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 $R_\mathrm{i} = 24$ 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 $R_\mathrm{c} = 43.2$ cm, the knuckle has an internal radius of $R_\mathrm{k} = 5.2$ 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.

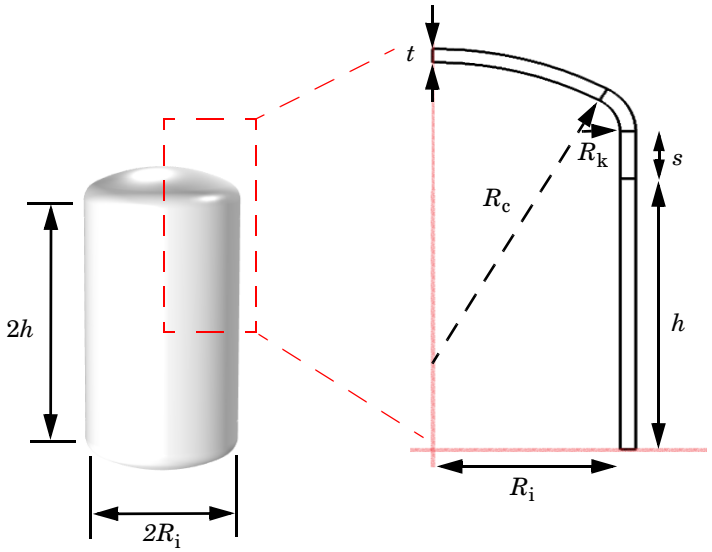

Figure 1: Schematic description of the container geometry and dimensions.

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

$$\begin{bmatrix} \sigma_{ys1} \\ \sigma_{ys2} \\ \sigma_{ys3} \\ \tau_{ys23} \\ \tau_{ys31} \\ \tau_{ys12} \end{bmatrix} = \begin{bmatrix} 381 \\ 381 \\ 450 \\ 240 \\ 240 \\ 220 \end{bmatrix} \text{MPa} \quad (1)$$

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; see Figure 2.

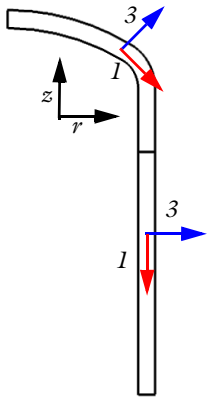

Figure 2: Orientation of local material coordinate system. The second principal direction is oriented in the circumferential direction, perpendicular to the rz-plane.

The curvilinear coordinates interface is used to compute an orthogonal system of coordinates able to follow the container's shape.

Results and Discussion

An approximative 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:

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

Following Hill's criterion, the yielding will occur when

$$F(\sigma_2-\sigma_3)^2 + G(\sigma_3-\sigma_1)^2 + H(\sigma_1-\sigma_2)^2 = 1 \tag{3}$$

or replacing by the expressions in Equation 2

$$p^2\left[F\left(\frac{R_i}{t}+1\right)^2+G\left(1+\frac{R_i}{2t}\right)^2+H\left(\frac{R_i}{2t}-\frac{R_i}{t}\right)^2\right]=1 \quad (4)$$

The material parameters, $F = G = 2.47\cdot10^{-18}$ $1/\text{Pa}^2$ and $H = 4.42\cdot10^{-18}$ $1/\text{Pa}^2$, 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 will undergo 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 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.

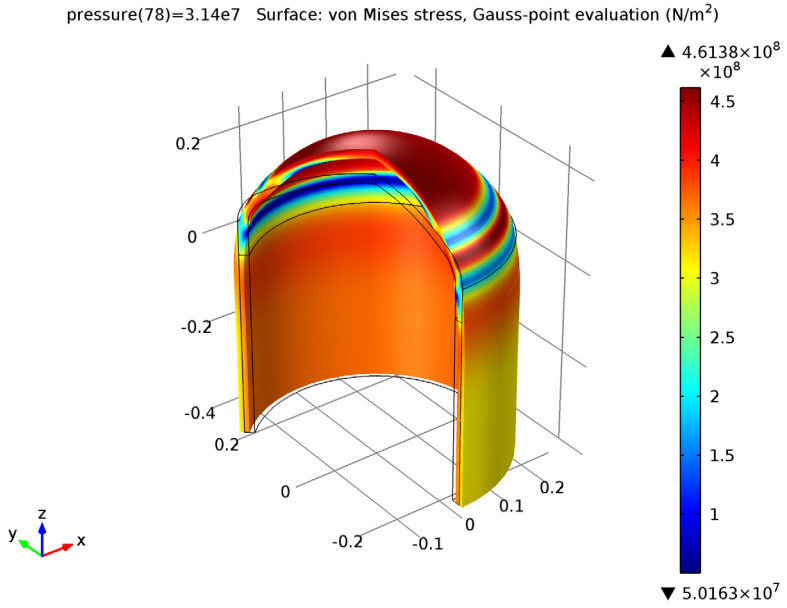

Figure 3: Effective stress at the onset of yielding.

