



Model created in COMSOL Multiphysics 6.4

Temperature-Dependent Plasticity in Pressure Vessel

Introduction

This example demonstrates how to use temperature dependent materials within the Nonlinear Structural Materials Module. Material data such as Young's modulus, yield stress and strain hardening have strong temperature dependencies.

A large container holds pressurized hot water. Several pipes are attached to the pressure vessel. Those pipes can rapidly transfer cold water in case of an emergency cooling. The pressure vessel is made of carbon steel with an internal cladding of stainless steel. In case of a fast temperature transient, the differences in thermal expansion properties between the materials cause high stresses.

Model Definition

GEOMETRY

The pressure vessel has the shape of a closed cylinder. Four pipes are attached at two levels along its height. At each level, the pipes are equidistantly spaced around the container.

The pipes are welded into the vessel and the welding can be considered as a chamfer between those two parts. The structure, together with its key dimensions, is presented in [Figure 1](#)

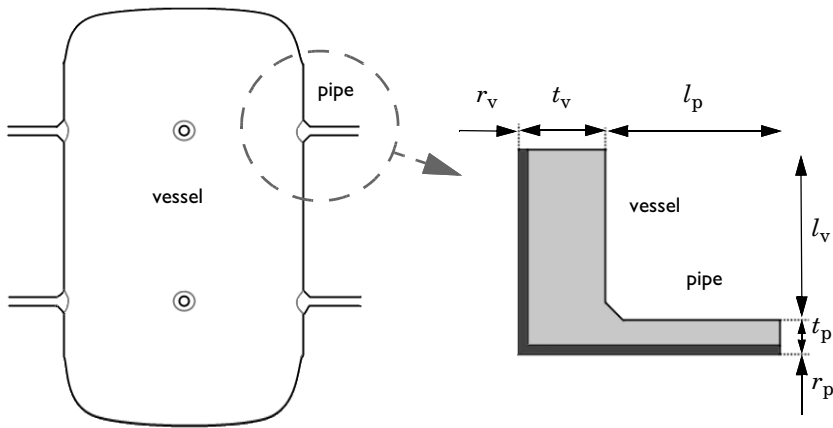


Figure 1: Pressure vessel and the dimensions for the vessel-pipe connection.

The structure has the following dimensions:

- Inner vessel radius, $r_v = 1000$ mm
- Inner pipe radius, $r_p = 60$ mm

- Vessel thickness, $t_v = 100$ mm
- Pipe thickness, $t_p = 40$ mm

The vessel length, l_v , and the pipe length, l_p , are large compared to the thickness of both parts. For modeling purposes, they need to be large enough so that local effects at the vessel-pipe connection can be disregarded. The chamfer extends 20 mm from the corner at the connection between the pipe and the vessel.

The dual material consists of a thin 10 mm layer of stainless steel (dark gray in [Figure 1](#)) that faces the water, and carbon steel (light gray in [Figure 1](#)) that faces the outside air.

In order to save computational time, only the connection between one pipe and the vessel is modeled, as shown on the left image of [Figure 1](#).

MATERIAL MODEL

The thermoelastic material data of stainless steel is given in [Table 1](#). [Table 2](#) shows the yield stress as a function of plastic strain at temperatures of 20, 100, 200, and 300°C.

TABLE 1: THERMOELASTIC MATERIAL DATA OF STAINLESS STEEL.

| Temperature (°C) | 20 | 100 | 200 | 300 |
|------------------|--------------------|----------------------|--------------------|----------------------|
| E (GPa) | 194 | 189 | 179 | 175 |
| α (1/°C) | $16 \cdot 10^{-6}$ | $16.5 \cdot 10^{-6}$ | $17 \cdot 10^{-6}$ | $17.5 \cdot 10^{-6}$ |
| c_p (J/(kg·K)) | 482 | 498 | 515 | 524 |
| k (W/(m·K)) | 13.9 | 14.9 | 17.0 | 18.0 |

TABLE 2: TEMPERATURE DEPENDENT YIELD STRESS OF STAINLESS STEEL.

| Temperature (°C) | 20 | 100 | 200 | 300 |
|-----------------------------|-----|-----|-----|-----|
| $\sigma_{ys}(0.0)$ (MPa) | 228 | 190 | 156 | 140 |
| $\sigma_{ys}(0.0004)$ (MPa) | 232 | 195 | 160 | 144 |
| $\sigma_{ys}(0.001)$ (MPa) | 238 | 201 | 166 | 148 |
| $\sigma_{ys}(0.002)$ (MPa) | 246 | 210 | 173 | 155 |
| $\sigma_{ys}(0.004)$ (MPa) | 250 | 215 | 177 | 158 |
| $\sigma_{ys}(0.001)$ (MPa) | 263 | 230 | 189 | 169 |

The yield stress of the carbon steel is two or three times higher than that of stainless steel. It is therefore considered as purely elastic. Its material properties are shown in [Table 3](#).

