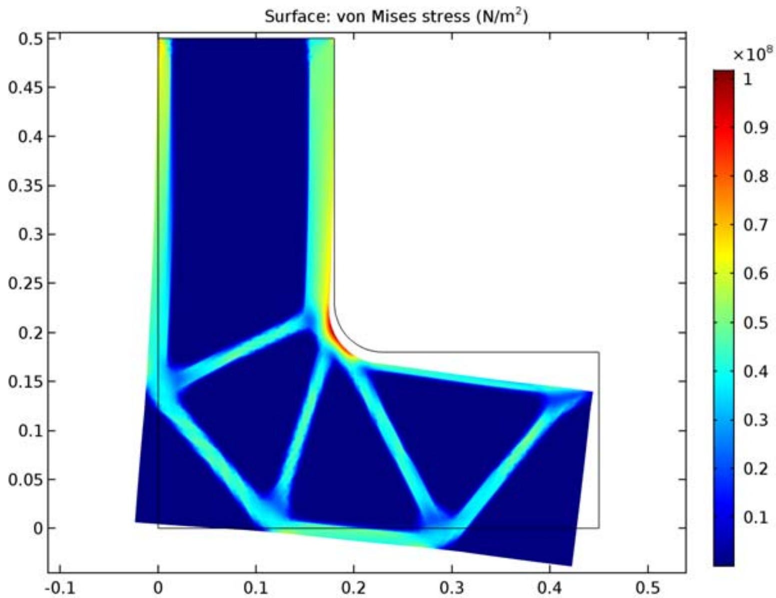


Figure 4: The von Mises stress of the optimal distribution of the control variable.

Reference

1. M.P. Bendsøe and O. Sigmund, *Topology Optimization Theory, Methods, and Applications*, Springer, 2004.

Notes About the COMSOL Implementation

A Solid Mechanics interface represents the structural properties of the bracket, while an Optimization interface allows adding the control variable, objective, and constraints for the optimization problem. The internal strain energy is a predefined variable, `solid.Ws`, readily available to use as the objective function for the optimization problem.

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. 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.

Application Library path: `Optimization_Module/Topology_Optimization/loaded_knee`

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 In the **Select Physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
F_load	10[kN]	1E4 N	Applied load
p	5	5	SIMP penalization factor
q	0.25	0.25	Regularization factor
h0	15[mm]	0.015 m	Initial mesh size
hmax	15[mm]	0.015 m	Current mesh size
WsRef	1	1	Reference strain energy
WsMaxFactor	1/0.4	2.5	Maximum allowed relative strain energy
d	2[cm]	0.02 m	Bracket thickness

Note that the reference strain energy is initially set equal to 1 and must be updated before optimization is started.

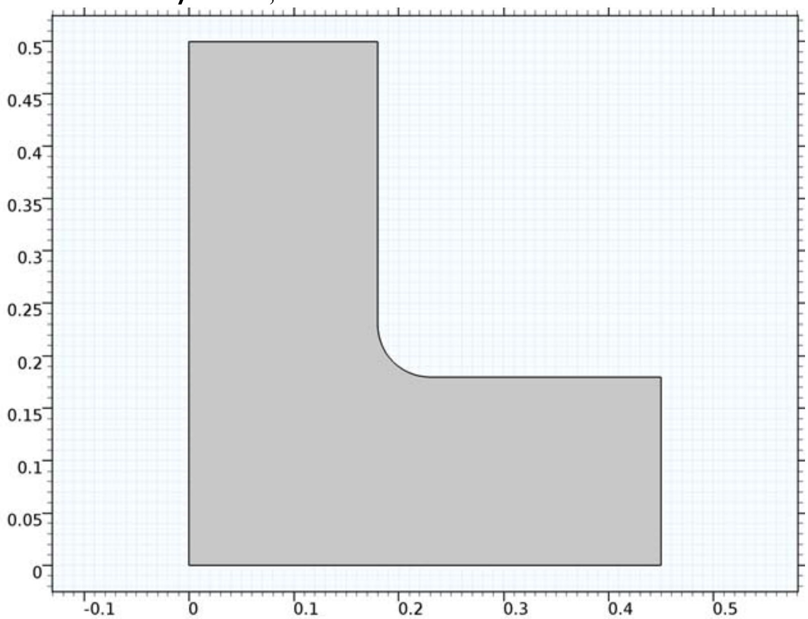
GEOMETRY 1

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

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

2 Browse to the application's Application Libraries folder and double-click the file `loaded_knee_geom_sequence.mph`.

3 On the **Geometry** toolbar, click **Build All**.



The geometry should now look like that in Figure 1. Note that the inserted geometry is parametrized and that the parameters used are automatically added to the list of global parameters in the model.

ADD MATERIAL

1 On 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 Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

Set up the penalized Young's modulus. Material property values can be accessed using the material's identifier and the property group identifier as scope. Since the control variable `rho_design` is not yet defined, the expression will temporarily be displayed in red, but this is nothing to worry about.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`E_SIMP`	`mat1.Enu.E* rho_design^p`		`Penalized Young's modulus`

Add a **Mass Properties** node to compute total mass and area.

4 In the **Model Builder** window, right-click **Definitions** and choose **Mass Properties**.

5 In the **Settings** window for Mass Properties, locate the **Density** section.

6 In the **Density expression** text field, type `mat1.def.rho*d*rho_design`.

Multiplying the material density `mat1.def.rho` by the bracket thickness `d` gives the area density of the solid structure. The factor `rho_design` accounts for the fraction of remaining material, which varies over the design domain.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for Solid Mechanics, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `d`.

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

2 In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `E_SIMP`.

In the next steps, you fix the structure for rigid body translation and rotation by prescribing zero displacements along one of the sides, then apply the load.

Fixed Constraint 1

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

2 Select Boundary 3 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 6 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

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

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

0	x
-F_load	y

OPTIMIZATION (OPT)

Define the control variable, which is `rho_design`, and associated shape functions. The initial value corresponds to a fully solid structure. Reduce the element order to 1 to get a linear interpolation without over- or undershoots.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Optimization (opt)**.

Control Variable Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Control Variable Field**.

2 Select Domain 1 only.

3 In the **Settings** window for Control Variable Field, locate the **Control Variable** section.

4 In the **Control variable name** text field, type `rho_design`.

5 In the **Initial value** text field, type `1`.

6 Locate the **Discretization** section. From the **Element order** list, choose **Linear**.

Next, constrain the control variable to the range $[10^{-9/p}, 1]$. According to Equation 2, the lower bound of this interval should be zero. However, this would cause the stiffness matrix of the Solid Mechanics interface to become singular. Instead, choose a lower bound which creates a contrast in stiffness of 9 orders of magnitude between solid and void areas.

Control Variable Bounds 1

1 In the **Model Builder** window, right-click **Control Variable Field 1** and choose **Control Variable Bounds**.

2 In the **Settings** window for Control Variable Bounds, locate the **Bounds** section.

3 In the **Lower bound** text field, type `1e-9^(1/p)`.

4 In the **Upper bound** text field, type `1`.

Integral Objective 1

1 On the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

Set up the mass minimization objective as an Integral Objective. Use the solid fraction as dimensionless objective function and multiply by a regularization weight term.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Objective, locate the **Objective** section.

4 In the **Objective expression** text field, type `(1-q)*rho_design/mass1.area`.

Integral Objective 2

1 On the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

Add a gradient norm regularization term in a second Integral Objective node.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Objective, locate the **Objective** section.

4 In the **Objective expression** text field, type `q*h0*hmax*(d(rho_design,x)^2+ d(rho_design,y)^2)/mass1.area`.

Integral Inequality Constraint 1

1 On the **Physics** toolbar, click **Domains** and choose **Integral Inequality Constraint**.

Set up an integral constraint that limits the strain energy, which is proportional to the compliance, to be at most `WsMaxFactor` times the reference energy `WsRef`.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Inequality Constraint, locate the **Constraint** section.

4 In the **Constraint expression** text field, type `solid.Ws/WsRef`.

5 Locate the **Bounds** section. In the **Upper bound** text field, type `WsMaxFactor`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Size

1 In the **Settings** window for Size, locate the **Element Size** section.

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `hmax`.

4 Click **Build All**.

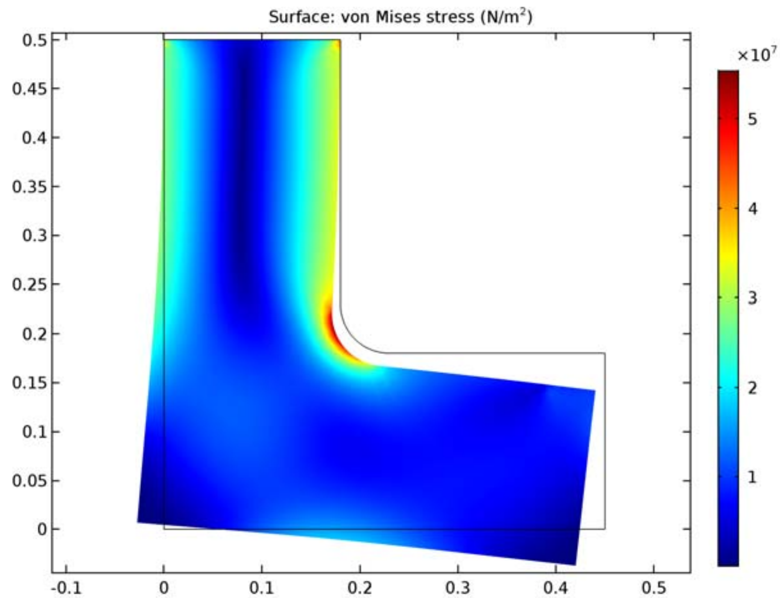
The resulting mesh is shown in Figure 2.

STUDY 1

First compute only a stationary solution on which you can evaluate a reference value for the objective, as well as set up suitable plots.

1 On the **Home** toolbar, click **Compute**.

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



RESULTS

2D Plot Group 2

1 In the **Model Builder** window, under **Results** click **2D Plot Group 2**.

2 In the **Settings** window for 2D Plot Group, type `Topology` in the **Label** text field.

Surface 1

Plotting the penalized Young's modulus gives a sharper and fairer picture of the topology, as seen by the Solid Mechanics interface.

