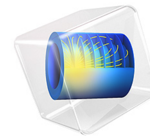
Created in COMSOL Multiphysics 6.0



Pressurized Orthotropic Container

Introduction

A container made of rolled steel is subjected to an internal overpressure. As an effect of the manufacturing method, one of the three material principal directions — the out-of- plane direction — has a higher yield stress than the other two. Hill's orthotropic plasticity is used to model the differences in yield strength. The model also shows how to define and use curvilinear coordinates aligned with the principal directions of the material, which in this case follow the contours of the container.

Model Definition

The container has the shape of a cylinder capped by two torispherical heads (also called Klöpper head). The cylinder has an internal radius of $R_i = 24$ cm, a height of $h = 80$ cm, and its thickness is $t = 2$ cm. The torispherical head is made out of three parts: the crown, the knuckle, and the flange. The crown has an internal radius of $R_c = 43.2$ cm, the knuckle has an internal radius of $R_k = 5.2$ cm, and the straight flange is $s = 7$ cm in height, see Figure 1.

Because of 2D axial symmetry and reflection symmetry, it is sufficient to model a quarter of the container, see Figure 1. The red lines define the rotation symmetry axis and the reflection symmetry axis.

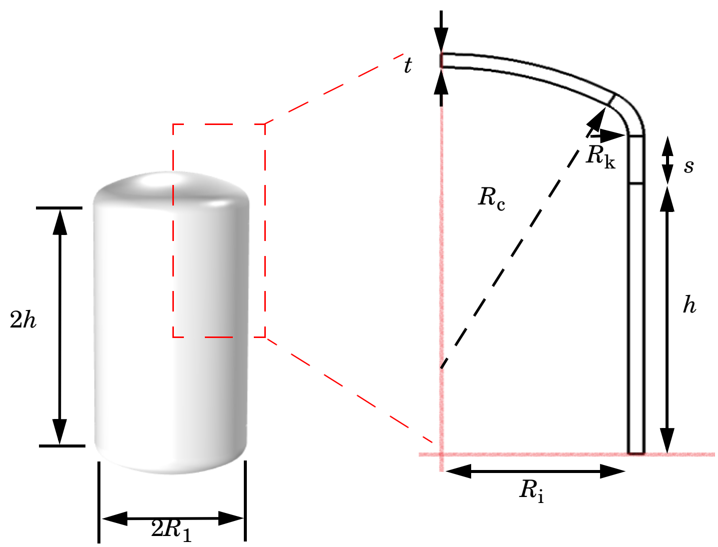


Figure 1: Schematic description of the container geometry and dimensions.

MATERIAL MODEL

The elastoplastic material is defined by a Young's modulus, E = 205 GPa and a Poisson's ratio, ν = 0.28. Hill's orthotropic plasticity governs the yielding, with the yield stress components given by

$$\begin{bmatrix} \sigma_{ys1} \\ \sigma_{ys2} \\ \sigma_{ys3} \\ \tau_{ys23} \\ \tau_{ys31} \\ \tau_{ys12} \end{bmatrix} = \begin{bmatrix} 381 \\ 381 \\ 450 \\ 240 \\ 240 \\ 220 \end{bmatrix} \text{MPa}$$

There is no hardening, so the material is perfectly plastic. The numbers in the subscripts denote the principal material directions, as indicated in the following section.

MATERIAL ORIENTATION

The rolled steel sheet has better mechanical properties in the out-of-plane direction, direction 3. To account for this anisotropy, use a special coordinate system that follows the shape of the component, see Figure 2.

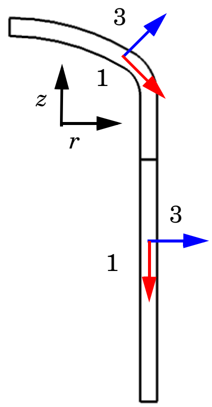


Figure 2: Orientation of the local material coordinate system. The second principal direction is oriented in the circumferential direction, perpendicular to the rz-plane.

The Curvilinear Coordinates interface is used to compute an orthogonal system of coordinates that follow the shape of the container.

