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.
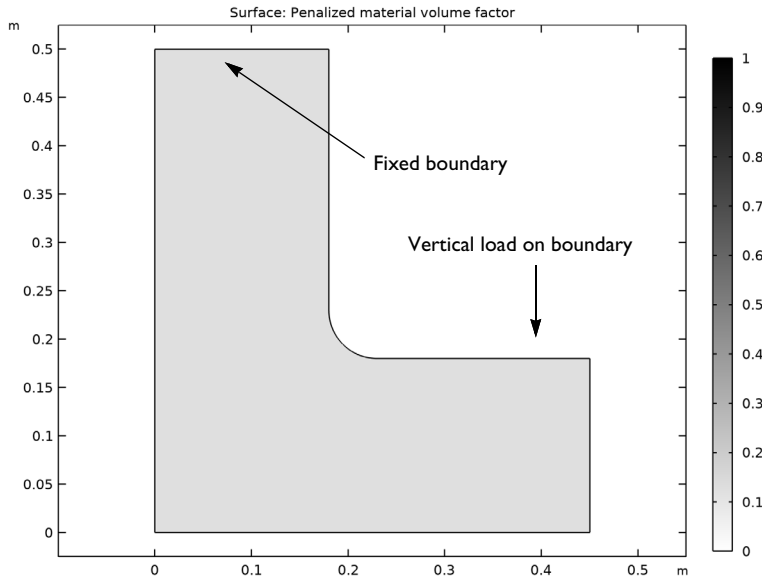

Figure 1: Geometry of the structure with loads and constraints.

This design is clearly overdesigned 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 that 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 material volume factor, θ_p, which is a function of the control variable, $0 \leq \theta_c \leq 1$:

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

Because the stiffness must for numerical reasons not vanish completely anywhere in the model, the penalized material volume factor, θ_p, is constrained such that $10^{-3} = \theta_{min} \leq \theta_p \leq 1$. The exponent $p \geq 1$ is a penalty factor that makes intermediate densities provide less stiffness compared to what they cost in weight. The material volume factor is related to the control variable via a Helmholtz filter and a projection function based on the hyperbolic tangent function:

$$\theta_f = R^2_{min}\nabla^2\theta_f + \theta_c$$
$$\theta = \frac{(\tanh(\beta(\theta_f - \theta_\beta)) + \tanh(\beta\theta_\beta))}{(\tanh(\beta(1 - \theta_\beta)) + \tanh(\beta\theta_\beta))}$$

Here θ_f, β, and θ_β are the filtered material volume factor, projection slope and projection point. The solution can be made independent of the mesh resolution by using a fixed value of the filter radius, R_{min}, but in this example we will not change the mesh and thus apply the default filter radius based on the local mesh element size.

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

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

the SIMP penalization forces θ_c toward 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

$$\theta_{avg} = \frac{1}{A}\int\limits_{\Omega} \theta(\mathbf{x})d\Omega \qquad (2)$$

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

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.
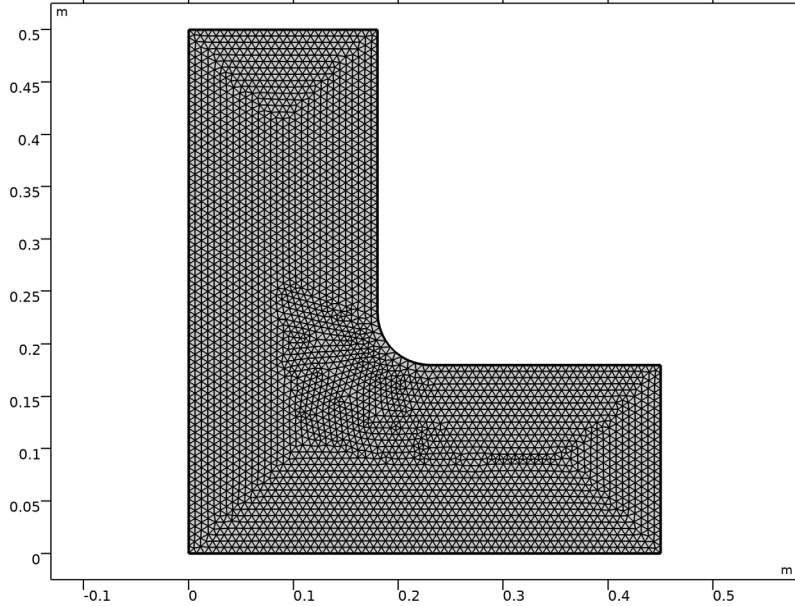

Figure 2: Finite element mesh of the structure.