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

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

3 In the **Expression** text field, type `E_SIMP`.

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

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

6 On the **Topology** toolbar, click **Plot**.

Global Evaluation 1

Evaluate a predefined Global Evaluation node to get a reference value for the strain energy constraint.

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

Use the value from the **Table 1** window to update the value of the parameter `WsRef`. This will normalize the compliance constraint in such a way that a maximum allowed relative compliance increase can be easily prescribed.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
WsRef	65.07	65.07	Reference strain energy

DEFINITIONS

Set up probes for the objective functions and constraint. Scale the objectives such that the values displayed become independent of the regularization factor.

Global Variable Probe 1 (var1)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for Global Variable Probe, locate the **Expression** section.

3 In the **Expression** text field, type `opt.iobj1/(1-q)`.

4 Select the **Description** check box.

5 In the associated text field, type `Solid fraction`.

Global Variable Probe 2 (var2)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for Global Variable Probe, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Optimization>opt.iobj2 - Objective value**.

3 Locate the **Expression** section. In the **Expression** text field, type `opt.iobj2/q`.

4 Select the **Description** check box.

5 In the associated text field, type `Gradient penalty`.

Global Variable Probe 3 (var3)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for Global Variable Probe, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Optimization>opt.iconstr1 - Constraint value**.

3 Locate the **Expression** section. Select the **Description** check box.

4 In the associated text field, type `Strain energy factor`.

STUDY 1

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

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **MMA**.

4 In the **Maximum number of objective evaluations** text field, type `50`.

This value is low enough to terminate the optimization process before the optimality tolerance limit has been reached, resulting in a warning from the solver. However, as you will be able to verify from the probes you just set up, it is sufficiently large for the objective function value to reach a stable level.

5 Locate the **Output While Solving** section. Select the **Plot** check box.

6 From the **Plot group** list, choose **Topology**.

7 On the **Study** toolbar, click **Compute**.

GLOBAL DEFINITIONS

Parameters

Reduce the maximum mesh element size to half its original value. This will increase the time required for each iteration, but rather few iterations are sufficient to refine the solution.

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
hmax	7.5[mm]	0.0075 m	Current mesh size

STUDY 1

Step 1: Stationary

Change the Stationary step to use the current solution as initial value of all variables. This automatically triggers a mapping from the old coarse mesh to the new, finer.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. 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 **Study 1**.

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

RESULTS

Topology

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The default plot shows the stress distribution in the optimized structure, clearly indicating that the load paths follow the optimized topology (Figure 4). Switch to the other plot group to display a sharper picture of the final design (Figure 3).

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).

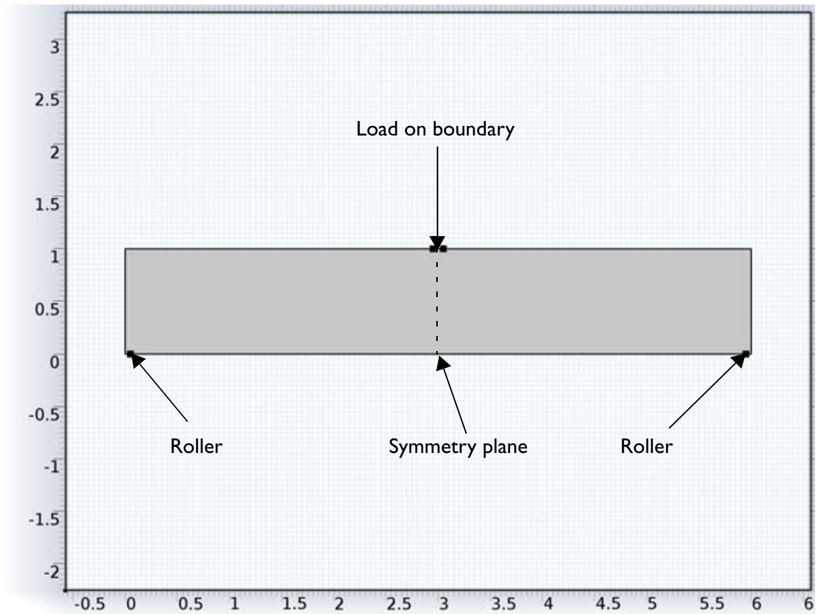


Figure 1: Geometry of the beam with loads and constraints.

The objective functional for the optimization, which defines the criterion for optimality, is the total *strain energy*. 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 load is 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.

Another condition which is more difficult to enforce is the requirement that the optimized topology is truly binary—any given point is either solid of void, nothing in between—but still without too fine detail or "checkerboard" patterns. The SIMP penalization method (see Ref. 1) accomplishes the first part. In the SIMP model, the stress tensor is considered to be a function of the true Young's modulus E_0 and the artificial density, ρ_{design}, which acts as control variable in the optimization problem:

$$E(\mathbf{x}) = \rho_{\mathrm{design}}(\mathbf{x})^p E_0$$
$$\mathbf{x} \in \Omega,$$

Since the stiffness must for numerical reasons not vanish completely anywhere in the model, the density parameter is constrained such that $10^{-9} \le \rho^p \le 1$. The exponent $p \ge 1$ is a penalty factor which makes intermediate densities provide less stiffness compared to what they cost in weight.

When there is an overall limit on the fraction $\gamma = 0.5$ of the domain area, A, occupied by solid material

$$0 \le \int_\Omega \rho_{\mathrm{design}}(\mathbf{x})d\Omega \le \gamma A$$

the SIMP penalization forces ρ_{design} toward either of its bounds. Increasing p therefore leads to a sharper solution.

At the same time, the solution must not contain too much fine detail, since that would compromise the reliability of the final design. Also it is desirable to make the solution as independent of the mesh resolution as possible. To accomplish this, some kind of regularization is needed. There are many possibilities. One of the easiest to implement is based on a combination of successive mesh refinements and a penalty applied to the gradient of the design variable.

The integral of the squared norm of the gradient of the design variable ρ_{design} measures, in some sense, the total variation of the design variable. For a sharp topology solution represented by linear shape functions, the magnitude of the gradient is inversely proportional to the mesh element size. The zone of intermediate densities is at the same time one element thick, wherefore its area is proportional to the mesh element size. A reasonably mesh- and problem-independent dimensionless penalty term is therefore of the form

$$\frac{h_0 h_{max}}{A}\int_\Omega |\nabla\rho_{\mathrm{design}}(\mathbf{x})|^2 d\Omega$$

where h_0 is an initial mesh size, governing the size of details in the solution and h_{max} is the current mesh size at a given level in the process.

This penalty term is, for the worst possible solution, on the order of 1. In order to balance it against the real strain energy objective, it is necessary to normalize also the strain energy. This can be accomplished quite easily simply by solving the structural problem for an initial configuration consisting of homogeneous density $\rho_{\mathrm{design}} = \gamma$ and then divide the strain energy functional by the total strain energy, W_{s0}, found for that configuration.

In addition, the objective and the penalty term must be balanced against each other, for example as a linear combination controlled by a parameter, q, to create a final composite objective function:

$$f = \frac{(1-q)}{W_{s0}}\int_{\Omega} W_s(\mathbf{x})d\Omega + q\frac{h_0 h_{max}}{A}\int_{\Omega} \left|\nabla\rho_{\mathrm{design}}(\mathbf{x})\right|^2 d\Omega$$

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.

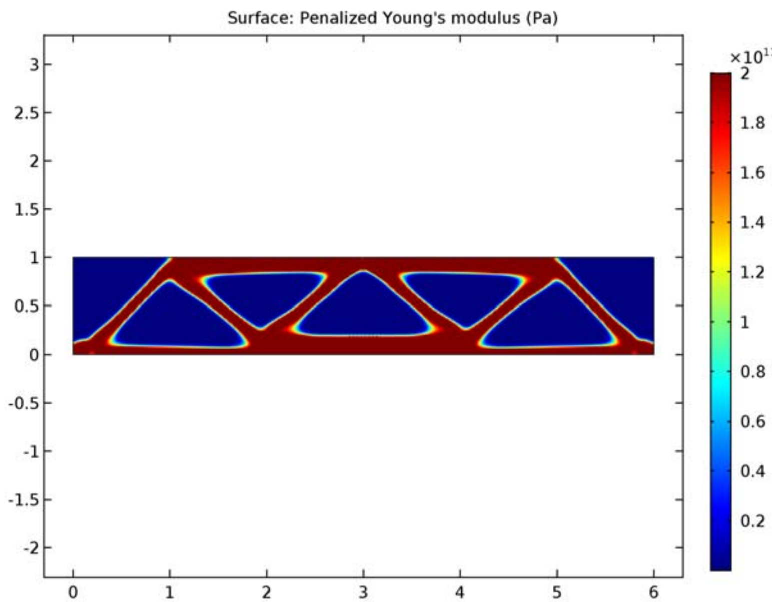


Figure 2: A plot of the penalized stiffness for the optimized design.

Notes About the COMSOL Implementation

A Solid Mechanics interface represents the structural properties of the beam, while an Optimization interface allows adding the control variable, objective, and constraints for the optimization problem. The internal strain energy 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 parametrized, making it easy to experiment with different beam sizes.

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. 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 In the **Select Physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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	3	Half beam width
b	1	1	Beam height
h0	0.05	0.05	Initial mesh size
hmax	0.05	0.05	Current mesh size
p	5	5	SIMP penalization factor
q	0.25	0.25	Regularization factor
gamma	0.5	0.5	Maximum area fraction
domainArea	a*b	3	Area of design domain
Ws0	1	1	Reference objective value

Note that the reference objective value is initially set equal to 1 and must be updated before optimization is started.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **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 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

4 In the **x** text field, type `0.1`.

Point 2 (pt2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **x** text field, type `a-0.1`.