TABLE 3: THERMOELASTIC MATERIAL DATA OF CARBON STEEL.

| Temperature (°C) | 20 | 100 | 200 | 300 |
|------------------|----------------------|----------------------|----------------------|----------------------|
| E (GPa) | 208 | 202 | 196 | 189 |
| α (1/°C) | $10.9 \cdot 10^{-6}$ | $12.4 \cdot 10^{-6}$ | $13.8 \cdot 10^{-6}$ | $14.9 \cdot 10^{-6}$ |
| c_p (J/(kg·K)) | 489 | 519 | 546 | 569 |
| k (W/(m·K)) | 51.2 | 48.3 | 45.5 | 42.7 |

The heat transfer coefficient between steel and air is $10 \text{ W}/(\text{m}^2 \cdot \text{K})$, and between steel and water it is $100 \text{ W}/(\text{m}^2 \cdot \text{K})$.

BOUNDARY CONDITIONS

A pressure of 70 bar acts on the inner walls of the vessel and the pipe. The temperature on the inside of the pressure vessel is initially at 280°C , while the outside air remains at 50°C . Suddenly and instantaneously, cold water at 20°C is pumped through the pipe into the vessel, where the hot water needs 30 minutes to cool down to 20°C . The cooling speed is constant.

MODEL ASSUMPTIONS

Due to symmetries, only a 45° sector of the vessel is modeled. The influence of the hot water pressure at the end of the vessel is approximated with an axial stress of 33.3 MPa , which is 4.76 times the inner pressure. The parameters l_v and l_p are both set to 200 mm.

Results and Discussion

Three studies are performed in this analysis. In an initial step, the mechanical and thermal stationary state is computed. This serves as initial conditions for a transient step which solves the heat transfer problem only, where the cold water flows through the pipe, cooling the initially hot water in the vessel. A comparison of the temperature profiles, before and after the event, are shown in [Figure 2](#). After 30 min, the water inside the vessel has cooled down to 20°C , but the container is still locally more than 100°C warmer. This leads to large gradients in the thermal strains.

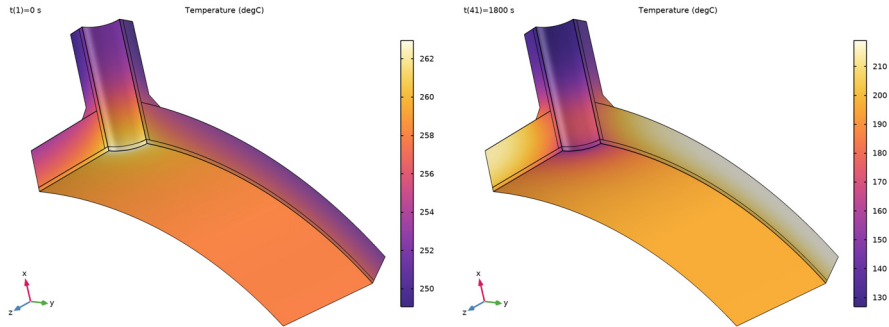


Figure 2: Temperature profiles before (left) and after the cooling event (right).

The last step solves the elastoplastic deformation with temperature-dependent material parameters. The development of plastic strains in the stainless steel layer is shown in Figure 3. From the figure it can be seen that a plastic zone develops as the vessel cools down. Initially, when the vessel is at steady state, some plastic strains are generated by stresses caused by differences in the thermal expansion of the two steels. In a real structure, such stresses would have been relaxed after the first service cycle. In the transient study, when the vessel cools down and the yield limit increases, the pipe deforms plastically in other locations.

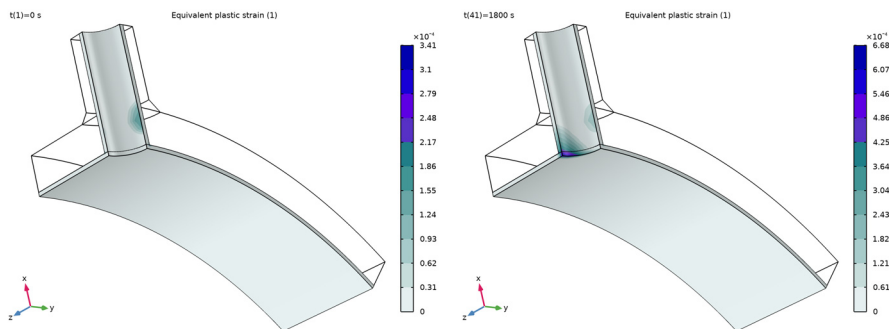


Figure 3: Plastic strains before (left) and after the cooling event (right).

As the temperature decreases, the yield stress increases, which means that warm parts are more sensitive to high stresses. [Figure 4](#) shows the von Mises stress after 30 min of cooling.