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.
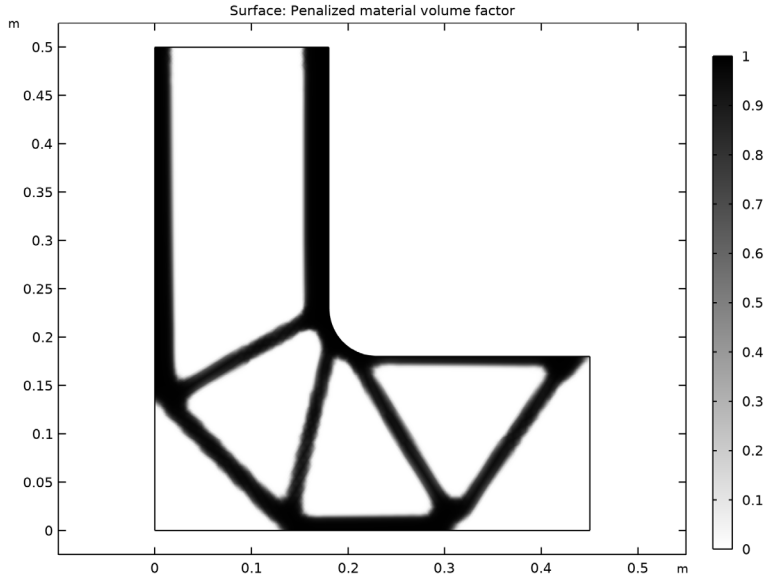

Figure 3: Distribution of the control variable after optimization.

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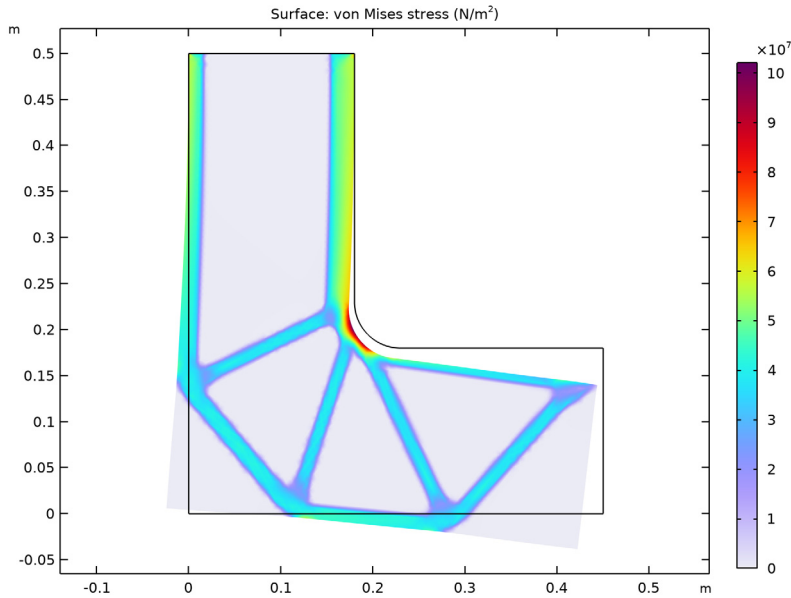
Figure 4: The von Mises stress of the optimal distribution of the control variable.