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

5 Right-click **Point 2 (pt2)** and choose **Build Selected**.

ADD MATERIAL

1 On 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 Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Structural steel (mat1)

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

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

Set up the penalized Young's modulus. Material property values can be accessed using the material's identifier and the property group identifier as namespace. Since the control variable `rho_design` is not yet defined, the expression will temporarily be displayed in red, but this is nothing to worry about.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`E_SIMP`	`mat1.Enu.E* rho_design^p`		`Penalized Young's modulus`

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for Solid Mechanics, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

Linear Elastic Material 1

Set Young's modulus to the modified, penalized, expression.

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

2 In the **Settings** window for Linear Elastic Material, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `E_SIMP`.

Roller 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundary 6 only.

Roller 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundary 2 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

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

2 Select Boundary 5 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

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

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

0	x
-100[kN]	y

OPTIMIZATION (OPT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Optimization (opt)**.

Control Variable Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Control Variable Field**.

2 Select Domain 1 only.

3 In the **Settings** window for Control Variable Field, locate the **Control Variable** section.

4 In the **Control variable name** text field, type `rho_design`.

5 In the **Initial value** text field, type `gamma`.

6 Locate the **Discretization** section. From the **Element order** list, choose **Linear**.

Control Variable Bounds 1

1 Right-click **Control Variable Field 1** and choose **Control Variable Bounds**.

2 In the **Settings** window for Control Variable Bounds, locate the **Bounds** section.

3 In the **Lower bound** text field, type `1e-9^(1/p)`.

4 In the **Upper bound** text field, type `1`.

Control Variable Field 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Optimization (opt)>Control Variable Field 1** node.

Integral Objective 1

1 On the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Objective, locate the **Objective** section.

4 In the **Objective expression** text field, type `(1-q)*solid.Ws/Ws0`.

Integral Objective 2

1 On the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Objective, locate the **Objective** section.

4 In the **Objective expression** text field, type `q*h0*hmax*(d(rho_design,x)^2+d(rho_design,y)^2)/domainArea`.

Integral Inequality Constraint 1

1 On the **Physics** toolbar, click **Domains** and choose **Integral Inequality Constraint**.

2 Select Domain 1 only.

3 In the **Settings** window for Integral Inequality Constraint, locate the **Constraint** section.

4 In the **Constraint expression** text field, type `rho_design`.

5 Locate the **Bounds** section. In the **Upper bound** text field, type `gamma*domainArea`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `hmax`.

5 Click **Build All**.

STUDY 1

First compute only a stationary solution on which you can evaluate a reference value for the objective, as well as set up suitable plots.

1 On the **Home** toolbar, click **Compute**.

RESULTS

Data Sets

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

Mirror 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.

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

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

4 In row **Point 2**, set **X** to a and **Y** to b.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

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

4 On the **Stress (solid)** toolbar, click **Plot**.

2D Plot Group 2

1 In the **Model Builder** window, under **Results** click **2D Plot Group 2**.

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

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

4 Right-click **Results>2D Plot Group 2** and choose **Rename**.

5 In the **Rename 2D Plot Group** dialog box, type `Topology` in the **New label** text field.

6 Click **OK**.

Surface 1

Plotting the penalized Young's modulus gives a sharper and fairer picture of the topology, as seen by the Solid Mechanics interface.

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

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

3 In the **Expression** text field, type `E_SIMP`.

4 On the **Topology** toolbar, click **Plot**.

Global Evaluation 1

Evaluate a predefined **Global Evaluation** node to get a reference value for the strain energy objective.

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

TABLE

1 Go to the **Table** window.

Read the value from the results table and update the value of the parameter `Ws0`. This will make the initial value of the integral strain energy objective equal to 1 when optimization is started, making it easier to tune the regularization.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
`Ws0`	`64.7`	64.7	`Reference objective value`

DEFINITIONS

Set up probes for the objective functions and constraint. Scale the objectives such that the values displayed become independent of the regularization factor.

Global Variable Probe 1 (var1)

1. On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2. In the **Settings** window for Global Variable Probe, locate the **Expression** section.
3. In the **Expression** text field, type `opt.iobj1/(1-q)`.
4. Select the **Description** check box.
5. In the associated text field, type `Normalized strain energy`.

Global Variable Probe 2 (var2)

1. On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2. In the **Settings** window for Global Variable Probe, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Optimization>opt.iobj2 - Objective value**.
3. Locate the **Expression** section. In the **Expression** text field, type `opt.iobj2/q`.
4. Select the **Description** check box.
5. In the associated text field, type `Gradient penalty`.

Global Variable Probe 3 (var3)

1. On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2. In the **Settings** window for Global Variable Probe, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Optimization>opt.iconstr1 - Constraint value**.
3. Locate the **Expression** section. In the **Expression** text field, type `opt.iconstr1/(gamma*domainArea)`.

4 Select the **Description** check box.

5 In the associated text field, type `Mass utilization`.

STUDY 1

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

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **MMA**.

4 In the **Optimality tolerance** text field, type `1E-6`.

5 In the **Maximum number of objective evaluations** text field, type `100`.

6 Locate the **Output While Solving** section. Select the **Plot** check box.

7 From the **Plot group** list, choose **Topology**.

8 On the **Study** toolbar, click **Compute**.

GLOBAL DEFINITIONS

Parameters

Reduce the maximum mesh element size to half its original value. This will increase the time required for each iteration, but rather few iterations are sufficient to refine the solution.

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`hmax`	`0.025`	0.025	`Current mesh size`

STUDY 1

Optimization

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

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 In the **Maximum number of objective evaluations** text field, type `50`.

Step 1: Stationary

Change the Stationary step to use the current solution as initial value of all variables. This automatically triggers a mapping from the old coarse mesh to the new, finer.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.
3 Locate the **Values of Dependent Variables** section. 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 **Study 1**.
6 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

Topology

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

   The default plots show the stress distribution in the optimized structure, clearly indicating that the load paths follow the optimized topology. Switch to the other plot group to display a sharper picture of the final design.

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

Multistudy Optimization of a Bracket

Introduction

In some application fields, there is a strong focus on weight reduction. For example, this is the case in the automotive industry, where every gram has a distinct price tag.

In this model, the weight of a mounting bracket is reduced, given an upper bound on the stresses and a lower bound on the first natural frequency.

The bracket is used for mounting a heavy component on a vibrating foundation. It is thus important to keep the natural frequency well above the excitation frequency in order to avoid resonances. The bracket is also subjected to shock loads, which can be treated as a static acceleration load. This gives an optimization problem, where results from two different study types must be considered simultaneously.

Note: This application requires the Structural Mechanics Module, the CAD Module, and the Design Module.

Model Definition

The original bracket together with a sketched mounted component are shown in Figure 1. The bracket is made of steel.

The component, which can be considered as rigid when compared with the bracket, has its center of gravity at the center of the circular cutout in the bracket. The mass is 4.4 kg, the moment of inertia around its longitudinal axis is $7.1 \cdot 10^{-4}$ kg·m^2, and the moment of inertia around the two transverse axes is $9.3 \cdot 10^{-4}$ kg·m^2.

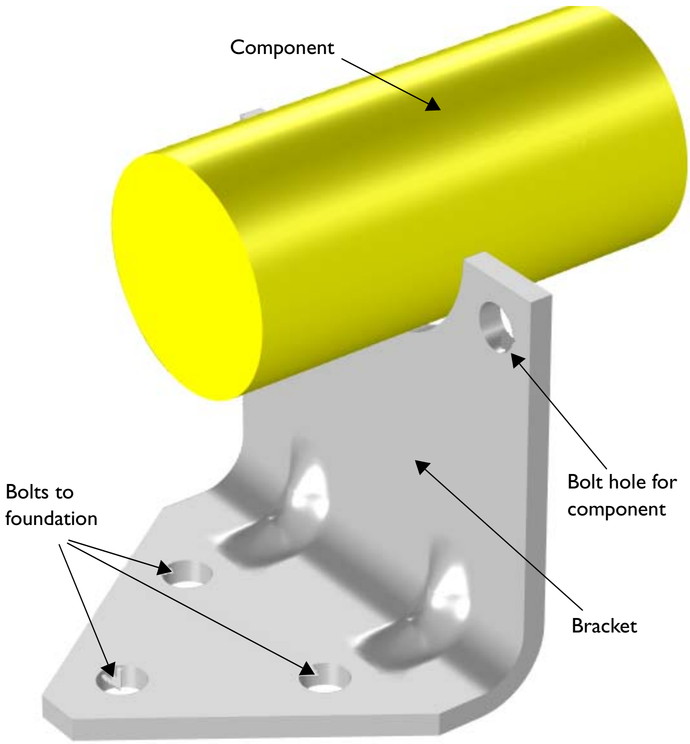


Figure 1: Bracket supporting a heavy component.

The idea is to reduce the weight by drilling holes in the vertical surface of the bracket, and at the same time change the dimensions of the indentations, in order to offset the loss in stiffness.

OPTIMIZATION PARAMETERS

Six geometrical parameters are used in the optimization. They are summarized in Table 1 and shown in Figure 2.

TABLE 1: GEOMETRICAL PARAMETERS

Parameter	Description	Lower limit mm	Upper limit mm
rC	Radius of the central hole	3	15
zCo	Vertical distance from the bend to the edge of the central hole	1	23

TABLE 1: GEOMETRICAL PARAMETERS

Parameter	Description	Lower limit mm	Upper limit mm
rO	Radius of the outer hole	3	15
zOo	Vertical distance from the bend to the edge of the outer hole	3	29
yOo	Horizontal distance from the edge of the bracket to outer hole	8	30
wInd	Width of the indentation	8	20

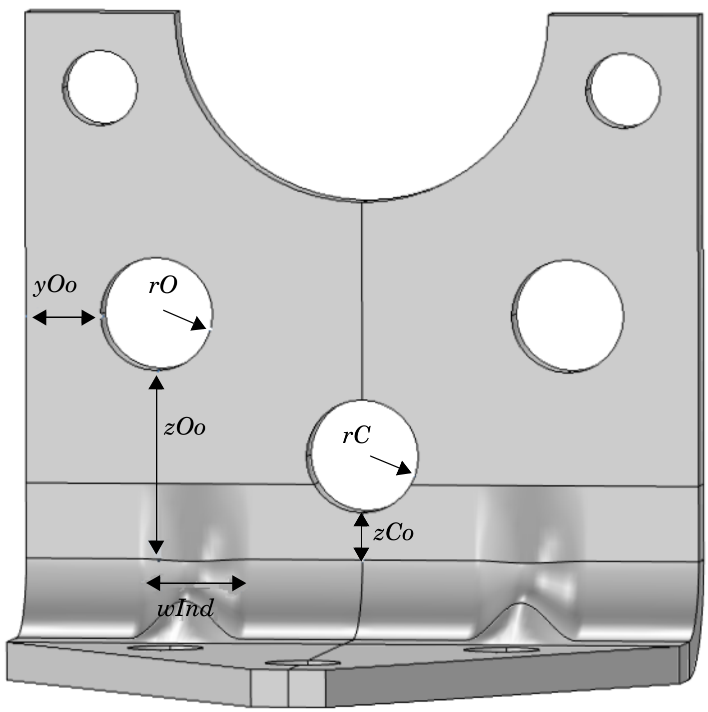


Figure 2: Optimization parameters.