Results and Discussion

An approximate analytical solution can be obtained for the cylindrical part of the container. The principal stresses in the center of the container can be estimated from the internal radius R_i, the wall thickness t, and the internal pressure p:

$$\begin{aligned}\sigma_1 &= p\frac{R_i}{2t}\\ \sigma_2 &= p\frac{R_i}{t}\\ \sigma_3 &= -p\end{aligned} \tag{1}$$

Following Hill's criterion, yielding occurs when

$$F(\sigma_2-\sigma_3)^2+G(\sigma_3-\sigma_1)^2+H(\sigma_1-\sigma_2)^2 = 1$$

or replacing by the expressions in Equation 1

$$p^2\left[F\left(\frac{R_i}{t}+1\right)^2+G\left(1+\frac{R_i}{2t}\right)^2+H\left(\frac{R_i}{2t}-\frac{R_i}{t}\right)^2\right] = 1$$

The material parameters, $F = G = 2.47\cdot10^{-18}\ 1/\mathrm{Pa}^2$ and $H = 4.42\cdot10^{-18}\ 1/\mathrm{Pa}^2$, give the analytical onset of yielding in the center of the cylinder at $p = 37.8$ MPa. Given the curvature of the knuckle, the material in the torispherical head undergoes plastic deformation below this onset pressure.

Figure 3 shows the von Mises stress at 10% yielded volume, which happens when the inner pressure reaches 31.4 MPa. For isotropic steel with a yield stress of 381 MPa, the 10% yielded volume is reached when $p = 29$ MPa. Therefore, with orthotropic steel, the pressure needed to reach 10% yielded volume is about 8% higher than when using isotropic steel. The extent of plastic strains is shown also shown in Figure 4

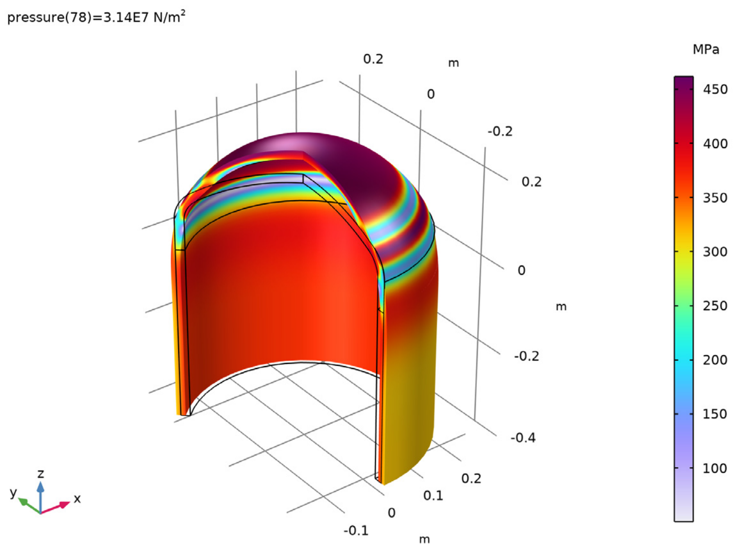


Figure 3: Distribution of von Mises stress at 10% yielded volume.

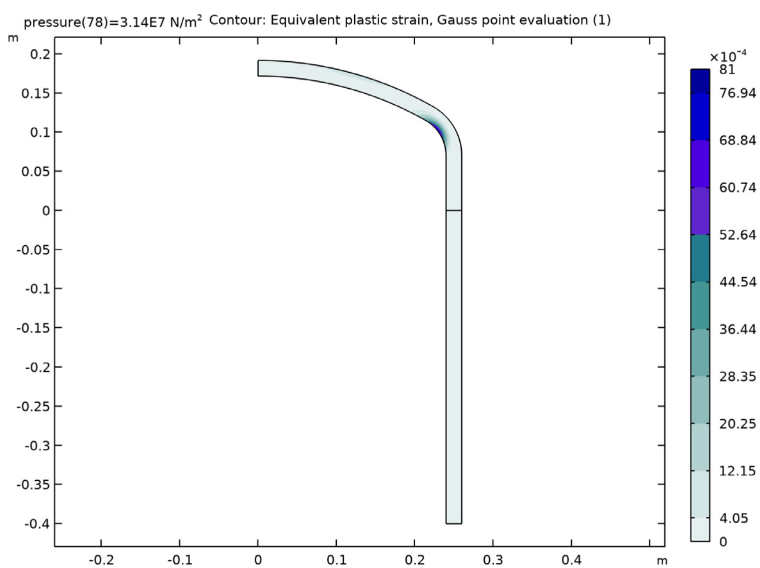


Figure 4: Equivalent plastic strain at 10% yielded volume

Notes About the COMSOL Implementation

Hill orthotropic plasticity is available in COMSOL as a built-in option in the **Plasticity** node, where either Hill's coefficients or initial yield stresses can be given. The yield strength values can also be specified in the material node.

