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 exemplifies how to define and use a base vector system aligned with the principal directions of the material, which in this case follow the contours of the container.

Model Definition

The structure has the shape of a cylinder capped by two half spheres. The cylinder has a radius $R_0 = 25$ cm, height $H_0 = 80$ cm, and thickness $T_0 = 2$ cm; see Figure 1. The cylinder radius and thickness are also the radius and thickness of the half spheres. Because of 2D axial symmetry and reflection symmetry, it is sufficient to model a quarter of the container; see Figure 1. The red dash-dotted line defines the rotation symmetry axis whereas the red dashed line is the reflection symmetry axis. The radius is measured to the center of the thin wall. The variable p denotes the internal pressure.

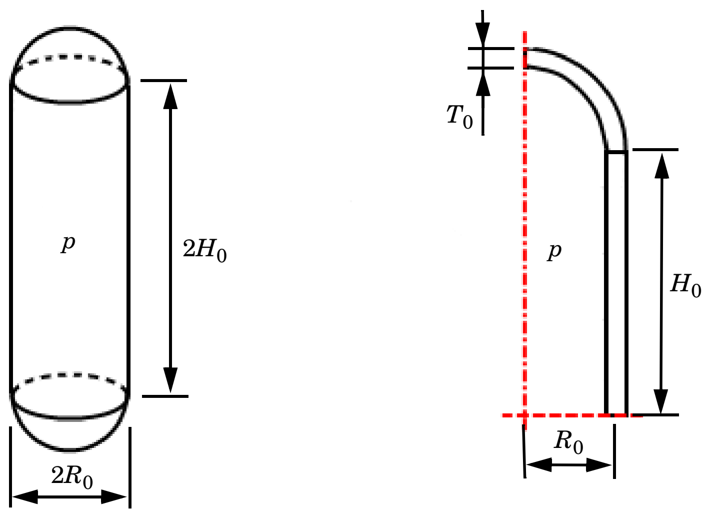

Figure 1: Schematic description of the container geometry and dimensions.

MATERIAL MODEL

The elastoplastic material is defined by a Young's modulus, E, of 210 GPa and a Poisson's ratio, ν, of 0.30. Hill's orthotropic plasticity governs the yielding, with the yield stress components given by

$$\begin{bmatrix} \sigma_{ys1} \\ \sigma_{ys2} \\ \sigma_{ys3} \\ \tau_{ys23} \\ \tau_{ys31} \\ \tau_{ys12} \end{bmatrix} = \begin{bmatrix} 381 \\ 381 \\ 450 \\ 240 \\ 240 \\ 220 \end{bmatrix} \mathrm{MPa} \tag{1}$$

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 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.

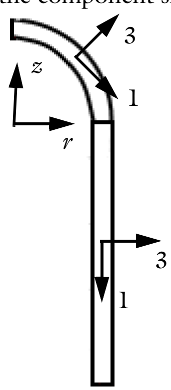

Figure 2: Orientation of local material coordinate system. The second principal direction is oriented in the circumferential direction, perpendicular to the rz-plane.

The container structure can be split into two domains: one representing the spherical cap, and another representing the cylinder. For the cylindrical part the material coordinate system does not change and is given by

$$\begin{aligned} \bar{e}_1 &= -\bar{e}_z \\ \bar{e}_2 &= \bar{e}_\varphi \\ \bar{e}_3 &= \bar{e}_r \end{aligned} \tag{2}$$

where $\bar{e}_i$ denotes the direction vector of the coordinate i.

For the spherical part, the coordinate system is defined by two simple rotations; see Figure 3.

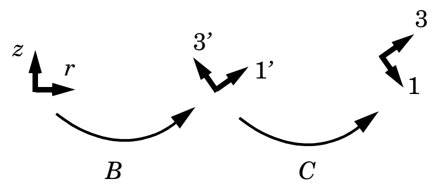

Figure 3: Transformation of the material orientation in the spherical part.

In the first rotation, B, the coordinate system is rotated with an angle $\alpha = \mathrm{atan}\frac{z}{r}$ and in the second rotation, C, the principal directions swaps.