CONSTRAINTS

- The lowest natural frequency must be at least 60 Hz.
- When exposed to a peak acceleration of 4*g* in all three global directions simultaneously, the effective stress is not allowed to exceed 80 MPa anywhere. This criterion is

non-differentiable, because the location of the peak stress can jump from one place to another. A gradient-free optimization algorithm must thus be used.

- There must be at least 3 mm of material between two holes, or between a hole and an edge. This criterion is enforced both through the limits on the control parameters and as constraints. The geometrical constraints are shown in Figure 3.

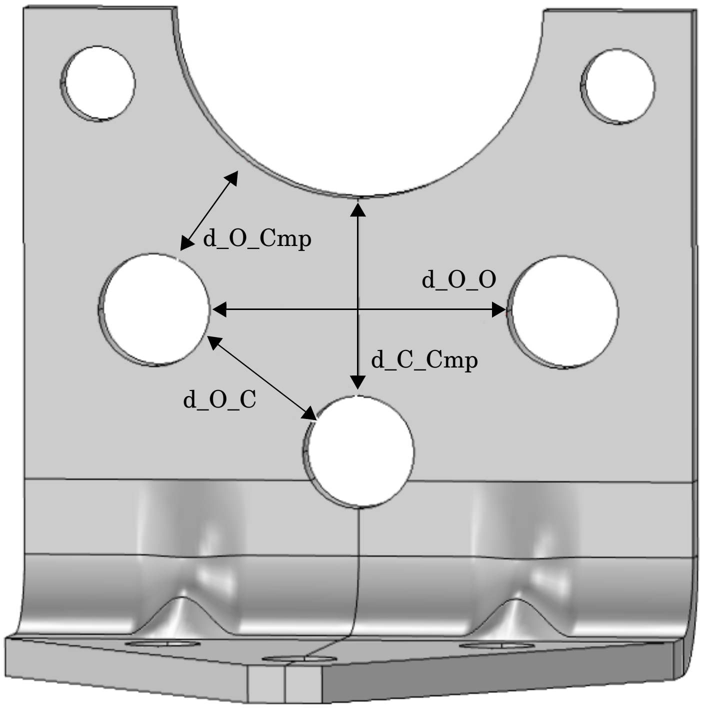


Figure 3: Geometrical constraints.

The COBYLA solver uses sampling in the control variable space to approximate both the objective function, the constraints, and the control variable bounds. Individual samples may be computed outside the bounds and in violation of the constraints. Therefore, it is important to parameterize the geometry in such a way that it is robust with respect to (small) constraint and bound violations.

Bounds and linear constraints are generally satisfied to high precision at the optimum point returned by the solver, but nonlinear constrains are often slightly violated. The reason is that the solver tends to converge from the outside of the feasible domain and terminates before the constraints are completely satisfied. Tightening the solver tolerances will decrease the constraint violation but is often not worth the computational effort; it is better to specify constraints with a safety margin.

Results and Discussion

The initial geometry used in the optimization is shown in Figure 4. Three rather small holes have been introduced.

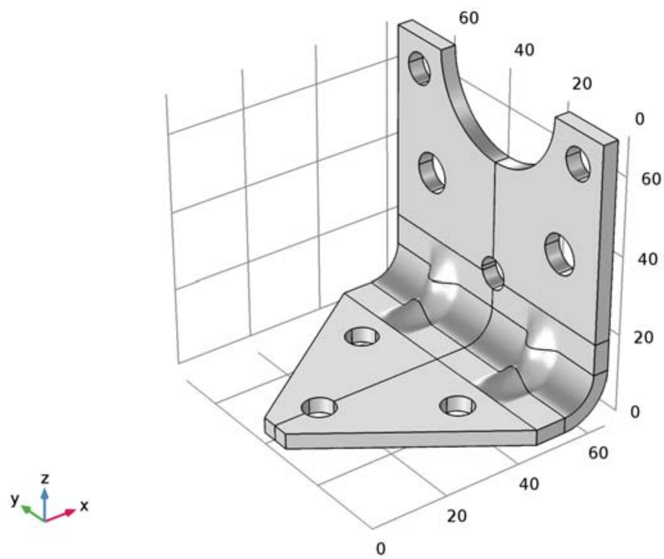


Figure 4: Initial geometry.

The optimal values of the geometrical parameters are shown in Table 2.

TABLE 2: OPTIMAL VALUES

PARAMETER	OPTIMAL VALUE MM	LOWER LIMIT MM	UPPER LIMIT MM
rC	11.8	3	15
zCo	2.7	1	23
rO	9.0	3	15
zOo	9.5	3	29
yOo	22.3	8	30
$wInd$	20	8	20

The weight of the optimized bracket is about 178 g, a reduction of 27 g from the original 205 g. The stresses from the shock load on the optimized geometry are shown in Figure 5

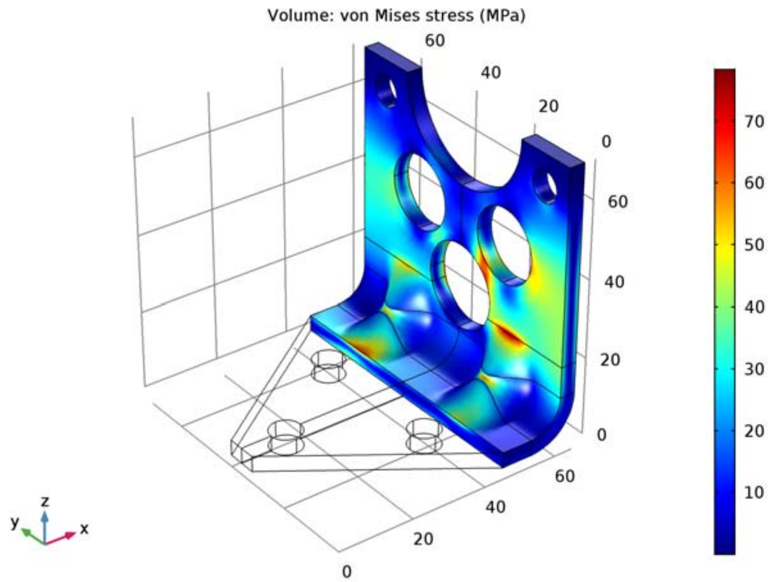


Figure 5: Stresses at peak load in the optimized design.

The optimal solution gives three fairly large holes, and the widest possible indentation.

There are several possible arrangements of the holes that give the same weight reduction within a small tolerance. It is therefore possible that the design variables are not always the same at convergence.

Notes About the COMSOL Implementation

The component mounted on the bracket is not modeled in detail. It is replaced by a Rigid Connector having the equivalent inertial properties.

Application Library path: `Optimization_Module/Shape_Optimization/multistudy_bracket_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 **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>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

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

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root nodes'**Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **MPa**.

Start building the geometry.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

4 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zy-plane**.

4 In the **x-coordinate** text field, type `lX-rOut`.

5 Click **Show Work Plane**.

Parametric Curve 1 (pc1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for Parametric Curve, locate the **Expressions** section.

3 In the **xw** text field, type `if((abs(s-0.5)<wInd/lY),dInd/2*(1+cos(pi*lY/if(wInd>4[mm],wInd,4[mm])*(s-0.5))),0)`.

4 In the **yw** text field, type `s*lY/2`.

5 Right-click **Parametric Curve 1 (pc1)** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **pc1** only.

3 In the **Settings** window for Copy, locate the **Displacement** section.

4 In the **xw** text field, type `thk`.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **2**, set **xw** to `thk`.

Copy 2 (copy2)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **b1** only.

3 In the **Settings** window for Copy, locate the **Displacement** section.

4 In the **yw** text field, type `lY/2`.

Convert to Solid 1 (csol1)

1 On the **Work Plane** toolbar, click **Conversions** and choose **Convert to Solid**.

2 In the **Model Builder** window, right-click **Convert to Solid 1 (csol1)** and choose **Build Selected**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for Revolve, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `90`.

5 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

6 Find the **Point on the revolution axis** subsection. In the **x** text field, type `lX-rOut`.

7 In the **z** text field, type `rOut`.

8 Find the **Direction of revolution axis** subsection. In the **y** text field, type `-1`.

9 Click **Build All Objects**.

Block 1 (blk1)

1 On 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 `lX-rOut-2*thk`.

4 In the **Depth** text field, type `lY/2`.

5 In the **Height** text field, type `thk`.

6 Click **Build All Objects**.

Loft 1 (loft1)

1 On the **Geometry** toolbar, click **Loft**.

2 In the **Settings** window for Loft, locate the **General** section.

3 Clear the **Unite with input objects** check box.

4 Click to expand the **Start profile** section. Locate the **Start Profile** section. Find the **Start profile** subsection. Select the **Active** toggle button.

5 On the object **rev1**, select Boundary 1 only.

6 Click to expand the **End profile** section. Locate the **End Profile** section. Find the **End profile** subsection. Select the **Active** toggle button.

7 On the object **blk1**, select Boundary 5 only.

8 Click **Build All Objects**.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **loft1** only.

3 In the **Settings** window for Mirror, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Plane of Reflection** section. In the **x** text field, type `lX-rOut`.

6 In the **z** text field, type `rOut`.

7 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type `1`.