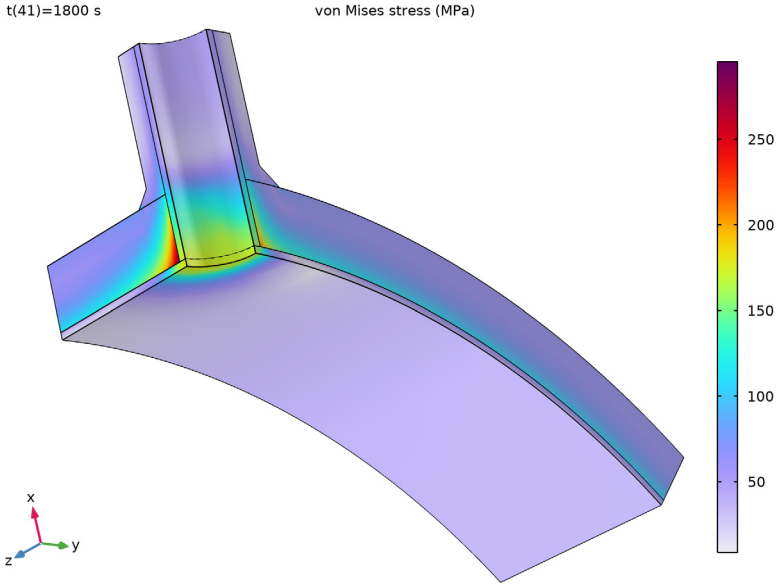


Figure 4: Distribution of von Mises stress after cooling down the hot water.

[Figure 5](#) shows the membrane plus bending stress intensity distribution during the cooling. It is clear that the location of the maximum intensity changes over time. At the end of the cooling the maximum remains close to the nozzle.

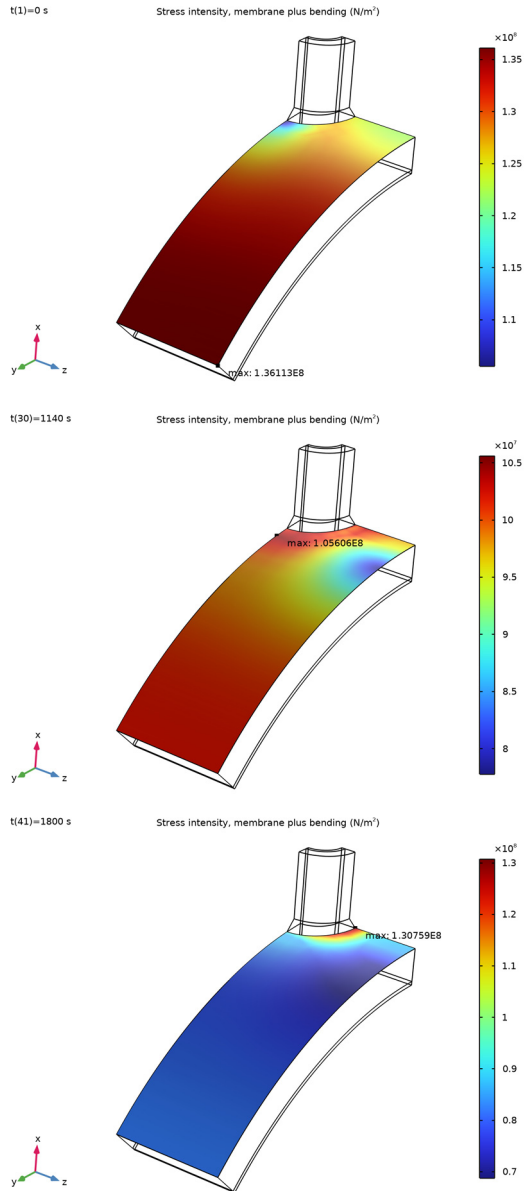


Figure 5: Membrane plus bending stress intensity at initial time (top), 1140 s (middle), and 1800 s (bottom).

Figure 6 shows the development of the maximum of the membrane plus bending stress intensity during the cooling. One can notice that there is a minimum at around 1140 s.

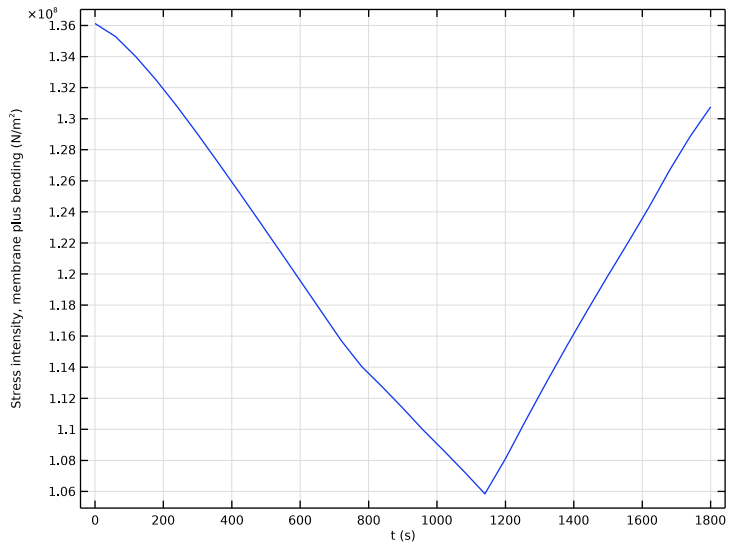


Figure 6: Maximum of membrane plus bending stress intensity overtime.