A coordinate system that follows the geometrical shape is created by the Curvilinear Coordinates interface. For axisymmetric geometries, the new base vectors, x_1 and x_3, are computed for the geometry. In a case of geometric nonlinearity, the coordinates R and Z define the positions with respect to the initial configuration (*material frame*) whereas r and z define the positions with respect to the deformed configuration (*spatial frame*). Generally, material properties are defined in terms of the initial configuration. To assign the new base vector system to the component, select **Curvilinear System** from the **Coordinate system** list in the **Settings** window for **Linear Elastic Material**.

Figure 5 visualizes the base vector system defined by the curvilinear system using an arrow surface plot. The red arrows denote direction 1, while the blue arrows denote direction 3. The out-of-plane direction is used as direction 2 (not plotted). The normal of the inner surface is oriented to the interior of the container, which enables to directly apply the inner pressure load, see Figure 6,

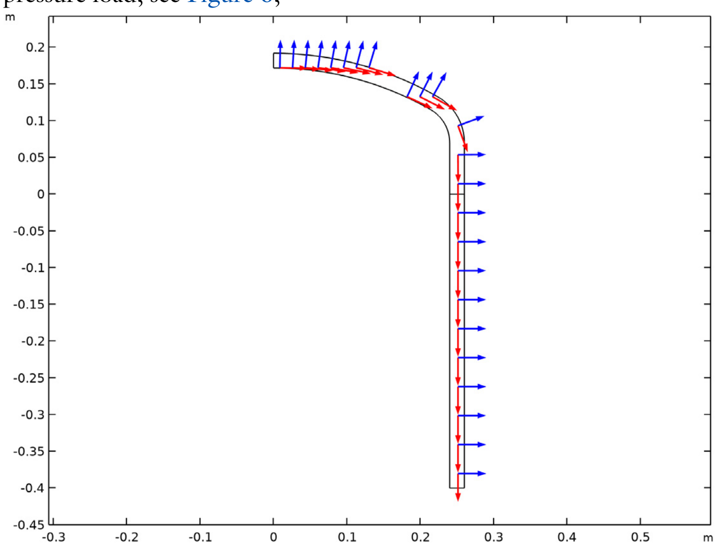


Figure 5: Orientation of the curvilinear system.

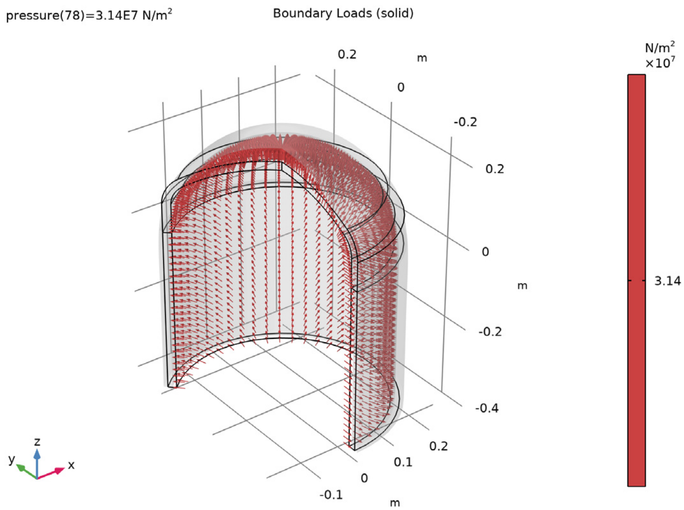


Figure 6: Orientation and amplitude of the applied pressure load.

A stop condition is added to the parametric solver, so that the simulation stops when 10% of the material has exceeded the yield limit. Unless you are performing a failure analysis, it is not necessary to compute the whole plastic history, and the stop condition saves much computation time from being spent in the strongly nonlinear regime.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/orthotropic_container`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters from the appended file.

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 Click **Load from File**.

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

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 Locate the **Advanced** section. From the **Frame** list, choose **Material (R, PHI, Z)**.