8 Click **Build All Objects**.

Block 2 (blk2)

1 On 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 `thk`.

4 In the **Depth** text field, type `lY/2`.

5 In the **Height** text field, type `lZ-rOut-2*thk`.

6 Locate the **Position** section. In the **x** text field, type `lX-thk`.

7 In the **z** text field, type `rOut+2*thk`.

8 Click **Build All Objects**.

9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cylinder 1 (cyl1)

1 On 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 `dCmp/2`.

4 In the **Height** text field, type `3*thk`.

5 Locate the **Position** section. In the **x** text field, type `lX-2*thk`.

6 In the **y** text field, type `lY/2`.

7 In the **z** text field, type `lZ`.

8 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

9 Click **Build All Objects**.

Cylinder 2 (cyl2)

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `bDia/2`.

4 Locate the **Position** section. In the **y** text field, type `bDia`.

5 In the **z** text field, type `lZ-bDia`.

6 Click **Build All Objects**.

Cylinder 3 (cyl3)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 2 (cyl2)** and choose **Duplicate**.

2 In the **Settings** window for Cylinder, locate the **Position** section.

3 In the **x** text field, type `lX-rOut-2*thk-bDia`.

4 In the **y** text field, type `lY/4`.

5 In the **z** text field, type `-thk`.

6 Locate the **Axis** section. From the **Axis type** list, choose **z-axis**.

7 Click **Build All Objects**.

Cylinder 4 (cyl4)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 3 (cyl3)** and choose **Duplicate**.

2 In the **Settings** window for Cylinder, locate the **Position** section.

3 In the **x** text field, type `1.5*bDia`.

4 In the **y** text field, type `lY/2`.

5 Click **Build All Objects**.

Cylinder 1 (cyl1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Cylinder 5 (cyl5)

1 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

2 In the **Radius** text field, type `r0`.

3 Locate the **Position** section. In the **y** text field, type `y0`.

4 In the **z** text field, type `z0`.

5 Click **Build All Objects**.

Cylinder 6 (cyl6)

1 Right-click **Component 1 (comp1)>Geometry 1>Cylinder 5 (cyl5)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `rC`.

4 Locate the **Position** section. In the **y** text field, type `lY/2`.

5 In the **z** text field, type `zC`.

6 Click **Build All Objects**.

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yx-plane**.

4 In the **z-coordinate** text field, type `2*thk`.

5 Click **Show Work Plane**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

4 Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **2**, set **xw** to `lY/2-bDia/2`.

6 Find the **Added segments** subsection. Click **Add Linear**.

7 Find the **Control points** subsection. In row **2**, set **yw** to `lX-rOut-2*thk` and **xw** to `0`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. Click **Close Curve**.

10 Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

Work Plane 2 (wp2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (mm)
3*thk

Difference 1 (dif1)

1 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

3 Select the objects **rev1**, **blk1**, **mir1**, **blk2**, and **loft1** only.

4 In the **Settings** window for Difference, locate the **Difference** section.

5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the objects **cyl5**, **cyl6**, **cyl3**, **cyl2**, **ext1**, **cyl1**, and **cyl4** only.

7 From the **Repair tolerance** list, choose **Relative**.

8 Click **Build All Objects**.

Mirror 2 (mir2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

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

3 Select the **Keep input objects** check box.

4 Locate the **Point on Plane of Reflection** section. In the **y** text field, type `lY/2`.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

6 In the **z** text field, type `0`.

7 Click **Build All Objects**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

ADD MATERIAL

1 On 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 Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

   The exact way the bolts clamp the bracket to the foundation is not important for the results in the part being optimized.

2 In the **Settings** window for Fixed Constraint, type `Fixed (Bolts)` in the **Label** text field.

3 Select Boundaries 10–13 and 15–22 only.

Rigid Connector 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

   The attached component has a high stiffness, and is bolted to the two upper bolt holes. It is modeled as being rigid, with only mass properties.

2 In the **Settings** window for Rigid Connector, type `Rigid Connector (Mounted component)` in the **Label** text field.

3 Select Boundaries 48, 49, 52, 53, and 75–78 only.

4 Locate the **Center of Rotation** section. From the list, choose **User defined**.

5 Specify the $\mathbf{X}_c$ vector as

`lX-thk/2`	X
`lY/2`	Y
`lZ`	Z

Mass and Moment of Inertia 1

1 On the **Physics** toolbar, click **Attributes** and choose **Mass and Moment of Inertia**.

2 In the **Settings** window for Mass and Moment of Inertia, locate the **Mass and Moment of Inertia** section.

3 In the m text field, type `mCmp`.

4 From the list, choose **Diagonal**.

5 In the **I** table, enter the following settings:

`IXCmp`	`0`	`0`
`0`	`IYZCmp`	`0`
`0`	`0`	`IYZCmp`

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

Free Triangular 1

Select Boundaries 4, 8, 26, 30, 35, 39, 41, 44, 60, and 64 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
2 In the **Settings** window for Size, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 Select Boundaries 4, 8, 26, 30, 44, and 64 only.

Size 2

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1>Size 1** and choose **Duplicate**.
2 In the **Settings** window for Size, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extra fine**.
4 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.
5 Select Boundaries 35, 39, 41, and 60 only.
6 Click **Build Selected**.
7 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.
2 In the **Number of elements** text field, type `3`.
3 Click **Build All**.

STUDY 1

Run an eigenfrequency study on the initial geometry.

1 In the **Settings** window for Study, type `Eigenfrequency study` in the **Label** text field.
2 On the **Home** toolbar, click **Compute**.

SOLID MECHANICS (SOLID)

Add the peak loads, and a perform a stationary study.

Body Load 1

1 On the **Physics** toolbar, click **Domains** and choose **Body Load**.

2 In the **Settings** window for Body Load, type `Body load 4g on bracket` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

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

4*g_const*solid.rho	x
4*g_const*solid.rho	y
4*g_const*solid.rho	z

Rigid Connector (Mounted component)

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Rigid Connector (Mounted component)** and choose **Applied Force**.

Applied Force 1

1 In the **Settings** window for Applied Force, type `Force 4g on mounted component` in the **Label** text field.

2 Locate the **Applied Force** section. Specify the **F** vector as

4*g_const*mCmp	x
4*g_const*mCmp	y
4*g_const*mCmp	z

ADD STUDY

1 On 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 **Preset Studies>Stationary**.

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

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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

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

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

DEFINITIONS

Prepare for the optimization by adding variables for the bracket mass and the maximum stress.

Integration 1 (intop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2. In the **Settings** window for Integration, locate the **Source Selection** section.
3. From the **Selection** list, choose **All domains**.

Maximum 1 (maxop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.
2. In the **Settings** window for Maximum, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 25, 26, 29, 30, 34, 35, 38, 39, 41, 44, 60, 64, and 81–84 only.

Variables 1

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for Variables, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
`mass`	`intop1(solid.rho)`	t	`Bracket mass`
`maxStress`	`maxop1(solid.mises)`	MPa	`Maximum stress`

RESULTS

Add a plot for monitoring the geometry and stresses in the optimized region.

3D Plot Group 3

1. On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for 3D Plot Group, type `Stress in optimized region` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Stationary study/Solution 2 (sol2)**.

Volume 1

1. Right-click **Stress in optimized region** and choose **Volume**.
2. In the **Settings** window for Volume, locate the **Expression** section.
3. In the **Expression** text field, type `solid.mises`.

Filter 1

1 Right-click **Results>Stress in optimized region>Volume 1** and choose **Filter**.
2 In the **Settings** window for Filter, locate the **Element Selection** section.
3 In the **Logical expression for inclusion** text field, type `dom>2`.
4 From the **Element nodes to fulfill expression** list, choose **All**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

ROOT

Set up the optimization study.

ADD STUDY

1 On 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 **Empty Study**.
4 Click **Add Study** in the window toolbar.
5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.
2 In the **Settings** window for Study, type `Optimization study` in the **Label** text field.

Optimization

On the **Study** toolbar, click **Optimization**.

OPTIMIZATION STUDY

In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

No Study

1 On the **Study** toolbar, click **Study Reference**.
2 In the **Settings** window for Study Reference, type `Eigenfrequency` in the **Label** text field.
3 Locate the **Study Reference** section. From the **Study reference** list, choose **Eigenfrequency study**.
4 On the **Study** toolbar, click **Study Reference**.
5 In the **Settings** window for Study Reference, type `Stationary` in the **Label** text field.

6 Locate the **Study Reference** section. From the **Study reference** list, choose **Stationary study**.

Optimization

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

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **COBYLA**.

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

Expression	Description	Evaluate for
comp1.mass	Bracket mass	ref2

5 Locate the **Optimization Solver** section. In the **Optimality tolerance** text field, type 0.1.

The first eigenfrequency is to be used in the optimization.

6 Locate the **Objective Function** section. From the **Solution** list, choose **Use first**.

7 Locate the **Control Variables and Parameters** section. Click **Load from File**.

8 Browse to the application's Application Libraries folder and double-click the file multistudy_bracket_optimization_ctrlvars.txt.

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

Expression	Lower bound	Upper bound	Evaluate for
real(comp1.solid.freq)	minFreq		ref
comp1.maxStress/maxStressLimit		1	ref2
d_O_Cmp	3		ref
d_C_Cmp	3		ref
d_O_C	3		ref
d_O_O	3		ref

10 Locate the **Output While Solving** section. Select the **Plot** check box.

11 From the **Plot group** list, choose **Stress in optimized region**.

If some configurations are not valid, the optimization procedure should still continue. The default is to stop if an error occurs.

Solution 3 (sol3)

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

Parametric Sweep 1

1 In the **Settings** window for Parametric Sweep, locate the **Error** section.

2 Clear the **Stop if error** check box.

Parametric Sweep 2

1 In the **Model Builder** window, under **Optimization study>Job Configurations** click **Parametric Sweep 2**.

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

3 Clear the **Stop if error** check box.

Parametric Sweep 3

1 In the **Model Builder** window, under **Optimization study>Job Configurations** click **Parametric Sweep 3**.

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

3 Clear the **Stop if error** check box.

Run the optimization.