Figure 7 shows the computed and linearized stress at initial time (solid) and at the end of the cooling process (dotted line) compared to the computed, spatially resolved stress component projected in the membrane direction. The jumps in the computed values originate from the different material properties in the two domains of the wall of the pressure vessel. Thus, the computed stress projected in the membrane direction features a jump over the thickness of the wall. The linearized membrane stress is by definition constant and the linearized bending stress is by definition linear over the wall thickness.

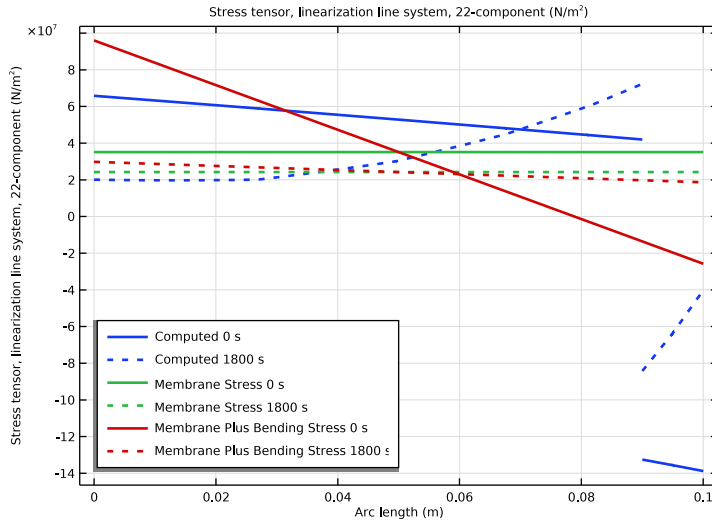


Figure 7: Computed stress (blue), Membrane stress (green), and Membrane plus bending stress (red) at initial time (solid line) and final time (dotted line).

Notes About the COMSOL Implementation

COMSOL Multiphysics can handle material data depending on several parameters. Use Interpolation functions, which you can select from the **Definitions** or **Materials** nodes. You can type in the data in a table or define your function in a text file. Use the symbol % in a text file to include comments or headers.

In this example, the Young’s modulus $E(T)$, coefficient of thermal expansion $\alpha(T)$, thermal conductivity $k(T)$, and heat capacity at constant pressure $C_p(T)$ are defined in the Materials node from interpolated data. The initial yield stress $\sigma_{y0}(T)$ and the nonlinear hardening function $\sigma_h(T, \epsilon_{pe})$ are defined by data imported from a text file.

Interpolation functions can handle any number of arguments. For convenience, specify units (Pa, m, s, and so on) for the function and arguments. When applicable, COMSOL Multiphysics automatically scales any input into the correct unit. For more details see the section *Operators, Functions, and Constants* in the *COMSOL Multiphysics Reference Manual*.

In this example there are two cuts that are “almost symmetry cuts” in the sense that they should stay plane but are still allowed to move in the normal direction. One is the cut in

the pipe, and the other is the cut through the pressure vessel wall. In order to accomplish this, a **Symmetry** node is used, but with free displacement as a normal direction condition. Thus the normal direction displacement is constant, but yet determined by the solution. The **Symmetry** node has different flavors of normal direction constraints, where the default is equivalent to a traditional symmetry condition.


Heat transfer in solids is a time-dependent phenomenon, and since plasticity is a path-dependent process, it is important to capture the evolution of temperature profiles accurately. You need to limit the time step in the BDF solver settings to account for the rate at which thermal loads change in time. Use a different study step to compute the elastoplastic deformation after computing the heat transfer. Since this is not a coupled problem, this segregated approach reduces computational time.

Application Library path: Nonlinear_Structural_Materials_Module/
Plasticity/temperature_dependent_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 > Thermal–Structure Interaction > Thermal Stress, Solid**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 Click  **Done**.