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 is combined with a **Density Model** to represent the structural properties of the bracket, while the objective and constraint are defined directly in a **Topology Optimization** study step. 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 the study step. For topology optimization, either the MMA, IPOPT, 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 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

| Name	Expression	Value	Description
F_load	10[kN]	10000 N	Applied load
h0	7.5[mm]	0.0075 m	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
beta	1	1	Projection slope

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

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

3 In the **Geometry** toolbar, click ▦ **Build All**.

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

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
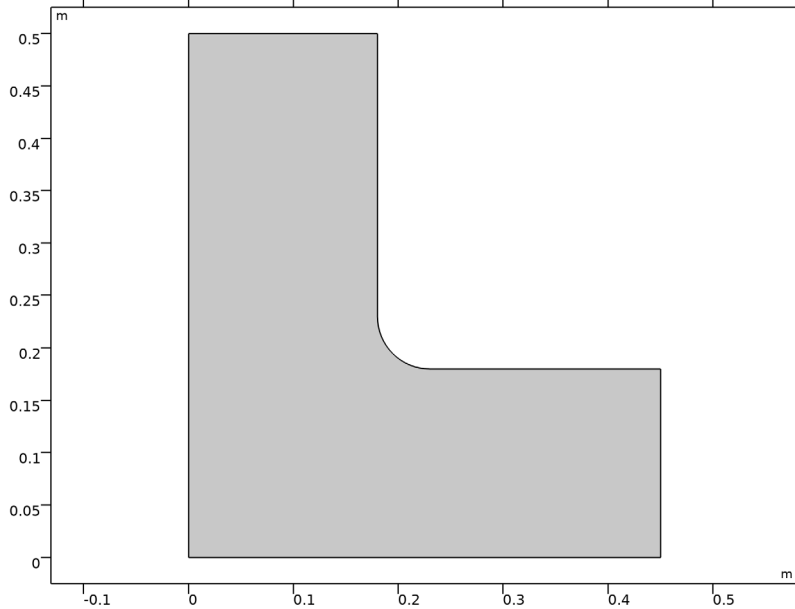
The geometry should now look like that in Figure 1. Note that the inserted geometry is parameterized and that the parameters used are automatically added to the list of global parameters in the model.

ADD MATERIAL

1 In the **Home** toolbar, click ![icon] **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Structural steel**.

4 Click **Add to Global Materials** in the window toolbar.

5 In the **Home** toolbar, click ![icon] **Add Material** to close the **Add Material** window.

MATERIALS

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

Add a **Density Model** feature, which can be used to distinguish between void and solid regions. This variable is coupled back to the **Solid Mechanics** interface via the Young's

modulus. The filter radius should not be smaller than the mesh element size, so the default will work, but a fixed value has to be chosen to make the result of the optimization mesh independent.

COMPONENT 1 (COMP1)

Density Model 1 (dtopo1)

1 In the **Physics** toolbar, click ⌖ **Optimization** and choose **Topology Optimization**.

2 Select Domain 1 only.

3 In the **Model Builder** window, click **Density Model 1 (dtopo1)**.

4 In the **Settings** window for **Density Model**, locate the **Projection** section.

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

6 In the β text field, type beta.

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

DEFINITIONS

Mass Properties 1 (mass1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Physics Utilities>Mass Properties**.

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

3 From the **Density source** list, choose **From physics interface**.

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.

MATERIALS

Topology Link 1 (toplnk1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Topology Link**.

2 In the **Settings** window for **Topology Link**, locate the **Link Settings** section.

3 From the **Topology source** list, choose **Density Model 1 (dtopo1)**.

4 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All domains**.

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.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundary 3 only.

Boundary Load 1

1 In 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 $\mathbf{F}_{tot}$ vector as

0	x
-F_load	y

MESH 1

Free Triangular 1

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type h0.