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
y_vol	intop1(solid.epeGp>0)/ intop1(1)		Yielded volume fraction

GEOMETRY 1

Circle 1 (c1)

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

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

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

4 In the **Sector angle** text field, type `alpha`.

5 Locate the **Position** section. In the **z** text field, type `sf-(Rc-hi)`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90-alpha`.

Circle 2 (c2)

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

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

3 In the **Radius** text field, type `Rc+th`.

4 In the **Sector angle** text field, type `alpha`.

5 Locate the **Position** section. In the **z** text field, type `sf-(Rc-hi)`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90-alpha`.

Crown

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, type `Crown` in the **Label** text field.

3 Select the object **c2** only.

4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **c1** only.

Circle 3 (c3)

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

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

3 In the **Radius** text field, type `Rk+th`.

4 In the **Sector angle** text field, type `90-alpha`.

5 Locate the **Position** section. In the **r** text field, type `Ri-Rk`.

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

7 Click **Build Selected**.

Circle 4 (c4)

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

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

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

4 In the **Sector angle** text field, type `90-alpha`.

5 Locate the **Position** section. In the **r** text field, type `Ri-Rk`.

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

7 Click **Build Selected**.

Knuckle

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c3** only.

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

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

5 Select the object **c4** only.

6 Click **Build Selected**.

7 In the **Label** text field, type `Knuckle`.

Flange

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

2 In the **Settings** window for **Rectangle**, type `Flange` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `th`.

4 In the **Height** text field, type `sf`.

5 Locate the **Position** section. In the **r** text field, type `Ri`.

6 Click **Build All Objects**.

Klopper Head

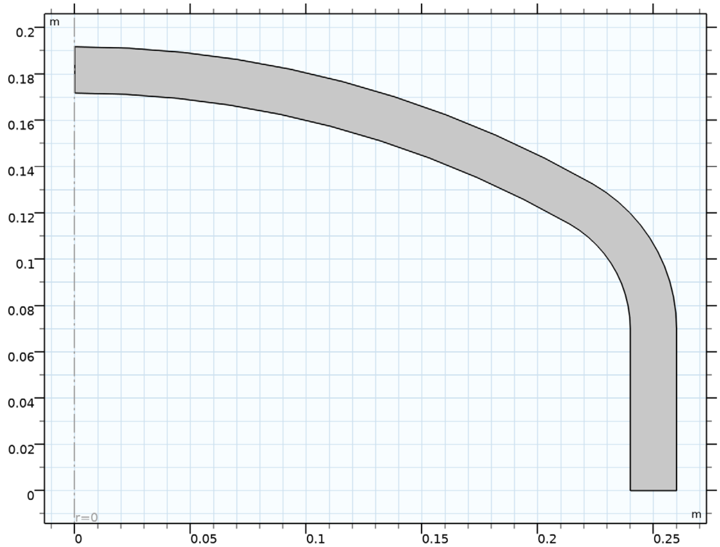
1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Klopper Head` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

5 Click **Build All Objects**.



Cylinder

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

2 In the **Settings** window for **Rectangle**, type `Cylinder` in the **Label** text field.

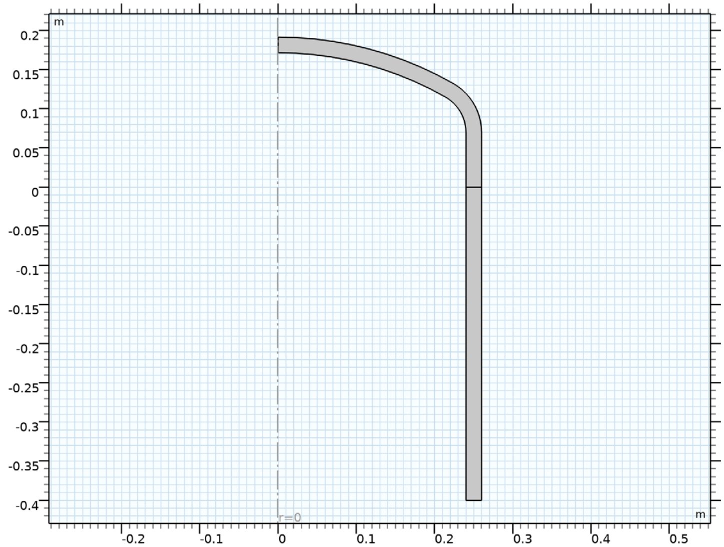
3 Locate the **Size and Shape** section. In the **Width** text field, type `th`.

