Model created in COMSOL Multiphysics 6.4

Twisting and Bending of a Metal Frame

Introduction

A thin-walled frame member with a central cutout is subjected to bending and twisting. The stresses around the cutout are expected to be above the yield stress of the material.

This example demonstrates how to model plastic deformation in thin structures. The **Linear Elastic Material, Layered** in the **Shell** interface is used to define the load scenario, and the **Plasticity** subnode to quantify the plastic deformation around the cutout. The residual stress and plastic strains after removal of the load are investigated.

Model Definition

The load carrying beam is shown in Figure 1. It has a length of 1.1 m, a thin-walled square cross section with dimensions 160-by-160 mm^2 and a wall thickness of 6 mm. The cutout is centrally placed on one of the faces and is 100 mm long, 80 mm wide, and has a fillet with a radius of 10 mm in each corner.

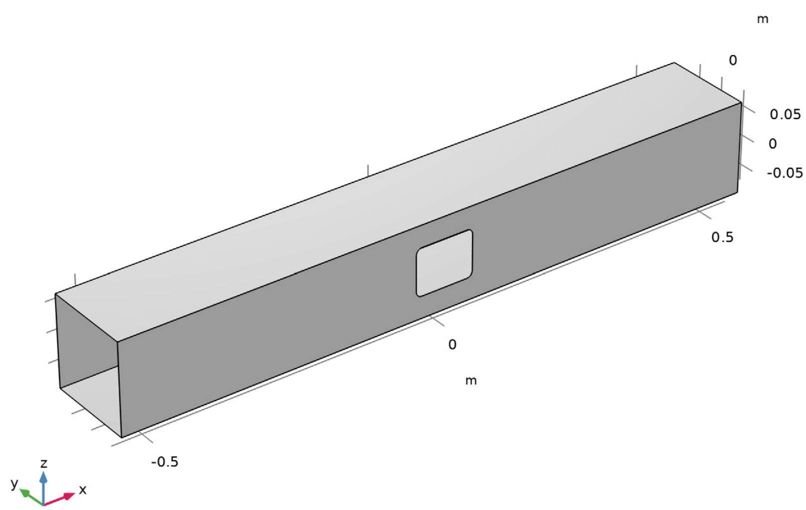

Figure 1: Geometry of the frame member.

The loading consists of a bending moments in the y direction, and a twisting moment around the x-axis. Both loads can vary independently and are applied at the right end of the frame at x = 1.1 m. The left end, x = 0, is clamped.

The frame is made out of steel with Young's modulus, Poisson's ratio, and density given as E = 200 GPa, ν = 0.3, and ρ = 7850 kg/m^3, respectively. Yielding of the steel is governed by a von Mises plasticity model with an initial yield stress σ_{ys0} = 355 MPa. A tangent modulus E_t = 100 MPa defines the isotropic hardening behavior.

Results and Discussion

Figure 2 shows the distribution of the von Mises stress at the peak of the applied loads. The effect of the cutout on the stress field is clearly visible. Due to the applied loads, stresses on the inner side of the frame are higher than on the outside.

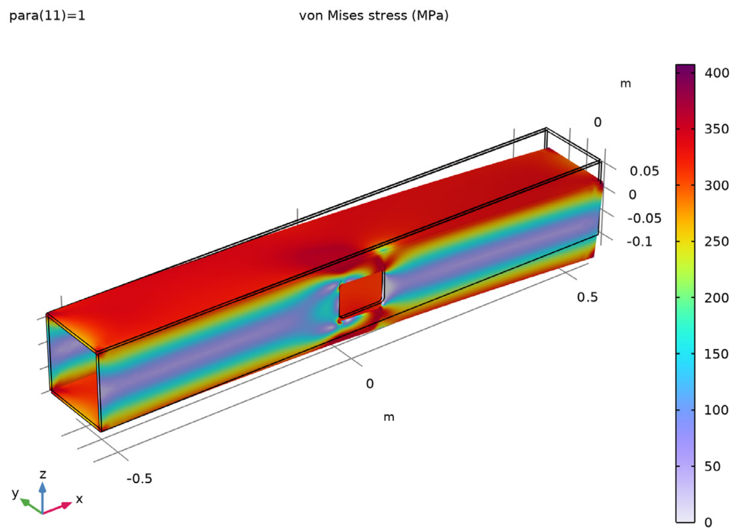

Figure 2: Distribution of von Mises stress in the frame at the maximum load.