4 On the **Study** toolbar, click **Compute**.

RESULTS

On the last line of **Objective Table 2** you will find the optimal set of parameters, and the minimum weight. Note that the value in the **Objective** table can be colored orange if the solution violates a constraint slightly, but is still accepted within the tolerances.

On the last line of **Global Constraints Table 6** you will find the values of the natural frequency and maximum stress in the optimized configuration, as well as the values of the other constraints.

Examine the stress distribution in the optimized configuration.

Stress in optimized region

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results>Stress in optimized region**.

2 On the **Stress in optimized region** toolbar, click **Plot**.

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

Created in COMSOL Multiphysics 5.2a

Minimizing the Flow Velocity in a Microchannel

Introduction

Topology optimization of the Navier-Stokes equations is encountered in different branches and applications, such as in the design of ventilation systems for cars and optimal reactors. A common technique applicable to such problems is to let the distribution of porous material vary continuously. In this model, the objective is to find the optimal distribution of a porous material in a microchannel such that the horizontal velocity component at the center of the channel is minimized.

The model is inspired by Ref. 1.

Model Definition

The model geometry (Figure 1) consists of three regions: the inlet channel, the design domain, and the outlet channel. A prescribed pressure drop between the inlet and the outlet drives the flow.

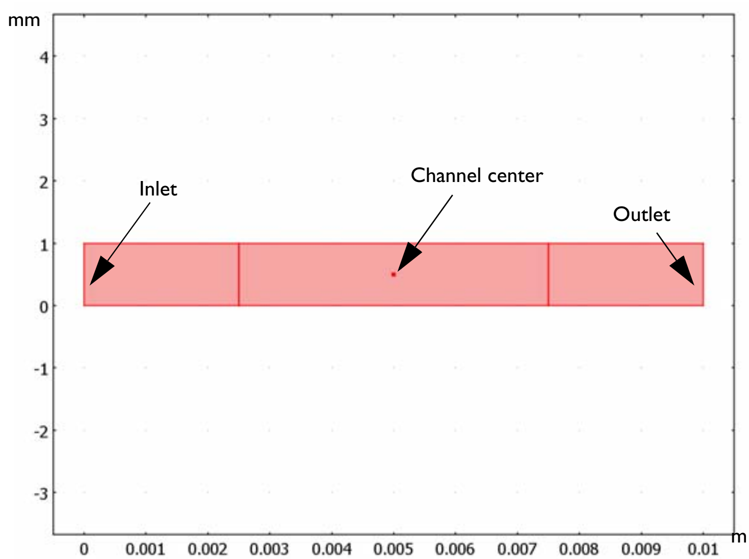


Figure 1: The model geometry.

The fluid flow in the channel is described by the Navier-Stokes equations

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \nabla\cdot\eta(\nabla\mathbf{u}+(\nabla\mathbf{u})^T) - \alpha(\gamma)\mathbf{u}$$
$$\nabla\cdot\mathbf{u} = 0$$

where $-\alpha(\gamma)\mathbf{u}$ is a force term in which the coefficient

$$\alpha(\gamma) \equiv \alpha_{\max}\frac{q(1-\gamma)}{q+\gamma} \tag{1}$$

characterizes the flow in a porous medium. In Equation 1, α is interpreted as a continuous mapping determined by the function γ: $\Omega \to [0, 1]$, which in the limit of decreasing Darcy number and decreasing mesh size should be a discrete-valued function. When γ equals 1, α is zero, corresponding to free flow. Conversely, for $\gamma = 0$, $\alpha = \alpha_{\max}$, where $\alpha_{\max}$ is related to the dimensionless Darcy number, Da, according to

$$\mathrm{Da} = \frac{\eta}{\alpha_{\max}L^2}$$

The convergence of the optimization process depends on three important factors: the Darcy number, the mesh size, and the coefficient q. Rewriting Equation 1 it is easily seen that $\alpha/\alpha_{\max} \to 1-\gamma$ in the limit $q \to \infty$. In this limit, γ can be interpreted as the local porosity, ranging between 0 (filled) and 1 (open channel).

Results and Discussion

Figure 2 displays the design variable, γ, which represents the distribution of porous material. As the plot shows, γ is either 0 or 1 in most of the domain, with a narrow transition zone in between. The width of this transition zone is mesh dependent; you can reduce it by decreasing the mesh-element size. Alternatively, decreasing the Darcy number also gives harder boundaries at the interface between porous and open domains.

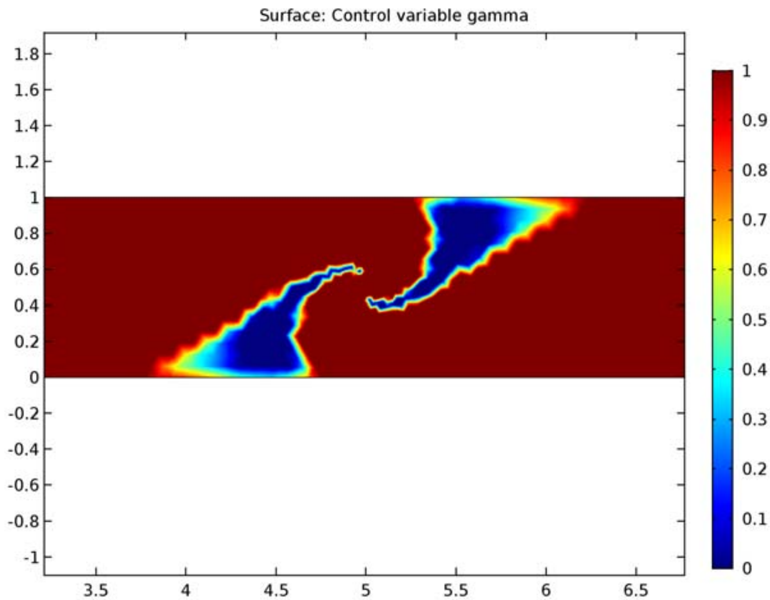


Figure 2: Distribution of porous material; the red areas represent open channel.

A question that naturally arises in this type of problems concerns uniqueness of the solution. In this case, there is at least one more solution that gives exactly the same result; because the channel has no upside or downside, a solution mirrored around the axis $y = 0.5$ mm would give exactly the same flow.

Figure 3 contains a surface plot of the horizontal velocity component and a streamline plot of the velocity field resulting from the optimization process. In addition, the contour $\gamma = 0.5$ indicates the border between the open channel and filling material. The plots reveal how the flow turns around, with a negative horizontal velocity at the center of the channel. Note also that the x-velocity has a minimum of roughly -14 mm/s at the design point.

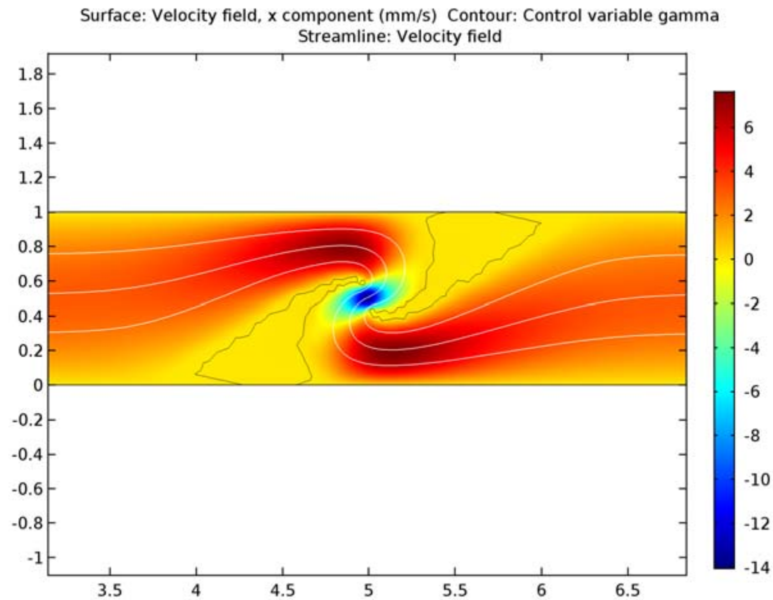


Figure 3: The horizontal velocity (surface plot) and velocity field (streamlines) after optimization. In addition, the contour γ = 0.5 indicates the border between open channel and filling material.

If you were to increase the streamline density in the above plot, some streamlines passing through the barriers would appear. This effect is due to a small amount of leakage, which can be reduced further by increasing the mesh resolution.

Figure 4 shows the pressure distribution, verifying the prescribed pressure drop through the channel length. The pressure drop is naturally concentrated to the region with porous material.

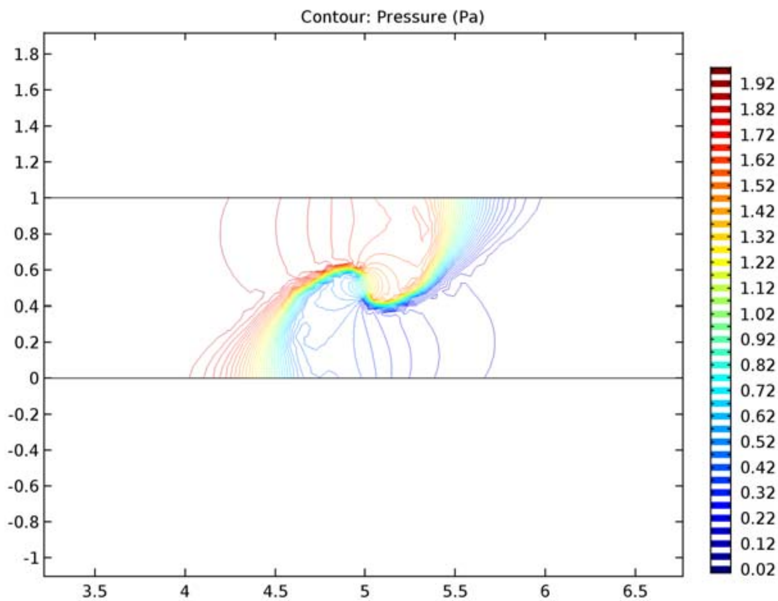


Figure 4: Pressure distribution in the channel, the pressure drop is concentrated to the porous domain.