These rotations are written in the matrix form

$$\begin{bmatrix} \bar{e}_1 \\ \bar{e}_2 \\ \bar{e}_3 \end{bmatrix} = \begin{bmatrix} 0 & 0 & -1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{bmatrix} \begin{bmatrix} \cos\alpha & 0 & \sin\alpha \\ 0 & 1 & 0 \\ -\sin\alpha & 0 & \cos\alpha \end{bmatrix} \begin{bmatrix} \bar{e}_r \\ \bar{e}_\varphi \\ \bar{e}_z \end{bmatrix} \tag{3}$$

and the local coordinate system for the spherical part of the container is given by

$$\begin{aligned} \bar{e}_1 &= \sin(\alpha)\bar{e}_r - \cos(\alpha)\bar{e}_z \\ \bar{e}_2 &= \bar{e}_\varphi \\ \bar{e}_3 &= \cos(\alpha)\bar{e}_r + \sin(\alpha)\bar{e}_z \end{aligned} \tag{4}$$

Results and Discussion

An approximative analytical solution can be obtained for the cylindrical part of the container. For the inner wall it is approximately

$$\sigma_1 = p\frac{R_0}{2T_0}$$
$$\sigma_2 = p\frac{R_0}{T_0} \qquad (5)$$
$$\sigma_3 = -p$$

Following Hill's criterion, the yielding will occur for

$$p^2\left[F\left(\frac{R_0}{T_0}+1\right)^2 + G\left(1+\frac{R_0}{2T_0}\right)^2 + H\left(\frac{R_0}{2T_0}-\frac{R_0}{T_0}\right)^2\right] = 1 \qquad (6)$$

Using the material parameters, $F = G = 2.47 \cdot 10^{-18}\ 1/\mathrm{Pa}^2$ and $H = 4.42 \cdot 10^{-18}\ 1/\mathrm{Pa}^2$, the analytical onset of orthotropic yielding occurs for $p = 36.5$ MPa as compared to $p = 35.8$ MPa which is the result calculated by COMSOL.

Figure 4 shows the von Mises stress contours at the onset of yielding. For isotropic steel with yield stress of 381 MPa, the yield stress is reached when $p = 32.4$ MPa. Therefore, with orthotropic steel, the pressure needed for the onset of plasticity is about 10% higher.

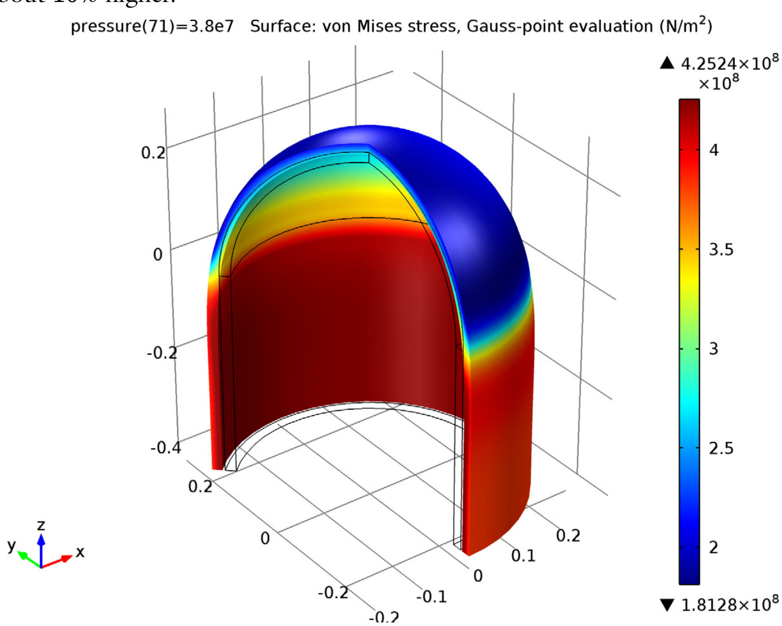

Figure 4: Effective stress at the onset of yielding.