5 Click ⊞ **Build All**.

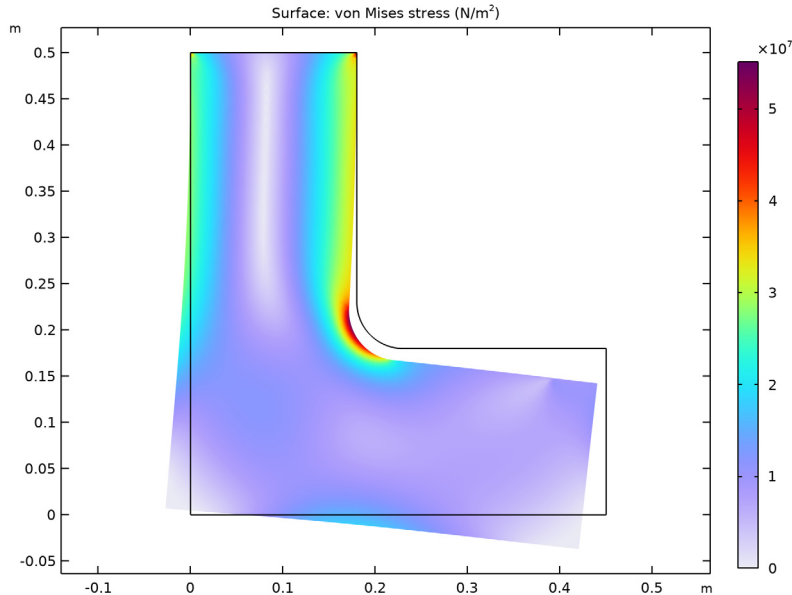The resulting mesh is shown in Figure 2.

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

Stress (solid)
Click the ⊕ **Zoom Extents** button in the **Graphics** toolbar.

Surface: von Mises stress (N/m²)

Penalized Material Volume Factor

1 In the **Model Builder** window, expand the **Results>Topology Optimization** node, then click **Output material volume factor**.

2 In the **Settings** window for **2D Plot Group**, type `Penalized Material Volume Factor` in the **Label** text field.

Surface 1
Plotting the Penalized Material Volume Factor gives a sharper and fairer picture of the topology, as seen by the **Solid Mechanics** interface.

1 In the **Model Builder** window, expand the **Penalized Material Volume Factor** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions> Density Model 1>Auxiliary variables>dtopo1.theta_p - Penalized material volume factor**.

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

4 Click ▮▮▮ **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

8 From the **Color table transformation** list, choose **Reverse**.

9 In the **Penalized Material Volume Factor** toolbar, click ⬛ **Plot**.

Global Evaluation 1

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

2 Right-click **Results>Derived Values** and choose **Global Evaluation**.

3 In the **Settings** window for **Global Evaluation**, click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Global>solid.Ws_tot - Total elastic strain energy - J**.

4 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

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
WsRef	1.3[J]	1.3 J	Reference strain energy

Set up probes for the objective functions and constraint.

DEFINITIONS

Global Variable Probe 1 (var1)

1 In 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)> Definitions>Density Model 1>Global>dtopo1.theta_avg - Average material volume factor**.

3 Locate the **Expression** section.

4 Select the **Description** check box. In the associated text field, type `Solid fraction`.

Global Variable Probe 2 (var2)

1 In 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)> Solid Mechanics>Global>solid.Ws_tot - Total elastic strain energy - J**.

3 Locate the **Expression** section. In the **Expression** text field, type `solid.Ws_tot/WsRef`.

4 Select the **Description** check box. In the associated text field, type `Strain energy factor`.

STUDY 1

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

Topology Optimization

1 In the **Study** toolbar, click 🎯 **Optimization** and choose **Topology Optimization**.

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

3 In the **Maximum number of iterations** text field, type 50.

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.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>Density Model 1>Global> comp1.dtopo1.theta_avg - Average material volume factor**.

5 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>comp1.solid.Ws_tot - Total elastic strain energy - J**.

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

Expression	Lower bound	Upper bound
comp1.solid.Ws_tot/WsRef		WsMaxFactor

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

8 From the **Plot group** list, choose **Penalized Material Volume Factor**.

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

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

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

Parametric Sweep

1 In the **Study** toolbar, click ⠿ **Parametric Sweep**.

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

3 Click ＋ **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
beta (Projection slope)	1 8	

5 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

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

7 In the **Study** toolbar, click ═ **Compute**.

RESULTS

Penalized Material Volume Factor

Click the ⟨+⟩ **Zoom Extents** button in 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).