Notes About the COMSOL Implementation

Hill orthotropic plasticity is available in COMSOL as a built-in option under the Plasticity feature, 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 **Linear Elastic Material** settings window's **Coordinate System Selection** section.

Figure 4 visualizes the base vector system defined by the curvilinear system using Arrow Surface plots. The red arrows denote the direction 1, while the blue arrows denote direction 3

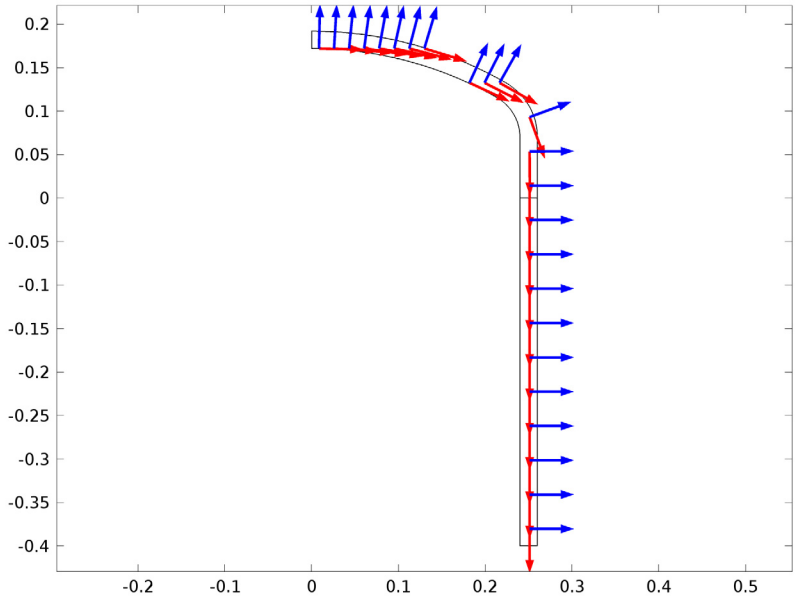

Figure 4: Orientation of the curvilinear system.

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 will save much computation time from being spent in the strongly nonlinear regime.

Model Library path: `Nonlinear_Structural_Materials_Module/Plasticity/orthotropic_container`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click the **2D Axisymmetric** button.

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 **Preset Studies for Selected Physics>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters from file.

Parameters

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

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

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

DEFINITIONS

Integration 1

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

2 In the **Integration** settings window, 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)**.

5 Select the **Compute integral in revolved geometry** check box.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, 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

1 In the **Model Builder** window, under **Component 1** right-click **Geometry 1** and choose **Circle**.

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

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

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

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

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

Circle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.

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

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

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

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

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

Difference 1

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

2 Select the object **c2** only.

3 In the **Difference** settings window, locate the **Difference** section.

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

5 Select the object **c1** only.

6 Right-click **Component 1>Geometry 1>Difference 1** and choose **Rename**.

7 In the **Rename Difference** dialog box, type `Crown` in the **New name** edit field.

8 Click **OK**.

Circle 3

1 Right-click **Geometry 1** and choose **Circle**.

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

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

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

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

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

7 Click the **Build Selected** button.

Circle 4

1 Right-click **Geometry 1** and choose **Circle**.

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

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

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

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

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

7 Click the **Build Selected** button.

Difference 2

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

2 Select the object **c3** only.

3 In the **Difference** settings window, locate the **Difference** section.

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

5 Select the object **c4** only.

6 Click the **Build Selected** button.

7 In the **Model Builder** window, right-click **Difference 2** and choose **Rename**.

8 In the **Rename Difference** dialog box, type `Knuckle` in the **New name** edit field.

9 Click **OK**.

Rectangle 1

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `th`.

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

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

6 Right-click **Component 1>Geometry 1>Rectangle 1** and choose **Rename**.

7 In the **Rename Rectangle** dialog box, type `Flange` in the **New name** edit field.

8 Click **OK**.

9 Right-click **Component 1>Geometry 1>Flange** and choose **Build All Objects**.

Union 1

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

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

3 In the **Union** settings window, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Right-click **Component 1>Geometry 1>Union 1** and choose **Rename**.

6 In the **Rename Union** dialog box, type `Klopper head` in the **New name** edit field.

7 Click **OK**.

8 Right-click **Component 1>Geometry 1>Klopper head** and choose **Build All Objects**.

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

Rectangle 2

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `th`.

