Pressurized Orthotropic Container — Shell Version

Introduction

A container made of rolled steel is subjected to an internal overpressure. As an effect of the manufacturing method of the steel sheet, 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.

This example demonstrates how to perform the analysis of the container using the **Shell** interface and the **Linear Elastic Material, Layered** model.

Model Definition

The metal 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

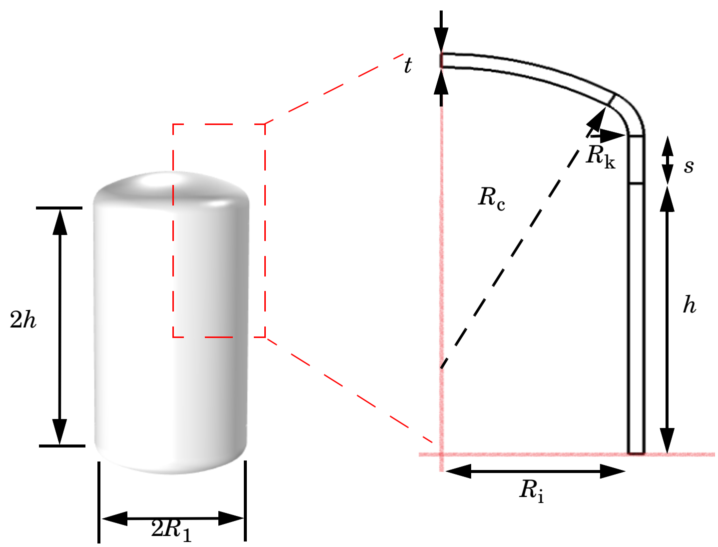


Figure 1: Schematic description of the container geometry and dimensions.

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.

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

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

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 component shape has to be used; see Figure 2. When using the Shell interface, this coordinate system coincides with the local coordinates of the shell.

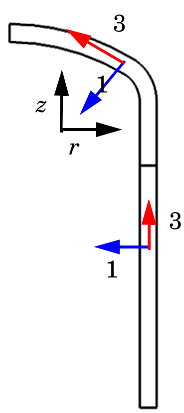


Figure 2: Orientation of local material coordinate system. The second principal direction is oriented in the circumferential (out-of-plane) direction, perpendicular to the rz-plane.

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 internal pressure p:

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

Following Hill's criterion, the 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

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

The material parameters, $F = G = 2.47\cdot 10^{-18}\ 1/\mathrm{Pa}^2$ and $H = 4.42\cdot 10^{-18}\ 1/\mathrm{Pa}^2$, 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 30.4 MPa. For isotropic steel with yield stress of 381 MPa, the 10% yielded volume is reached when $p = 28.4$ MPa. Therefore, with orthotropic steel, the pressure needed to reach 10% yielded volume is about 7% higher than when using an isotropic steel sheet. The extent of plastic strains is shown in Figure 4.

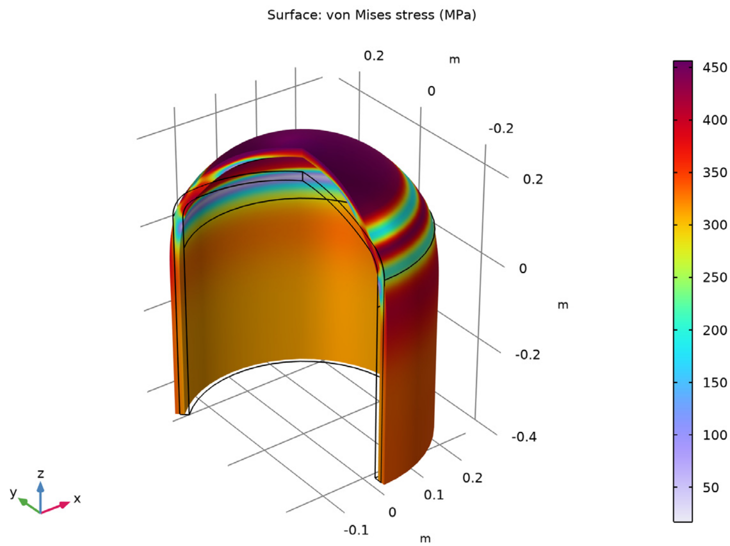


Figure 3: Distribution of von Mises stress at 10% yielded volume.