Figure 3 shows the residual stress on the outer surface of the frame after removal of the applied loads. Large stresses are visible in the vicinity of the cutout.

It is possible to observe the development of plastic strains in the structure during loading. Figure 4 shows the parts of the frame where plastic deformations have occurred at the end of the loading cycle. Note that plastic strains are not uniform through the shell thickness.

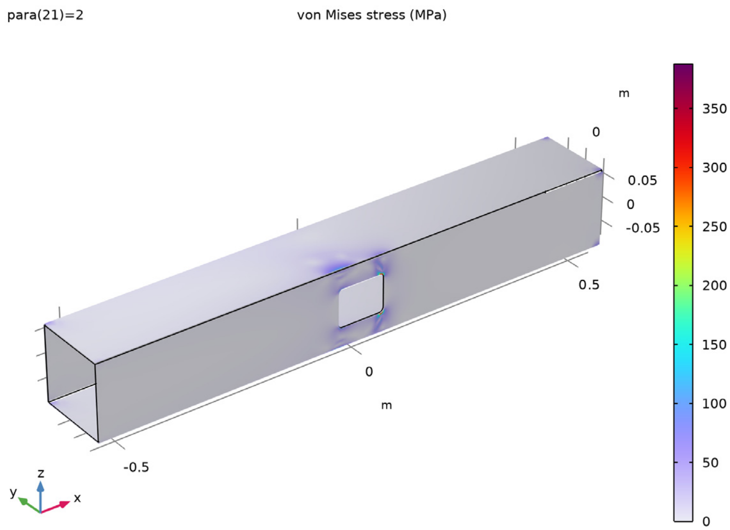

Figure 3: Residual stress on the outside of the frame after removal of the applied loads.

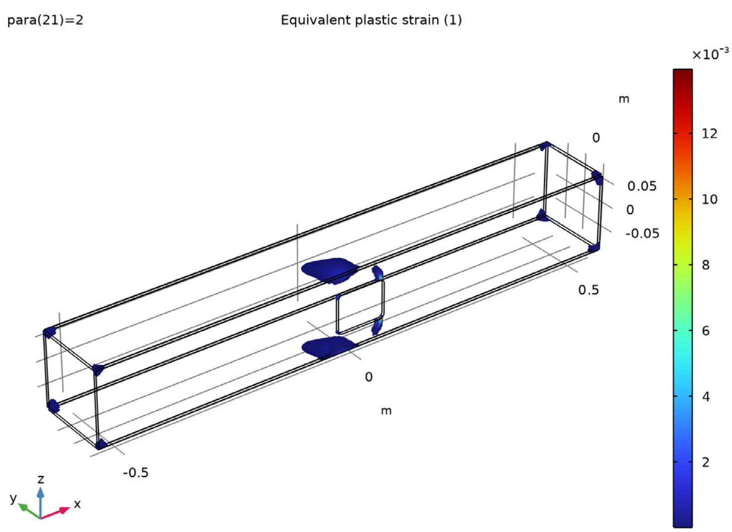

Figure 4: Equivalent plastic strain in the frame at end of the loading cycle.

Notes About the COMSOL Implementation

The structure is modeled with the **Shell** interface and the **Linear Elastic Material, Layered** model. This feature enables to model phenomena that are thickness dependent, such as **Plasticity**. The user interface is similar to what is available in the **Solid Mechanics** interface.

In COMSOL Multiphysics, several load types can be applied to a **Rigid Connector** node. In this example, the **Applied Moment** node is used twice to apply a twisting moment around the x-axis, and a bending moment in the y direction. A piecewise function is used to ramp up the magnitude of these moments, and to remove the loads to observe the residual stress and plastic strains.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/frame_with_cutout_plasticity`

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** > **Shell (shell)**.

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
para	0	0	Parameter

Piecewise 1 (pw1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global** > **Piecewise**.