GLOBAL DEFINITIONS



Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

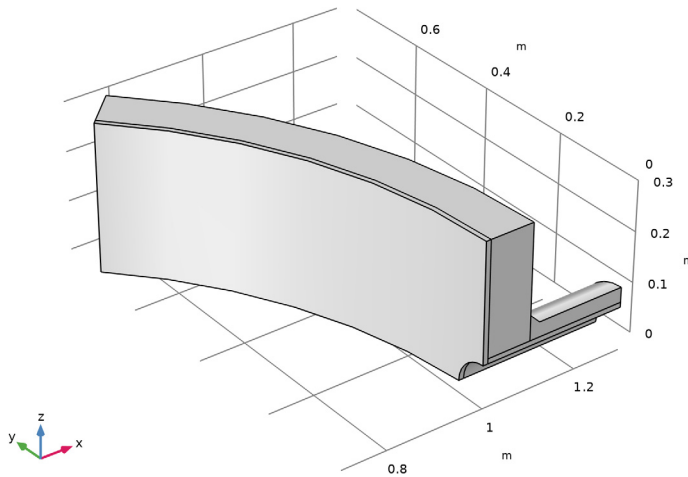
| Name | Expression | Value | Description |
|------------------|------------|--------|---|
| internalPressure | 70[bar] | 7E6 Pa | Internal pressure |
| t | 0[s] | 0 s | Time variable; used for stationary analysis |

The thermal shock caused by the cold water is time dependent. Therefore, the time variable, t , needs to be set to zero so that the heat boundary conditions can be evaluated also in the static analysis.

GEOMETRY I

- 1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
- 2 Browse to the model's Application Libraries folder and double-click the file `temperature_dependent_plasticity_geom_sequence.mph`.
- 3 In the **Geometry** toolbar, click  **Build All**.
- 4 Click the  **Go to Default View** button in the **Graphics** toolbar.

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.




Full geometry instructions can be found in the section [Appendix — Geometry Modeling Instructions](#).

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Plasticity 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Plasticity**.
- 2 Select Domains 1 and 3 only.
- 3 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 4 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

MATERIALS

Stainless Steel

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type **Stainless Steel** in the **Label** text field.
- 3 Select Domains 1 and 3 only.

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

| Property | Variable | Value | Unit | Property group |
|-----------------|----------|-------|-------------------|-------------------------------------|
| Poisson's ratio | nu | 0.3 | | Young's modulus and Poisson's ratio |
| Density | rho | 8000 | kg/m ³ | Basic |

Add Young's modulus as a function of temperature for the stainless steel.

Interpolation 1 (int1)

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Materials > Stainless Steel (mat1)** node.
- 2 Right-click **Component 1 (comp1) > Materials > Stainless Steel (mat1) > Young's modulus and Poisson's ratio (Enu)** and choose **Functions > Interpolation**.
- 3 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 4 In the **Function name** text field, type **fE**.
- 5 In the table, enter the following settings:

| t | f(t) |
|-----|------|
| 20 | 194 |
| 100 | 189 |
| 200 | 179 |
| 300 | 175 |

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

7 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|------|
| fE | GPa |

Stainless Steel (mat1)

Add the temperature as a model input for the Young's modulus property.

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Materials > Stainless Steel (mat1)** click **Young's modulus and Poisson's ratio (Enu)**.

- 2 In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Model Inputs** section.
- 3 Click **+ Select Quantity**.
- 4 In the **Physical Quantity** dialog, type temperature in the text field.
- 5 In the tree, select **General > Temperature (K)**.
- 6 Click **OK**.
- 7 In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Output Properties** section.
- 8 In the table, enter the following settings:

| Property | Variable | Expression | Unit | Size |
|-----------------|----------|------------|------|------|
| Young's modulus | E | fE(T) | Pa | x |

Add the coefficient of thermal expansion as a function of temperature for the stainless steel.

Interpolation 1 (int1)

- 1 In the **Home** toolbar, click **f(x) Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type fA.
- 4 In the table, enter the following settings:

| t | f(t) |
|-----|---------|
| 20 | 16e-6 |
| 100 | 16.5e-6 |
| 200 | 17e-6 |
| 300 | 17.5e-6 |

- 5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

- 6 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|------|
| fA | 1/K |

Add the thermal conductivity as a function of temperature for the stainless steel.

Interpolation 2 (int2)

- 1 In the **Home** toolbar, click $f(x)$ **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type fK .
- 4 In the table, enter the following settings:

| t | f(t) |
|----------|-------------|
| 20 | 13.9 |
| 100 | 15.5 |
| 200 | 16.8 |
| 300 | 17.8 |

- 5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|-----------------|-------------|
| t | degC |

- 6 In the **Function** table, enter the following settings:

| Function | Unit |
|-----------------|-------------|
| fK | W/(m*degC) |

Add the heat capacity as a function of temperature for the stainless steel.

Interpolation 3 (int3)

- 1 In the **Home** toolbar, click $f(x)$ **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type fCp .
- 4 In the table, enter the following settings:

| t | f(t) |
|----------|-------------|
| 20 | 482 |
| 100 | 504 |
| 200 | 521 |
| 300 | 530 |

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

6 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|-------------|
| fCp | J/(kg*degC) |

Stainless Steel (mat I)

Add the temperature as a model input for the Basic properties.