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 in the following steps. From the **Model>Definitions** node's context menu, choose **Coordinate Systems>Base Vector System**. For axisymmetric geometries, the new base vectors, x_1 and x_3, are expressed in the base vectors r and z. 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*). In this case, which is geometrically linear, there is no difference. Generally, material properties should always be defined in terms of the initial configuration. The built-in variable `dom` contains the number of the domain, and can be used to specify a domain of validity; see Figure 5. In this model, `dom` = 1 represents the half sphere where Equation 4 is valid, while `dom` = 2 represents the cylinder where Equation 2 is used.

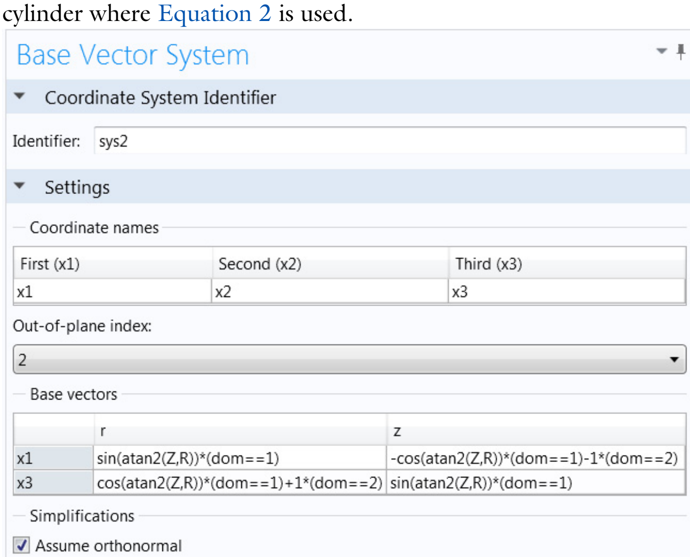

Figure 5: Definition of a new base vector system.

To assign the new base vector system to the component, select it from the **Coordinate system** list in the **Linear Elastic Material** settings window's **Coordinate System Selection** section.

Figure 6 visualizes the base vector system defined by Equation 2 and Equation 4 using a Coordinate System Surface plot (accessible from a 2D Plot Group node's context menu under More Plots).

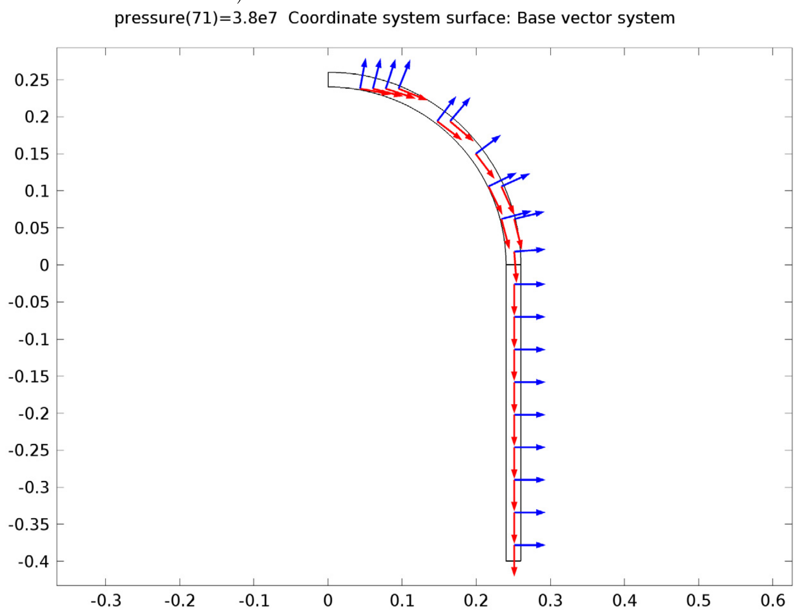

Figure 6: Orientation of the base vector system used in the material model.

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click the **Add** button.

4 Click the **Study** button.

5 In the tree, select **Preset Studies>Stationary**.

6 Click the **Done** button.

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 `0.26`.

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

5 Click the **Build Selected** button.

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 `0.24`.