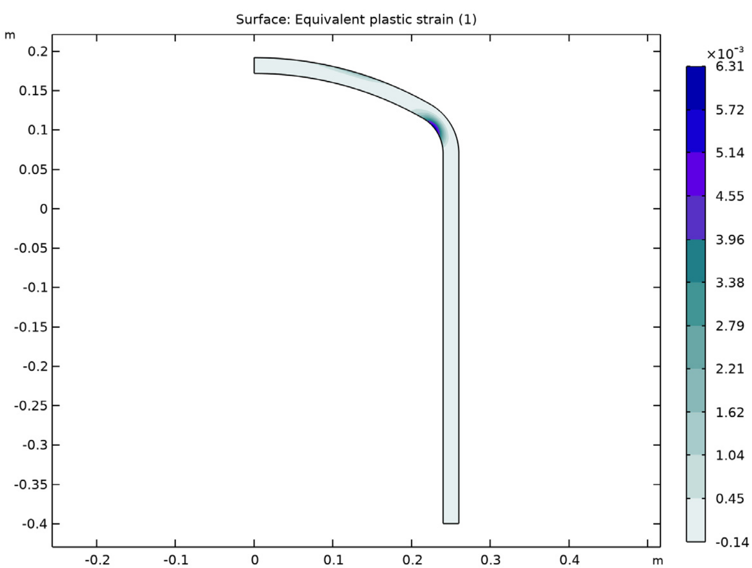


Figure 4: Equivalent plastic strain at 10% yielded volume.

Notes About the COMSOL Implementation

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.

Hill orthotropic plasticity is available in COMSOL as a built-in option under 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.

Figure 5 visualizes the boundary system and the thickness of the shell. 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 is oriented to the interior of the container, which enables to directly apply the inner pressure load, see Figure 6.

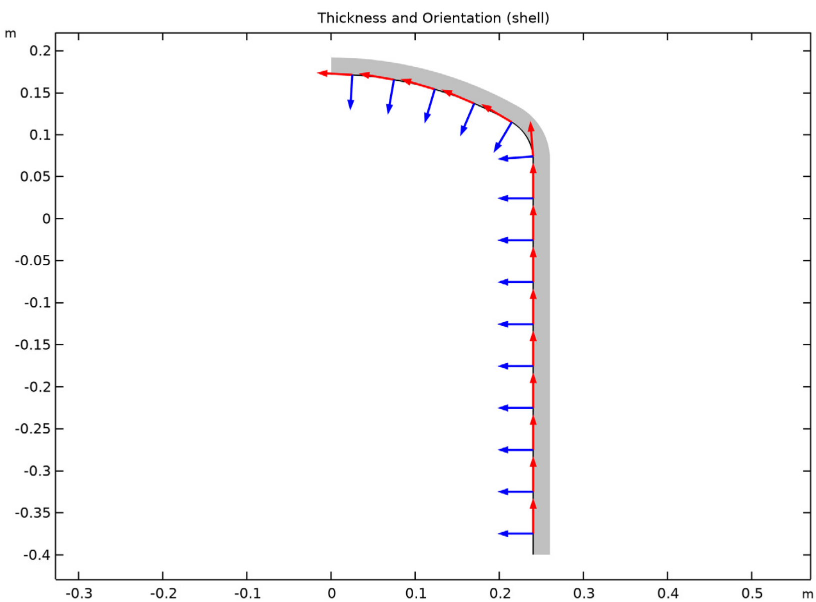


Figure 5: Thickness of the shell and orientation of the boundary system.