1 In the **Model Builder** window, under **Component 1 (comp1) > Materials > Stainless Steel (mat I)** click **Basic (def)**.

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

3 Click **+ Select Quantity**.

4 In the **Physical Quantity** dialog, type temperature in the text field.

5 In the tree, select **General > Temperature (K)**.

6 Click **OK**.

7 In the **Model Builder** window, click **Stainless Steel (mat I)**.

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

9 In the table, enter the following settings:




| Property | Variable | Value | Unit | Property group |
|------------------------------------|--|--------|----------|----------------|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | fK(T) | W/(m·K) | Basic |
| Heat capacity at constant pressure | Cp | fCp(T) | J/(kg·K) | Basic |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | fA(T) | 1/K | Basic |

10 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics > Elastoplastic Material > Elastoplastic Material Model**.

11 Click **+ Add to Material**.



Load the table containing yield stress as function of plastic strain and temperature.

Interpolation 1 (int1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 From the **Data source** list, choose **File**.
- 4 Click  **Browse**.
- 5 Browse to the model's Application Libraries folder and double-click the file `temperature_dependent_plasticity_function.txt`.
- 6 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.
- 7 In the **Unit** text field, type degC.
- 8 In the table, click to select the cell at row number 2 and column number 1.
- 9 In the **Unit** text field, type 1.
- 10 In the table, click to select the cell at row number 3 and column number 1.
- 11 In the **Name** text field, type sY.
- 12 In the **Unit** text field, type Pa.
- 13 Locate the **Definition** section. Click  **Import**.

Stainless Steel (mat1)

Add the temperature and the equivalent plastic strain as model inputs for the Elastoplastic material model properties.

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Materials > Stainless Steel (mat1)** click **Elastoplastic material model (ElastoplasticModel)**.
- 2 In the **Settings** window for **Elastoplastic Material Model**, locate the **Model Inputs** section.
- 3 Click  **Select Quantity**.
- 4 In the **Physical Quantity** dialog, type temperature in the text field.
- 5 In the tree, select **General > Temperature (K)**.
- 6 Click **OK**.
- 7 In the **Settings** window for **Elastoplastic Material Model**, locate the **Model Inputs** section.
- 8 Click  **Select Quantity**.
- 9 In the **Physical Quantity** dialog, type plastic strain in the text field.
- 10 In the tree, select **Solid Mechanics > Equivalent plastic strain (I)**.
- 11 Click **OK**.

12 In the **Settings** window for **Elastoplastic Material Model**, locate the **Output Properties** section.

13 In the table, enter the following settings:

| Property | Variable | Expression | Unit | Size |
|----------------------|----------|-----------------------------------|------|------|
| Initial yield stress | sigmags | $sY(T, 0)$ | Pa | x |
| Hardening function | sigmagh | $sY(T, \epsilon_{pe}) - sY(T, 0)$ | Pa | x |

The hardening function is the stress increase from the initial yield stress. As the full stress–strain curve is given, subtract the stress at zero equivalent plastic strain.

Carbon Steel

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type Carbon Steel in the **Label** text field.

3 Select Domains 2 and 4 only.

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

| Property | Variable | Value | Unit | Property group |
|-----------------|----------|-------|-------------------|-------------------------------------|
| Poisson's ratio | nu | 0.3 | | Young's modulus and Poisson's ratio |
| Density | rho | 8000 | kg/m ³ | Basic |

Add Young's modulus as a function of temperature for the carbon steel.

Interpolation 1 (int1)

1 In the **Model Builder** window, expand the **Component 1 (comp1) > Materials > Carbon Steel (mat2)** node.

2 Right-click **Component 1 (comp1) > Materials > Carbon Steel (mat2) > Young's modulus and Poisson's ratio (Enu)** and choose **Functions > Interpolation**.

3 In the **Settings** window for **Interpolation**, locate the **Definition** section.

4 In the **Function name** text field, type fE .

5 In the table, enter the following settings:

| t | f(t) |
|-----|------|
| 20 | 208 |
| 100 | 202 |
| 200 | 196 |
| 300 | 189 |

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

7 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|------|
| fE | GPa |

Add the temperature as a model input for the Young's modulus property.

Carbon Steel (mat2)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Materials > Carbon Steel (mat2)** click **Young's modulus and Poisson's ratio (Enu)**.
- 2 In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Model Inputs** section.
- 3 Click **+ Select Quantity**.
- 4 In the **Physical Quantity** dialog, type **temperature** in the text field.
- 5 In the tree, select **General > Temperature (K)**.
- 6 Click **OK**.

Add the coefficient of thermal expansion as a function of temperature for the carbon steel.