4 In the **Height** text field, type `hcyl`.

5 Locate the **Position** section. In the **r** text field, type `Ri`.

6 In the **z** text field, type `-hcyl`.

7 Click **Build All Objects**.



CURVILINEAR COORDINATES (CC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Curvilinear Coordinates (cc)**.

2 In the **Settings** window for **Curvilinear Coordinates**, locate the **Settings** section.

3 Select the **Create base vector system** check box.

Diffusion Method 1

In the **Physics** toolbar, click **Domains** and choose **Diffusion Method**.

Inlet 1

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

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

3 Click **Clear Selection**.

4 Select Boundary 1 only.

Diffusion Method 1

In the **Model Builder** window, click **Diffusion Method 1**.

Outlet 1

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

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

3 Click **Clear Selection**.

4 Select Boundary 3 only.

SOLID MECHANICS (SOLID)

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

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

2 Select Boundary 3 only.

Boundary Load 1

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

2 Select Boundaries 2, 4, 8, and 10 only.

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

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

5 In the p text field, type `pressure`.

Linear Elastic Material 1

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

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

3 From the **Coordinate system** list, choose **Curvilinear System (cc) (cc_cs)**.

Plasticity 1

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

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 From the **Yield function F** list, choose **Hill orthotropic plasticity**.

4 Find the **Isotropic hardening model** subsection. From the list, choose **Perfectly plastic**.

ADD MATERIAL

1 In 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>Steel AISI 4340**.

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

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

MATERIALS

Steel AISI 4340 (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Initial tensile and shear yield stresses	{ys1, ys2, ys3, ys4, ys5, ys6}	`{381e6,381e6,450e6,240e6,240e6,220e6}`	N/m²	Elastoplastic material model

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 4, 7, 10, and 11 only.

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

4 In the **Number of elements** text field, type `10`.

Distribution 3

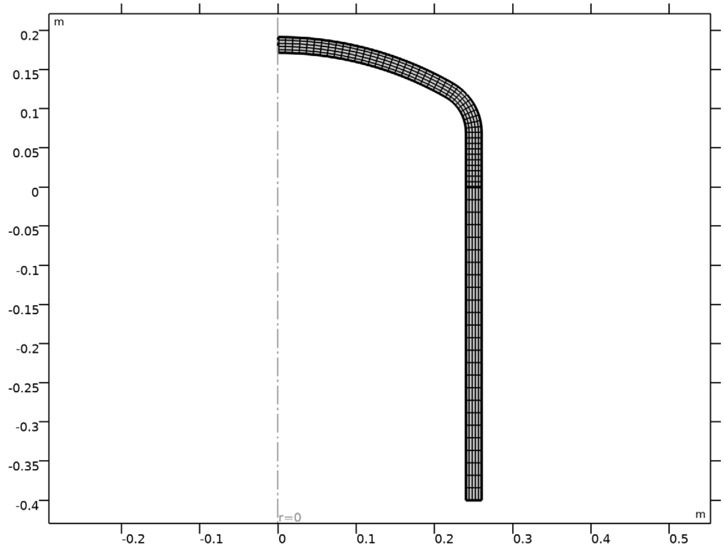
1 Right-click **Mapped 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `25`.

4 Select Boundaries 2, 6, 8, and 9 only.

5 Click **Build All**.



The mesh should consist of 350 quadrilateral elements, with 5 elements through the thickness. Finer elements are created at the knuckle since stress gradients are expected there.

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**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Stationary 2

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Curvilinear Coordinates (cc)**.

Set up an auxiliary continuation sweep for the **pressure** parameter.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	**Parameter value list**
pressure (Internal pressure)	`range(16e6, 2e5, 36e6)`

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Introduce a stop condition to stop the solver when a certain amount of material has yielded.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 2** node.

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2> Parametric 1** and choose **Stop Condition**.

5 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

6 Click **Add**.

7 In the table, enter the following settings:

Stop expression	**Stop if**	**Active**	**Description**
`0.1-comp1.y_vol`	Negative (<0)	√	`Stop expression 1`

Specify that the solution is to be stored both before and after the stop condition is reached.