Reference

1. L. Højgard Olesen, F. Okkels, and H. Bruus, "A High-level Programming-language Implementation of Topology Optimization Applied to Steady-state Navier-Stokes Flow," *Int. J. Num. Meth. Engrg*, vol. 65, pp. 975–1001, 2005.

Notes About the COMSOL Implementation

This model combines the Optimization and Laminar Flow interfaces. First, you calculate the solution for the flow in an empty channel (that is, with no porous material). You then solve the optimization problem. In each iteration, the software calculates a solution for the flow problem and feeds it to the optimization routine, which updates the design variables.

If the specified convergence criterion is fulfilled, the solution process terminates; otherwise the new design-variable values are used in the next calculation of the flow problem.

However, for design problems such as this one, there is a trade-off between computation time and convergence. The solution may be sufficiently improved long before convergence is reached in the mathematical sense. Therefore, it is useful to limit the number of steps taken by the optimization algorithm after which the solution can be evaluated and restarted if not yet satisfactory.

Setting up this kind of model with a general optimization routine requires quite a bit of work, but as you will discover, solving this problem with the built-in tools for optimization in COMSOL Multiphysics is easy.

Application Library path: `Optimization_Module/Topology_Optimization/reversed_flow`

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 **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
q	1	1	Optimization parameter
Da	1e-5	1E-5	Darcy number
L	1[mm]	0.001 m	Inlet height
u0	10[mm/s]	0.01 m/s	Flow velocity scale

The purpose of the constant u0 is to introduce a rough velocity scale that you can use to make the objective function dimensionless with a value of the order 1.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 2.5.

4 In the **Height** text field, type 1.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 5.

4 In the **Height** text field, type 1.

5 Locate the **Position** section. In the **x** text field, type 2.5.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 2.5.

4 In the **Height** text field, type 1.

5 Locate the **Position** section. In the **x** text field, type 7.5.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **x** text field, type `5`.

4 In the **y** text field, type `0.5`.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

The geometry should now look like that in Figure 1.

ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for Variables, locate the **Geometric Entity Selection** section.

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

4 Select Domain 2 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`alpha_max`	`spf.mu/(Da*L^2)`	Pa·s/m²	`Volume force coefficient, max value`
`alpha`	`alpha_max*q*(1-gamma)/(q+gamma)`		`Volume force coefficient`

LAMINAR FLOW (SPF)

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

3 In the **Settings** window for Laminar Flow, click to expand the **Discretization** section.

4 From the **Discretization of fluids** list, choose **P2+P1**.

This setting gives quadratic elements for the velocity field.

Volume Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Volume Force**.

2 Select Domain 2 only.

3 In the **Settings** window for Volume Force, locate the **Volume Force** section.

4 Specify the **F** vector as

`-alpha*u`	x
`-alpha*v`	y

In the next steps, you specify the pressure drop along the channel length by prescribing the pressure at the inlet and at the outlet. The resulting pressure gradient drives the flow in the channel.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for Inlet, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `2`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 10 only.

Keep all other boundaries at the default condition, which is the no-slip condition.

OPTIMIZATION (OPT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Optimization (opt)**.

Control Variable Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Control Variable Field**.

Only the center part of the channel geometry is needed in the optimization, so you only have to define the control variable there.

2 Select Domain 2 only.

3 In the **Settings** window for Control Variable Field, locate the **Control Variable** section.

4 In the **Control variable name** text field, type gamma.

5 In the **Initial value** text field, type 1.

6 Locate the **Discretization** section. From the **Element order** list, choose **Linear**.

This defines shape functions for gamma, which is the design variable used in the optimization. The initial value 1 corresponds to a channel free from porous material. In the next step, you constrain the design variable to the range [0,1].

Control Variable Bounds 1

1 In the **Model Builder** window, right-click **Control Variable Field 1** and choose **Control Variable Bounds**.

2 In the **Settings** window for Control Variable Bounds, locate the **Bounds** section.

3 In the **Upper bound** text field, type 1.

Point Sum Objective 1

1 On the **Physics** toolbar, click **Points** and choose **Integral Objective**.

2 Select Point 5 only.

3 In the **Settings** window for Point Sum Objective, locate the **Objective** section.

4 In the **Objective expression** text field, type u/u0.

This defines the objective function to be proportional to the x-component of the velocity at the center and normalized by the velocity-scale constant u0.

MESH 1

This model is naturally highly dependent on the mesh size. In this example, choose a dense mesh; note, however, that this mesh can be further improved.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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 growth rate** text field, type 1.1.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

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

3 Select Domain 2 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.08`.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Point**.

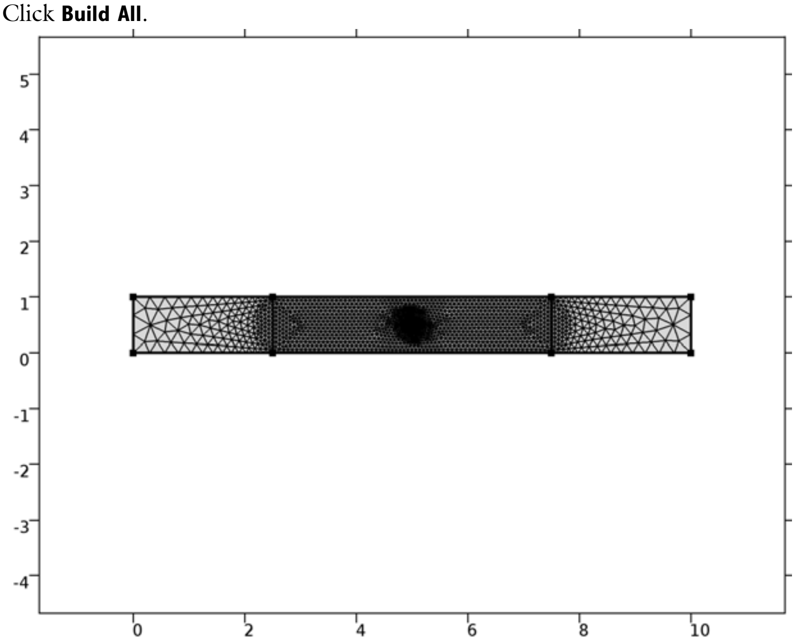
3 Select Point 5 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.02`.

7 Click **Build All**.



Although you can choose to solve the optimization problem directly, it can be good to check that the initial solution looks reasonable.

STUDY 1

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

RESULTS

Velocity (spf)

The resulting plots show the magnitude of the velocity field, the pressure, and the design variable distribution in the channel. Proceed to solve the optimization problem.

STUDY 1

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **MMA**.

4 In the **Maximum number of objective evaluations** text field, type `150`.

5 Locate the **Output While Solving** section. Select the **Plot** check box.

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

RESULTS

Velocity (spf)

The default plot shows the magnitude of the velocity field in the channel. Generate Figure 3 with the following instructions:

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>Velocity field>u - Velocity field, x component**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm/s**.

4 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Contour**.

5 In the **Settings** window for Contour, locate the **Expression** section.

6 In the **Expression** text field, type `gamma`.

7 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

8 In the **Levels** text field, type `0.5`.

9 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

10 From the **Color** list, choose **Black**.

11 Clear the **Color legend** check box.

12 Right-click **Velocity (spf)** and choose **Streamline**.

13 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v - Velocity field**.

14 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

15 In the **Density** text field, type 8.

16 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

17 On the **Velocity (spf)** toolbar, click **Plot**. Click the **Zoom Box** button on the **Graphics** toolbar and then use the mouse to zoom in.

Pressure (spf)

The second default plot shows the pressure distribution Figure 4.

2D Plot Group 3

The last plot shows the gamma distribution in the channel Figure 2.

Time-Dependent Optimization

Introduction

Nonlinear systems that are driven by a sinusoidal excitation often evolve toward a periodic steady-state solution. Such systems occur in electromagnetics, plasma physics, and electrochemistry.

Model Definition

The test model solves the following ordinary differential equation:

$$\left(\frac{1}{f}\right)\frac{du}{dt} = a\sin^2(\omega t) - bu - cu^2 \tag{1}$$

where a is 0.25, b is 0.05, and c is 0.015. The frequency, f, is set to 1. Because the equation is nonlinear (due to the u^2 term), it cannot be reformulated in the frequency domain by taking its Fourier transform. This ordinary differential equation is representative of the evolution of electronically excited metastable states in a capacitively coupled plasma. The initial value of u is set to be 0.25. For these conditions, it takes about 100 periods before u reaches its periodic steady-state solution. In a real plasma, it can take more than $100{,}000$ RF cycles before the plasma has attained its periodic steady-state solution. Solving such a problem for so many RF cycles creates an insurmountable computational burden.

The periodic steady-state solution can be immediately computed using time-dependent optimization. A control variable, u_0, is used as the initial condition for Equation 1. Next, an objective function is defined as:

$$g = (u - u_0)^2$$

When performing time-dependent optimization, the objective function is only evaluated at the last solution time. Thus, the global objective function seeks to make the initial value of u equal to the final value of u after exactly one period. This corresponds to the periodic steady-state solution to the problem.

Results and Discussion

The time evolution of u is plotted in Figure 1. A close-up of the final few periods is plotted in Figure 2. This shows that u has indeed reached its periodic steady-state solution.

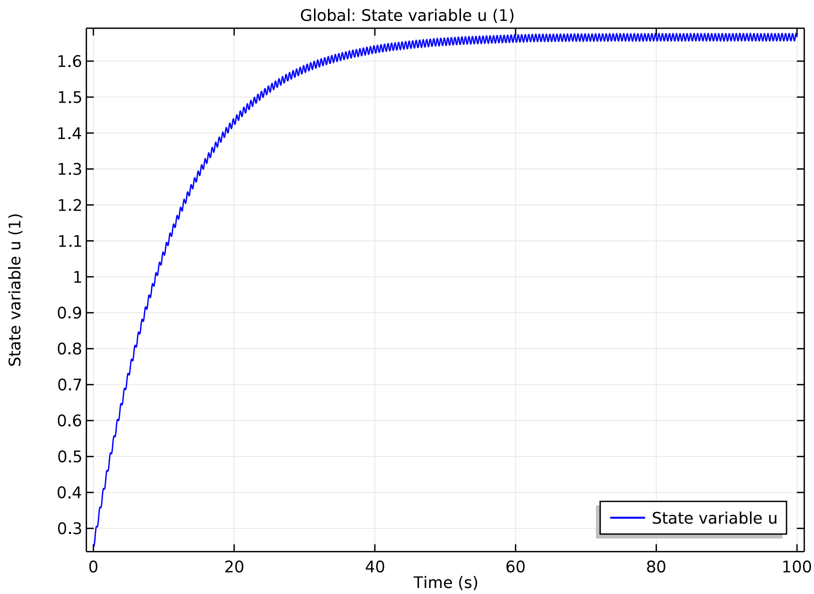


Figure 1: Plot of the evolution of u from its initial value of 0.25. There is a slow, steady increase in u over the first several periods along with oscillations at twice the driving frequency.