Interpolation 1 (int1)

- 1 In the **Home** toolbar, click **f(x) Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type **fA**.
- 4 In the table, enter the following settings:

| t | f(t) |
|-----|---------|
| 20 | 10.9e-6 |
| 100 | 12.4e-6 |
| 200 | 13.8e-6 |
| 300 | 14.9e-6 |

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

6 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|------|
| fA | 1/K |

Interpolation 2 (int2)

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

Add the thermal conductivity as a function of temperature for the carbon steel.

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

3 In the **Function name** text field, type fK.

4 In the table, enter the following settings:

| t | f(t) |
|-----|------|
| 20 | 51.2 |
| 100 | 48.3 |
| 200 | 45.5 |
| 300 | 42.7 |

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

6 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|------------|
| fK | W/(m*degC) |

Add the heat capacity as a function of temperature for the carbon steel.

Interpolation 3 (int3)

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

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

3 In the **Function name** text field, type fCp.

4 In the table, enter the following settings:

| t | f(t) |
|-----|------|
| 20 | 489 |
| 100 | 519 |
| 200 | 546 |
| 300 | 569 |

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|----------|------|
| t | degC |

6 In the **Function** table, enter the following settings:

| Function | Unit |
|----------|---------------|
| fCp | J / (kg*degC) |

Carbon Steel (mat2)

Add the temperature as a model input for the Basic properties.

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Materials > Carbon Steel (mat2)** click **Basic (def)**.
- 2 In the **Settings** window for **Basic**, locate the **Model Inputs** section.
- 3 Click **+ Select Quantity**.
- 4 In the **Physical Quantity** dialog, type **temperature** in the text field.
- 5 In the tree, select **General > Temperature (K)**.
- 6 Click **OK**.
- 7 In the **Model Builder** window, click **Carbon Steel (mat2)**.
- 8 In the **Settings** window for **Material**, locate the **Material Contents** section.


9 In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|------------------------------------|--|--------|----------|-------------------------------------|
| Young's modulus | E | fE(T) | Pa | Young's modulus and Poisson's ratio |
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | fK(T) | W/(m·K) | Basic |
| Heat capacity at constant pressure | Cp | fCp(T) | J/(kg·K) | Basic |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | fA(T) | 1/K | Basic |

DEFINITIONS


Add the time history for the temperature of the water in the pipe.

Step 1 (step1)

- 1 In the **Definitions** toolbar, click  **More Functions** and choose **Step**.
- 2 In the **Settings** window for **Step**, type pipeWaterTemp in the **Function name** text field.
- 3 Locate the **Parameters** section. In the **Location** text field, type 1 [s].
- 4 In the **From** text field, type 280 [degC].
- 5 In the **To** text field, type 20 [degC].

Add the time history for the temperature of the water in the pressure vessel.

Interpolation 1 (int1)

- 1 In the **Definitions** toolbar, click  **Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type vesselWaterTemp.
- 4 In the table, enter the following settings:

| t | f(t) |
|------|------|
| 0 | 280 |
| 1800 | 20 |

5 Locate the **Units** section. In the **Argument** table, enter the following settings:


| Argument | Unit |
|----------|------|
| t | s |

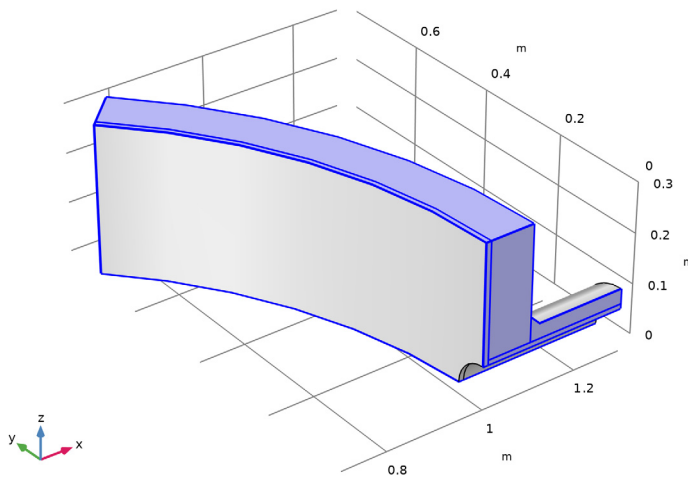
6 In the **Function** table, enter the following settings:

| Function | Unit |
|-----------------|------|
| vesselWaterTemp | degC |

Create an explicit selection to use in the symmetry boundary conditions.

Symmetry Boundaries

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry Boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 1, 3–5, 7, 8, 11, 13, 16, 17, 19, and 22 only.




HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1


- 1 In the **Model Builder** window, under **Component 1 (comp1) > Heat Transfer in Solids (ht)** click **Initial Values 1**.

- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the T text field, type 280[degC].


Symmetry 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.


Heat Flux 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 Select Boundaries 9, 20, and 23 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 From the **Flux type** list, choose **Convective heat flux**.
- 5 In the h text field, type 10.
- 6 In the T_{ext} text field, type 50[degC].