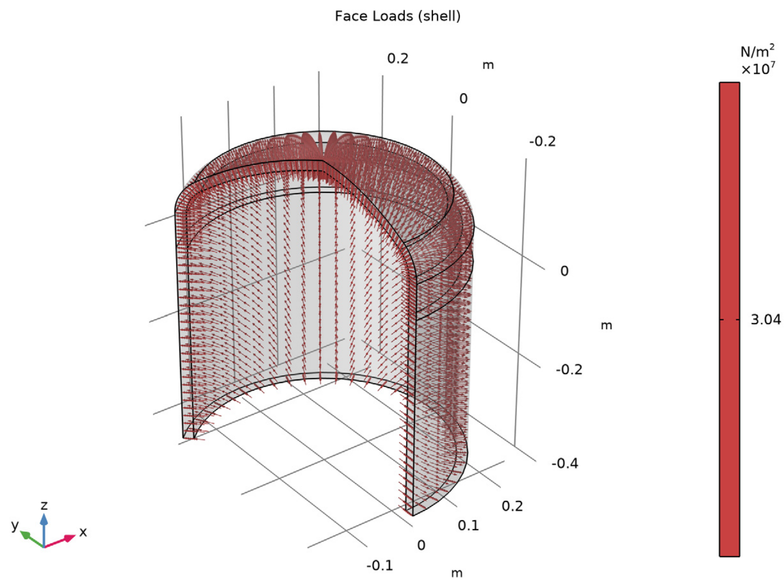


Figure 6: Orientation and amplitude of the applied pressure load.

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_shell`

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 **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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `orthotropic_container_parameters.txt`.

GEOMETRY 1

Crown

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, type `Crown` in the **Label** text field.
3 Locate the **Center** section. In the **z** text field, type `sf-(Rc-hi)`.
4 Locate the **Radius** section. In the **Radius** text field, type `Rc`.
5 Locate the **Angles** section. In the **Start angle** text field, type `90-alpha`.

Knuckle

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.
2 In the **Settings** window for **Circular Arc**, type `Knuckle` in the **Label** text field.
3 Locate the **Center** section. In the **r** text field, type `Ri-Rk`.
4 In the **z** text field, type `sf`.
5 Locate the **Radius** section. In the **Radius** text field, type `Rk`.
6 Locate the **Angles** section. In the **End angle** text field, type `90-alpha`.

Flange

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 Locate the **Starting Point** section. From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `Ri`.

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

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **r** text field, type `Ri`.

Cylinder

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 Locate the **Starting Point** section. From the **Specify** list, choose **Coordinates**.

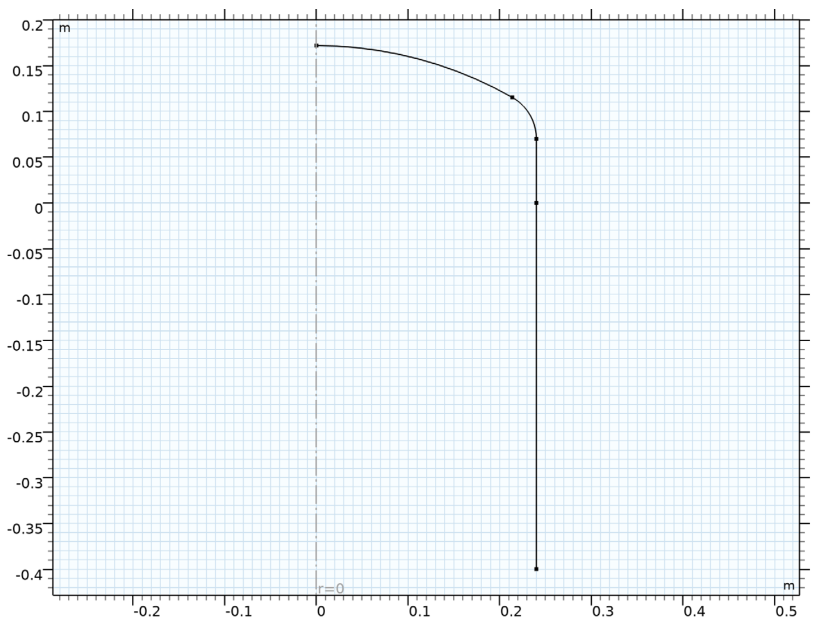
4 In the **r** text field, type `Ri`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **r** text field, type `Ri`.

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

8 Click **Build All Objects**.



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 **Geometric entity level** list, choose **Boundary**.