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

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

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

7 Right-click **Component 1>Geometry 1>Rectangle 2** and choose **Rename**.

8 In the **Rename Rectangle** dialog box, type `Cylinder` in the **New name** edit field.

9 Click **OK**.

10 Right-click **Component 1>Geometry 1>Cylinder** and choose **Build All Objects**.

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

CURVILINEAR COORDINATES

1 In the **Model Builder** window, under **Component 1** click **Curvilinear Coordinates**.

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

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

Diffusion Method 1

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

2 In the **Diffusion Method** settings window, locate the **Domain Selection** section.

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

Inlet 1

1 Right-click **Component 1>Curvilinear Coordinates>Diffusion Method 1** and choose **Inlet**.

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

3 Click **Clear Selection**.

4 Select Boundary 1 only.

Coordinate System Settings 1

1 In the **Model Builder** window, under **Component 1>Curvilinear Coordinates** click **Coordinate System Settings 1**.

2 In the **Coordinate System Settings** settings window, locate the **Settings** section.

3 From the **Second basis vector** list, choose **phi-axis**.

Outlet 1

1 In the **Model Builder** window, under **Component 1>Curvilinear Coordinates** right-click **Diffusion Method 1** and choose **Outlet**.

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

3 Click **Clear Selection**.

4 Select Boundary 3 only.

SOLID MECHANICS

Symmetry 1

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

2 Select Boundary 3 only.

Boundary Load 1

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

2 In the **Model Builder** window, under **Component 1>Solid Mechanics** right-click **Boundary Load 1** and choose **Rename**.

3 In the **Rename Boundary Load** dialog box, type `Boundary Load Pressure` in the **New name** edit field.

4 Click **OK**.

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

6 In the **Boundary Load** settings window, locate the **Force** section.

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

8 In the p edit field, type `pressure`.

Linear Elastic Material 1

1 In the **Linear Elastic Material** settings window, locate the **Coordinate System Selection** section.

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

Plasticity 1

1 Right-click **Component 1>Solid Mechanics>Linear Elastic Material 1** and choose **Plasticity**.

2 In the **Plasticity** settings window, locate the **Plasticity Model** section.

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

4 From the **Hardening model** list, choose **Perfectly plastic**.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material**.

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

3 In the tree, select **Built-In>Steel AISI 4340**.

4 In the **Add material** window, click **Add to Component**.

MATERIALS

Steel AISI 4340

1 In the **Model Builder** window, under **Component 1>Materials** click **Steel AISI 4340**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Initial tensile and shear yield stresses	ys	{381e6, 381e6, 450e6, 240e6, 240e6, 220e6}	N/m²	Elastoplastic material model

5 On the **Home** toolbar, click **Add Material**.

MESH 1

Mapped 1

In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Mapped**.

Distribution 1

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

2 Select Boundary 1 only.

Distribution 2

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

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

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Fixed number of elements**.

5 In the **Number of elements** edit field, type 10.

Distribution 3

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

2 In the **Distribution** settings window, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Fixed number of elements**.

4 In the **Number of elements** edit field, type 25.

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

6 Click the **Build All** button.

The mesh should consist of 350 quadrilateral elements, with 5 element through 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 **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	Solve for	Discretization
Solid Mechanics {solid}	×	physics

Step 2: Stationary 2

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

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

3 In the table, enter the following settings:

Physics	Solve for	Discretization
Curvilinear Coordinates {cc}	×	physics

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

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

5 Click **Add**.

6 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
pressure	`range(16e6, 2e5, 36e6)`

Solver 1

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

2 In the **Model Builder** window, expand the **Solver 1** node.

Introduce a stop condition to stop the solver when 10% of the steel volume has gone through plastic deformation.

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

4 Right-click **Parametric 1** and choose **Stop Condition**.

5 In the **Stop Condition** settings window, 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	√	`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 On the **Home** toolbar, click **Compute**.

RESULTS

Coordinate System (cc)

1 In the **2D Plot Group** settings window, click to expand the **Title** section.

2 From the **Title type** list, choose **None**.

3 Right-click **Results>Coordinate System (cc)** and choose **Rename**.

4 In the **Rename 2D Plot Group** dialog box, type `Material principal direction` in the **New name** edit field.

5 Click **OK**.

2D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **2D Plot Group 5** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Plastic strain 2D` in the **New name** edit field.

4 Click **OK**.

Plastic strain 2D

1 Right-click **Results>Plastic strain 2D** and choose **Surface**.

2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solid Mechanics>Strain (Gauss points)>Effective plastic strain (solid.epeGp)**.

3 On the **2D plot group** toolbar, click **Plot**.

The onset of plasticity can be investigated by evaluating the volume fraction of the material which has exceeded the yield stress. It occurs at a pressure between 31.2 MPa and 31.4 MPa.

1D Plot Group 6

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

2 In the **1D Plot Group** settings window, click to expand the **Title** section.

3 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.

6 On the **1D plot group** toolbar, click **Global**.

7 In the **Global** settings window, locate the **y-Axis Data** section.

8 In the table, enter the following settings:

Expression	Unit	Description
y_vol	1	Yielded volume fraction

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

10 On the **1D plot group** toolbar, click **Plot**.

11 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Rename**.

12 In the **Rename 1D Plot Group** dialog box, type `Yielded volume` in the **New name** edit field.

13 Click **OK**.

Stress, 3D (solid)

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

2 On the **3D plot group** toolbar, click **Plot**.