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

3 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	1	x
1	2	2-x

4 Locate the **Units** section. In the **Arguments** text field, type 1.

5 In the **Function** text field, type N*m.

GEOMETRY 1

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Width** text field, type 1.1.

5 In the **Depth** text field, type 0.154.

6 In the **Height** text field, type 0.154.

7 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1 (wp1)

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

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

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **blk1**, select Boundary 3 only.

5 Click **Go to Plane Geometry**.

Work Plane 1 (wp1) > Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 From the **Base** list, choose **Center**.

4 Locate the **Size and Shape** section. In the **Width** text field, type `80[mm]`.

5 In the **Height** text field, type `100[mm]`.

Work Plane 1 (wp1) > Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.

2 On the object **r1**, select Points 1–4 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `10[mm]`.

Difference 1 (dif1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Booleans and Partitions** > **Difference**.

2 Select the object **blk1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **wp1** only.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **dif1**, select Boundaries 1 and 6 only.

3 In the **Settings** window for **Delete Entities**, click **Build All Objects**.

ADD MATERIAL

1 In the **Materials** 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 Right-click and choose **Add to Component 1 (comp1)**.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

SHELL (SHELL)

Linear Elastic Material, Layered 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Linear Elastic Material, Layered**.

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

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

Plasticity 1

In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

MATERIALS

Structural steel (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Structural steel (mat1)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	6[mm]	m	Shell
Initial yield stress	sigmags	355[MPa]	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	100[MPa]	Pa	Elastoplastic material model

SHELL (SHELL)

Prescribed Displacement/Rotation 1

1. In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.
2. Select Edges 1, 2, 4, and 6 only.
3. In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. From the **Displacement in y direction** list, choose **Prescribed**.
6. From the **Displacement in z direction** list, choose **Prescribed**.

   Apply parametric twisting and bending moments at the other end using a rigid connector.

Rigid Connector 1

1. In the **Physics** toolbar, click **Edges** and choose **Rigid Connector**.
2. Select Edges 17–20 only.

Twisting Moment (x)

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Moment**.

2 In the **Settings** window for **Applied Moment**, type `Twisting Moment (x)` in the **Label** text field.

3 Locate the **Applied Moment** section. Specify the **M** vector as

`9000*pw1(para)`	x
`0`	y
`0`	z

Bending Moment (y)

1 Right-click **Twisting Moment (x)** and choose **Duplicate**.

2 In the **Settings** window for **Applied Moment**, type `Bending Moment (y)` in the **Label** text field.

3 Locate the **Applied Moment** section. Specify the **M** vector as

`0`	x
`60000*pw1(para)`	y
`0`	z

Create a custom mesh in order to increase the mesh density around the cutout.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **More Generators** and choose **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 `0.022`.

5 In the **Minimum element size** text field, type `2.2E-4`.

6 In the **Maximum element growth rate** text field, type `1.2`.

7 In the **Curvature factor** text field, type `0.2`.

8 In the **Resolution of narrow regions** text field, type `1`.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

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

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

Size Expression 1

1 Right-click **Free Triangular 1** and choose **Size Expression**.

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

3 In the **Size expression** text field, type `if((abs(X)>0.12)||(Y>0),0.022,0.008)`.

4 In the **Number of cells per dimension** text field, type `50`.

5 Click **Build All**.

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 **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.1,2)`	

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

Stress (shell)

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (para)** list, choose **1**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

2 In the **Stress (shell)** toolbar, click **Plot**.

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

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Stress, Slice (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Layered Material Slice 1

In the **Model Builder** window, expand the **Stress, Slice (shell)** node.

Deformation

1 In the **Model Builder** window, expand the **Layered Material Slice 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type 10.

4 In the **Stress, Slice (shell)** toolbar, click **Plot**.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Equivalent Plastic Strain (shell)**.

3 Click the **Add Result Template** button in the window toolbar.

4 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (shell)** node, then click **Surface 1**.

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

3 From the **Color table** list, choose **Rainbow**.

4 From the **Color table type** list, choose **Continuous**.

Filter 1

1 Right-click **Surface 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 `shell.epeGp>1E-5`.

4 In the **Equivalent Plastic Strain (shell)** toolbar, click **Plot**.