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

5 Click the **Build Selected** button.

Difference 1

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

2 Select the object **c1** only.

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

4 Select the **Objects to subtract** toggle button.

5 Select the object **c2** only.

6 Click the **Build Selected** button.

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 `0.02`.

4 In the **Height** edit field, type `0.4`.

5 Locate the **Position** section. In the **r** edit field, type `0.25`.

6 In the **z** edit field, type `-0.2`.

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

8 Click the **Build All Objects** button.

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

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
pressure	1[N/m^2]	1.000 N/m²	Internal pressure

DEFINITIONS

Base Vector System 2

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

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

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

	r	**z**
x1	sin(atan2(Z,R))*(dom==1)	-cos(atan2(Z,R))*(dom==1)-1*(dom==2)
x3	cos(atan2(Z,R))*(dom==1)+1*(dom==2)	sin(atan2(Z,R))*(dom==1)

4 Select the **Assume orthonormal** check box.

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 Go to the **Rename Boundary Load** dialog box and type `Boundary Load Pressure` in the **New name** edit field.
4 Click **OK**.
5 Select Boundaries 2 and 6 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 **Base Vector System 2**.

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**.

MATERIALS

Material 1

5 In the **Material** settings window, locate the **Material Contents** section.
6 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	210e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	1	kg/m³	Basic
Initial tensile and shear yield stresses	ys	{381e6, 381e6, 450e6, 240e6, 240e6, 220e6}	N/m²	Elastoplastic material model

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 6 and 7 only.
3 In the **Distribution** settings window, locate the **Distribution** section.
4 From the **Distribution properties** list, choose **Predefined distribution type**.
5 In the **Element ratio** edit field, type `4`.
6 In the **Number of elements** edit field, type `15`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 2 and 5 only.
3 In the **Distribution** settings window, locate the **Distribution** section.
4 From the **Distribution properties** list, choose **Predefined distribution type**.
5 In the **Number of elements** edit field, type `15`.
6 In the **Element ratio** edit field, type `4`.
7 Select the **Reverse direction** check box.
8 Click the **Build All** button.

   The mesh should consist of 150 elements. Finer elements are created at the connection between the cylinder and the half sphere since due to geometrical change stress gradients are expected there.

STUDY 1

Step 1: Stationary

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

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
2 In the **Stationary** settings window, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
pressure	`range(31e6, 0.1e6, 38e6)`

6 On the **Home** toolbar, click **Compute**.

RESULTS

2D Plot Group 3

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 3** and choose **Rename**.

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

4 Click **OK**.

Plastic strain 2D

1 Right-click **Results>2D Plot Group 3** 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 of the material which has exceeded the yield stress. It occurs at a pressure between 35.7 MPa and 35.8 MPa.

Derived Values

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 Select Domains 1 and 2 only.

3 In the **Surface Integration** settings window, locate the **Expression** section.

4 In the **Expression** edit field, type `solid.epeGp>0`.

5 Select the **Description** check box.

6 In the associated edit field, type `Volume having reached yield stress`.

7 Locate the **Integration Settings** section. Select the **Compute volume integral** check box.
8 Click the **Evaluate** button.

TABLE

In the **Table** window, click **Table Graph**.

RESULTS

1D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.
2 Go to the **Rename 1D Plot Group** dialog box and type `Yielded volume` in the **New name** edit field.
3 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 Go to the **Rename 2D Plot Group** dialog box and type `Material principal direction` in the **New name** edit field.
4 Click **OK**.

Material principal direction

1 On the **2D plot group** toolbar, click **More Plots** and choose **Coordinate System Surface**.
2 In the **Coordinate System Surface** settings window, locate the **Coordinate System** section.
3 From the **Coordinate system** list, choose **Base Vector System 2**.
4 On the **2D plot group** toolbar, click **Plot**.

Stress, 3D (solid)

1 In the **Model Builder** window, expand the **Results>Stress, 3D (solid)>Surface 1** node, then click **Deformation**.
2 In the **Deformation** settings window, locate the **Scale** section.
3 Select the **Scale factor** check box.
4 In the associated edit field, type `50`.

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