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

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

Variables 1

1 In the **Definitions** toolbar, click a= **Local 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(shell.llem1.xdintopall(shell.epeGp>0))/intop1(shell.llem1.xdintopall(1))`		`Yielded volume fraction`

The `shell.llem1.xdintopall` operator integrates the argument quantity across the thickness.

Set the boundary system to have the first tangent vector oriented upward and the normal oriented to the interior as shown on Figure 5. The latter will make it easier to apply the internal pressure.

Boundary System 1 (sys1)

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.

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

3 Find the **Coordinate names** subsection. From the **Axis** list, choose **z**.

4 Select the **Reverse normal direction** check box.

SHELL (SHELL)

Linear Elastic Material, Layered 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell (shell)** and choose **Material Models>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

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 σ_e list, choose **Hill orthotropic**.

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

Symmetry Plane 1

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

2 Select Point 3 only.

Face Load 1

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

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

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

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

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

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:

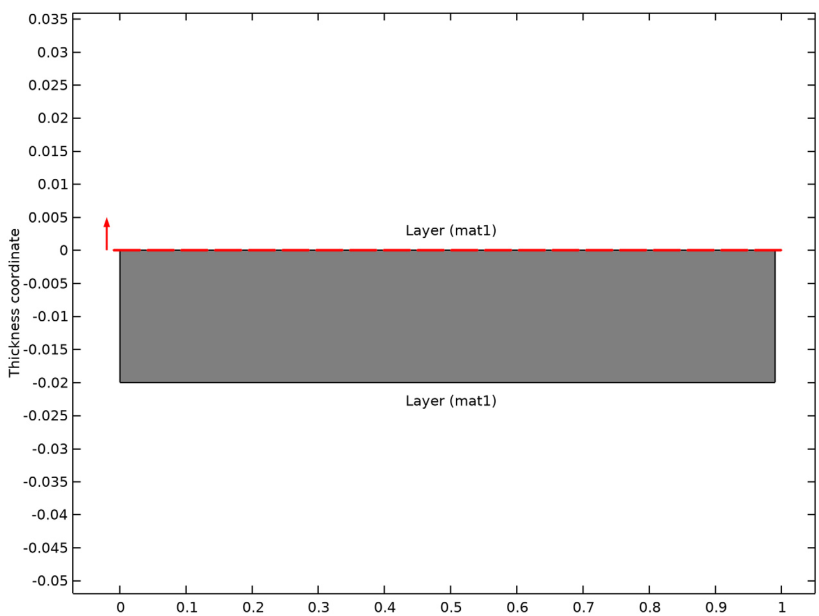
Property	Variable	Value	Unit	Property group
Thickness	lth	`th`	m	Shell
Initial tensile and shear yield stresses	{ys1, ys2, ys3, ys4, ys5, ys6}	`{381e6,381e6,450e6,240e6,240e6,220e6}`	N/m²	Elastoplastic material model
Mesh elements	lne	5	1	Shell

Five elements are set across the thickness to have same discretization as in the solid version of the model.

Since the geometry represents the inner side of the container and the normal is oriented to the interior, the geometry should be the top face of the shell layer.