It is also obvious from Figure 2 that over 1 period, the value of u at the beginning of the period is the same as at the end of the period. In Figure 3 the solution computed by the optimization solver is shown. Note that the forward problem is only solved for 1 period. In total the optimization solver computes the solution to the forward problem only 6 times resulting in a much reduced simulation time.

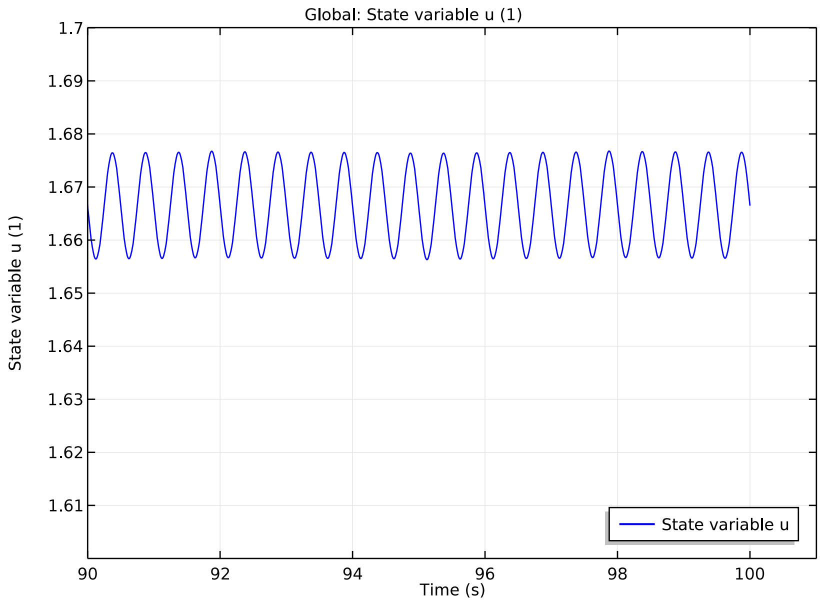


Figure 2: Close up of the last several cycles of the forward problem. The model has clearly reached its periodic steady state solution after 100 cycles.

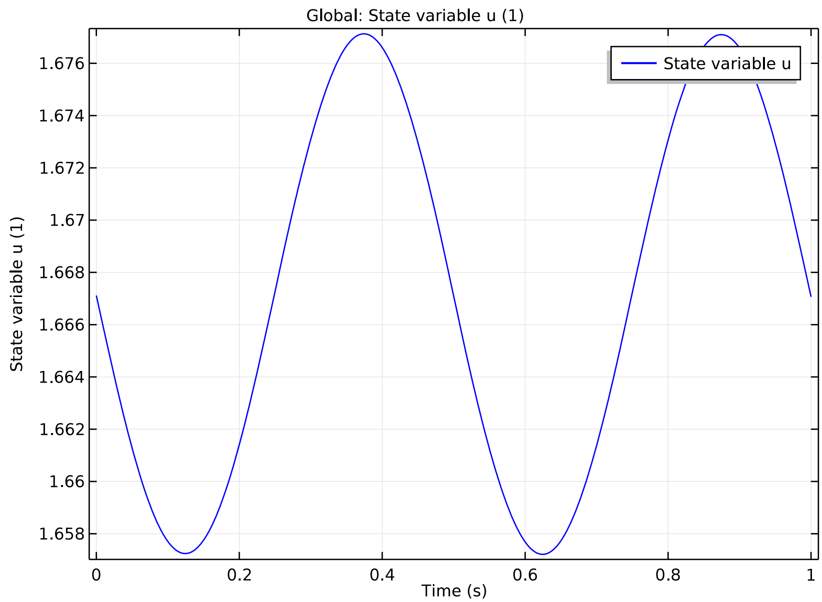


Figure 3: Plot of the solution computed by the optimization solver.

Reference

1. D.P. Lymberopoulos and D.J. Economou, "Fluid simulation of glow discharges: Effect of metastable atoms in argon," *J. Appl. Phys.* vol. 73, no. 8, 1993.

Application Library path: `Optimization_Module/Parameter_Estimation/time_dependent_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 **0D**.
2. In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
a	0.25	0.25	ODE constant 1
b	0.05	0.05	ODE constant 2
c	0.015	0.015	ODE constant 3
u0	0.25	0.25	Initial value
f	1[Hz]	1 Hz	Frequency
w	2*pi*f	6.2832 Hz	Angular frequency

GLOBAL ODES AND DAES (GE)

Define the ordinary differential equation with the periodic forcing function.

Global Equations 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.
2. In the **Settings** window for Global Equations, locate the **Global Equations** section.
3. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
u	((1/f)*ut-a*sin(w*t)^2+b*u+c*u^2)	u0	0

STUDY 1

The model needs to be solved for 100 periods before it reaches its periodic steady state solution.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
3. Select the **Relative tolerance** check box.
4. In the associated text field, type `1e-5`.
5. In the **Times** text field, type `range(0,0.01,100)`.

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
3. In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.
4. Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `0.0001`.
5. On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

1. In the **Model Builder** window, under **Results** click **1D Plot Group 1**.
2. In the **Settings** window for 1D Plot Group, click to expand the **Legend** section.
3. From the **Position** list, choose **Lower right**.
4. On the **1D Plot Group 1** toolbar, click **Plot**.

1D Plot Group 2

1. Right-click **Results>1D Plot Group 1** and choose **Duplicate**.
2. In the **Settings** window for 1D Plot Group, click to expand the **Axis** section.
3. Select the **Manual axis limits** check box.
4. In the **x minimum** text field, type `90`.
5. In the **y minimum** text field, type `1.6`.
6. In the **y maximum** text field, type `1.7`.
7. On the **1D Plot Group 2** toolbar, click **Plot**.

ROOT

Now add another study with an optimization step which can be used to immediately compute the periodic steady state solution for the differential equation.

ADD STUDY

1. On 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 **Preset Studies>Time Dependent**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
2. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
3. In the **Times** text field, type `range(0,0.002,1)`.
4. Select the **Relative tolerance** check box.
5. In the associated text field, type `1e-5`.

Optimization

1. On the **Study** toolbar, click **Optimization**.
2. In the **Settings** window for Optimization, locate the **Optimization Solver** section.
3. From the **Method** list, choose **SNOPT**.

   Add the difference between initial and final value in a cycle as error measure to be minimized.
4. Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
`(comp1.u-u0)^2`	`Squared error`	time

   Next, add the initial value as control parameter and set suitable bounds to help the solver.

5. Locate the **Control Variables and Parameters** section. Click **Add**.

6 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
u0	0.25	1	0	5

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.

5 Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `1e-5`.

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

The solver will issue a warning as a reminder that the objective function is only evaluated at the final time—which is indeed the desired behavior in this model.

RESULTS

1D Plot Group 3

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The periodic steady state solution is obtained (compare to Figure 2).

Created in COMSOL Multiphysics 5.2a

Shape Optimization of a Tuning Fork

Introduction

This model extends the model "Tuning Fork" in the COMSOL Multiphysics Application Libraries by adding a second study, in which the Parametric Sweep is replaced by an Optimization study node. The prong length L is determined by minimizing the objective function abs(f – 440 Hz), where f is the fundamental frequency of the fork. The result agrees with that found in the original model version. For a detailed description of the model geometry and setup, see Tuning Fork.

Application Library path: `Optimization_Module/Design_Optimization/tuning_fork_optimization`

Modeling Instructions

In this model version you determine the prong length by using an Optimization study node.

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

From the Application Libraries root, browse to the folder `COMSOL_Multiphysics/Structural_Mechanics` and double-click the file `tuning_fork.mph`.

ROOT

To keep the results of the parametric study, add a second study with an Eigenfrequency step set up the same way as before.

ADD STUDY

1 On 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 **Preset Studies>Eigenfrequency**.

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

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `1`.

5 Select the **Search for eigenfrequencies around** check box.

6 In the associated text field, type `440`.

Now, add optimization. The BOBYQA solver is generally the fastest of the derivative-free solvers when the objective function is smooth.

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for Optimization, locate the **Optimization Solver** section.

3 From the **Method** list, choose **BOBYQA**.

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

Expression	Description	Evaluate for
`(freq-440[Hz])^2`		eig

Next, add the control parameter. You can choose between the global parameters defined in your model. In this case, use the prong length.

5 Locate the **Control Variables and Parameters** section. Click **Add**.

Specify a length scale and suitable bounds.

6 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
`L`	`7.8[cm]`	`1[cm]`	`7[cm]`	`9[cm]`

The setup is now complete.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (solid) 1

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The default plot shows the eigenmode that corresponds to the optimized value of the cylinder length `L`.

To see the optimized value of the cylinder length, follow these steps:

Global Evaluation 2

On the **Results** toolbar, click **Global Evaluation**.

Derived Values

1 In the **Settings** window for Global Evaluation, locate the **Data** section.

2 From the **Data set** list, choose **Study 2/Solution 19 (sol19)**.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Solver>Control parameters>L - Cylinder length**.

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

Expression	Unit	Description
L	cm	Cylinder length

5 Click **New Table**.

TABLE

1 Go to the **Table** window.

The resulting cylinder length is close to 7.91 cm, which agrees with the value determined using a parametric sweep.

RESULTS

Mode Shape (solid) 1

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