8 Locate the **Output at Stop** section. From the **Add solution** list, choose **Steps before and after stop**.

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

RESULTS

Material Principal Direction

1 In the **Model Builder** window, under **Results** click **Coordinate system (cc)**.

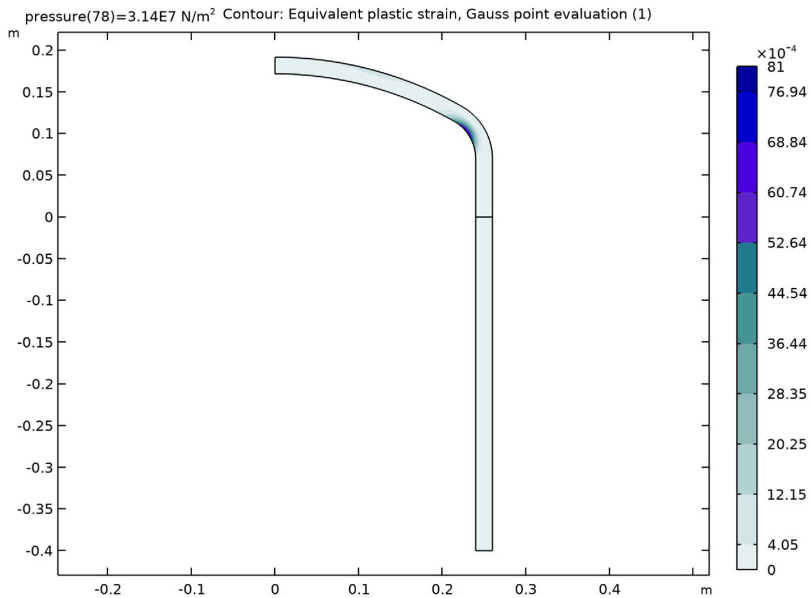
2 In the **Settings** window for **2D Plot Group**, type `Material Principal Direction` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Equivalent Plastic Strain (solid)

1 In the **Model Builder** window, click **Equivalent Plastic Strain (solid)**.

2 In the **Equivalent Plastic Strain (solid)** toolbar, click **Plot**.



The onset of plasticity can be investigated by evaluating the volume of the material which has exceeded the yield stress.

Yielded Volume

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

2 In the **Settings** window for **1D Plot Group**, type `Yielded Volume` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

4 Find the **Type and data** subsection. Clear the **Type** check box.

5 Clear the **Unit** check box.

Global 1

1 Right-click **Yielded Volume** and choose **Global**.

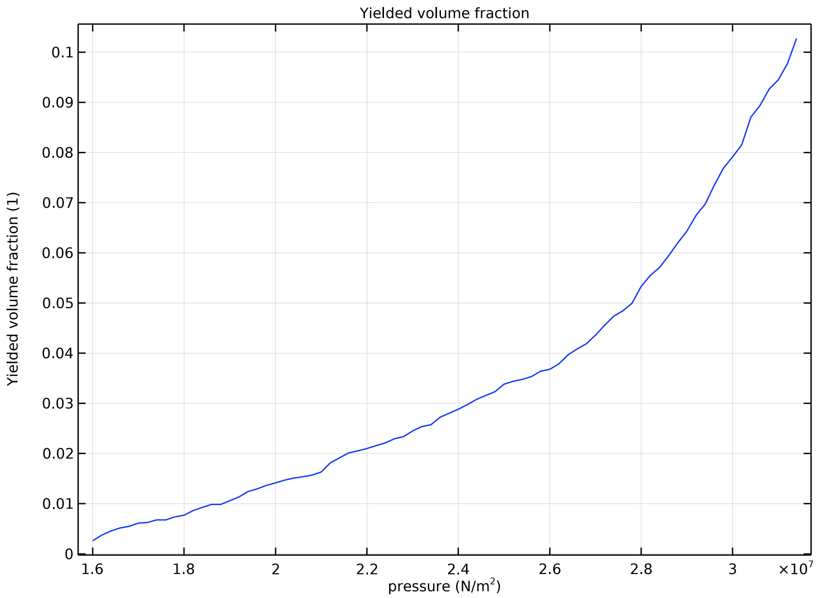
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`y_vol`	1	`Yielded volume fraction`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 In the **Yielded Volume** toolbar, click **Plot**.



Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

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

3 Select the **Show units** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress, 3D (solid)** node, then click **Surface 1**.

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

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

4 In the **Stress, 3D (solid)** toolbar, click **Plot**.