3 Locate the **Orientation and Position** section. From the **Position** list, choose **Top side on boundary**.

4 Click **Layer Cross-Section Preview** in the upper-right corner of the **Orientation and Position** section.



MESH 1

Distribution 1

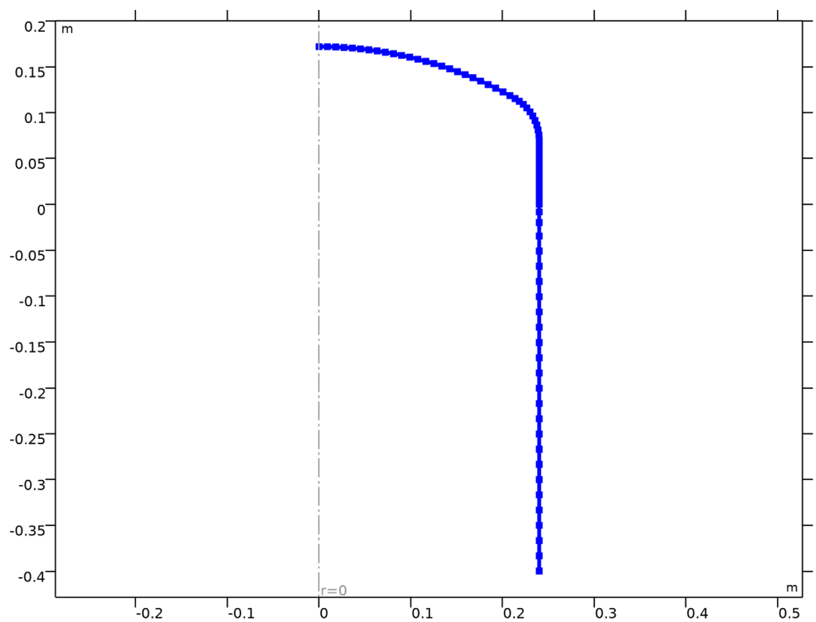
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Distribution**.
2 Select Boundaries 2 and 4 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.

Distribution 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Distribution**.
2 Select Boundaries 1 and 3 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `25`.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 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** check box.

4 Click **Add**.

5 In the table, click to select the cell at row number 1 and column number 3.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
pressure (Internal pressure)	`range(16e6,2e5,36e6)`	`N/m^2`

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

Solution 1 (sol1)

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

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

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

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1> 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 Clear the **Add warning** check box.

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

RESULTS

Surface 1

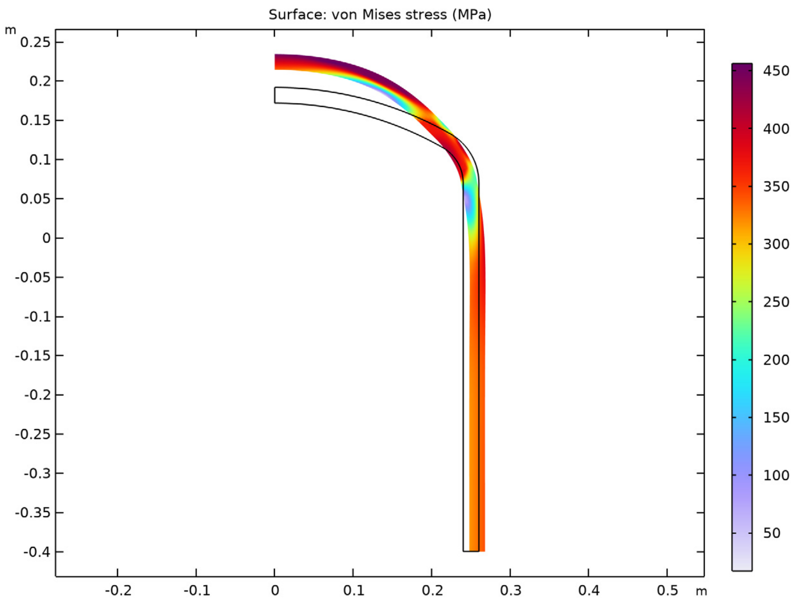
1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

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

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

Stress (shell)

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



Surface 1

1 In the **Model Builder** window, expand the **Results>Stress, 3D (shell)** 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 (shell)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

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

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

5 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Stress, Through Thickness (shell)**.

6 Click **Add Plot** in the window toolbar.

7 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Thickness and Orientation (shell)**.

8 Click **Add Plot** in the window toolbar.

9 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Applied Loads (shell)**.

10 Click **Add Plot** in the window toolbar.

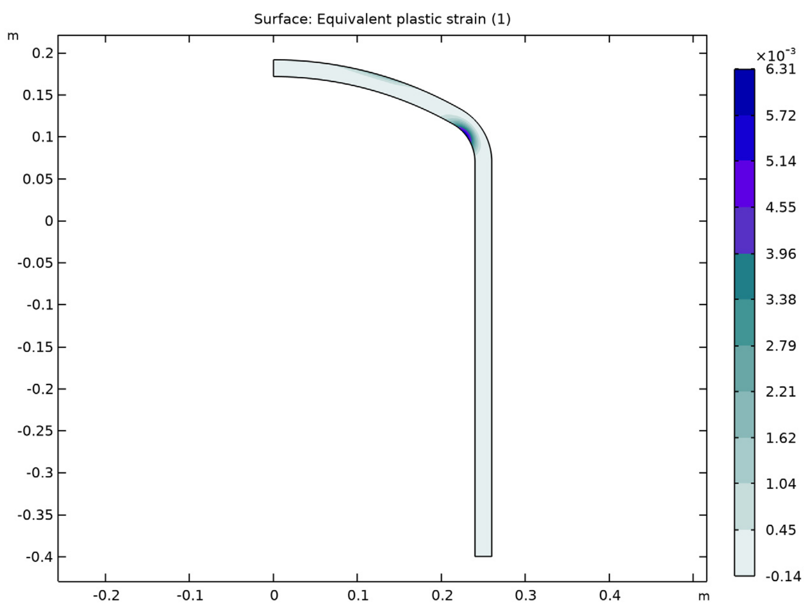
11 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Equivalent Plastic Strain (shell)

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (shell)**.

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



Through Thickness 1

1 In the **Model Builder** window, expand the **Stress, Through Thickness (shell)** node, then click **Through Thickness 1**.
2 In the **Settings** window for **Through Thickness**, locate the **Selection** section.
3 Click **Clear Selection**.
4 Select Point 2 only.
5 Locate the **x-Axis Data** section. From the **Unit** list, choose **MPa**.
6 In the **Stress, Through Thickness (shell)** toolbar, click **Plot**.

Improve the orientation plot group to show the true thickness of the geometry.

Thickness

1 In the **Model Builder** window, expand the **Thickness and Orientation (shell)** node.
2 Right-click **Thickness** and choose **Delete**.

Thickness and Orientation (shell)

In the **Model Builder** window, under **Results** click **Thickness and Orientation (shell)**.

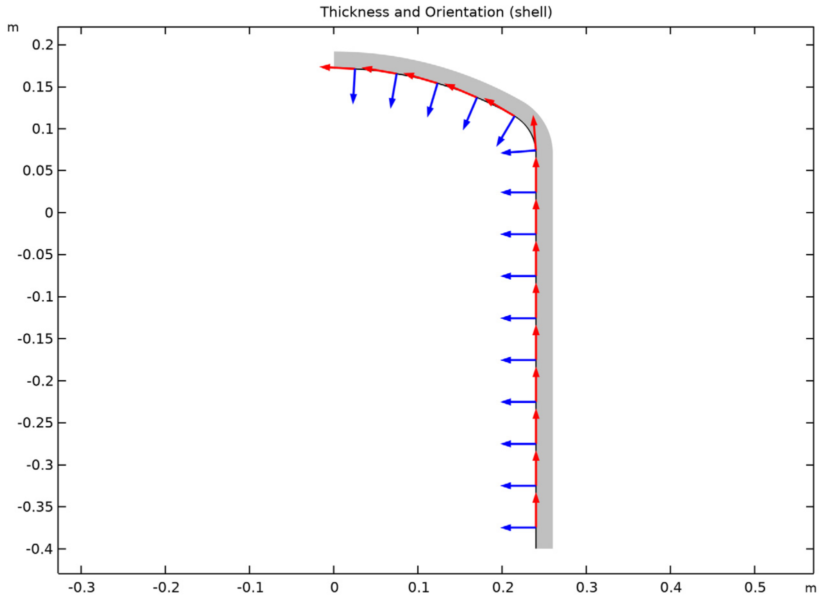
Surface 1

1 In the **Thickness and Orientation (shell)** toolbar, click **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Layered Material**.
4 From the **Solution parameters** list, choose **From parent**.
5 Locate the **Expression** section. In the **Expression** text field, type 1.
6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
7 From the **Color** list, choose **Gray**.

Shell Local System

1 In the **Model Builder** window, click **Shell Local System**.
2 In the **Settings** window for **Coordinate System Line**, locate the **Positioning** section.

3 In the **Number of points** text field, type 15.



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 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type Pressure (N/m²).

5 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

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