Heat Flux 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 Select Boundaries 10 and 15 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 From the **Flux type** list, choose **Convective heat flux**.
- 5 In the h text field, type 100.
- 6 In the T_{ext} text field, type pipeWaterTemp(t).

Heat Flux 3

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 From the **Flux type** list, choose **Convective heat flux**.
- 5 In the h text field, type 100.
- 6 In the T_{ext} text field, type vesselWaterTemp(t).

SOLID MECHANICS (SOLID)

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


Symmetry 1

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


- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.

Overwrite the extra boundaries of the explicit selection with another **Symmetry** node, but with free displacement as normal constraint.


Symmetry 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 4 and 8 only.
- 3 In the **Settings** window for **Symmetry**, click to expand the **Normal Direction Condition** section.
- 4 From the list, choose **Free displacement**.


Symmetry 3

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 24 and 25 only.
- 3 In the **Settings** window for **Symmetry**, locate the **Normal Direction Condition** section.
- 4 From the list, choose **Free displacement**.

Boundary Load 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 2, 10, and 15 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 `internalPressure`.


Boundary Load 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 4 and 8 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the \mathbf{f}_A vector as

| | |
|-----------------------|---|
| 0 | x |
| 0 | y |
| 4.76*internalPressure | z |

Now add a **Stress Linearization** node to compute the linearized stress in the vessel.

Stress Linearization 1

- 1 In the **Physics** toolbar, click  **Global** and choose **Stress Linearization**.
- 2 In the **Settings** window for **Stress Linearization**, locate the **Linearization** section.
- 3 From the **Type** list, choose **Distributed**.

Using the Distributed type, you can find the critical location. You can later on set the starting point of the stress linearization line.

- 4 Locate the **Boundary Selection** section. Click to select the **Activate Selection** toggle button.
- 5 Select Boundary 9 only.
- 6 Locate the **Domain Selection** section. Click to select the **Activate Selection** toggle button.
- 7 Select Domains 1 and 2 only.

MESH 1


Free Quad 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Quad**.
- 2 Select Boundary 19 only.

Size 1

- 1 Right-click **Free Quad 1** and choose **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.
- 5 Select the **Maximum element size** checkbox. In the associated text field, type 0.015.

Swept 1

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 4 only.

Distribution 1

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

Free Quad 2


- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Quad**.

2 Select Boundary 9 only.

Size 1

- 1 Right-click **Free Quad 2** and choose **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.
- 5 Select the **Maximum element size** checkbox. In the associated text field, type 0.05.


Swept 2

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.

Distribution 1

- 1 Right-click **Swept 2** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 From the **Distribution type** list, choose **Predefined**.
- 4 In the **Element ratio** text field, type 3.
- 5 Select the **Reverse direction** checkbox.


Swept 3

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 3 only.

Distribution 1


- 1 Right-click **Swept 3** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 2.

Swept 4


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

Distribution 1


- 1 Right-click **Swept 4** and choose **Distribution**.

- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 2.
- 4 Click  **Build Selected**.

STUDY 1: INITIALIZATION

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type Study 1: Initialization in the **Label** text field.
 In an Initialization study, the mechanical and thermal stationary state is computed. This serves the initial solution for a transient step.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.
- 4 In the **Study** toolbar, click  **Compute**.

ADD STUDY

- 1 In the **Study** toolbar, click  **Add Study** to open the **Add Study** window.
 In a second study, the transient temperature distribution is computed.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Solid Mechanics (solid)**.
- 5 Click the **Add Study** button in the window toolbar.



STUDY 2

Step 1: Time Dependent


- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Output times** text field, type range (0,0.2,2) range (60,60,1800).
 Analyze half an hour, storing the results once every minute. The first steps are refined in order to improve the convergence.
- 3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the **Method** list, choose **Solution**.
- 5 From the **Study** list, choose **Study 1: Initialization, Stationary**.

- 6 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study 1: Initialization, Stationary**.
- 9 In the **Model Builder** window, click **Study 2**.
- 10 In the **Settings** window for **Study**, type Study 2: Heat Transfer in the **Label** text field.
- 11 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
 Since cold water is suddenly injected in the vessel, it is important to capture the development accurately, so you want to enforce intermediate steps.
- 4 From the **Steps taken by solver** list, choose **Intermediate**.
- 5 In the **Study** toolbar, click  **Compute**.


ADD STUDY

- 1 Go to the **Add Study** window.
- 2 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies > Stationary**.
- 3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Heat Transfer in Solids (ht)**.
- 4 Click the **Add Study** button in the window toolbar.
 In a third study, the elastoplastic problem is computed using a stationary continuation study step.
- 5 In the **Study** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Stationary



- 1 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
- 2 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

- 3 From the **Method** list, choose **Solution**.
- 4 From the **Study** list, choose **Study 1: Initialization, Stationary**.
- 5 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 6 From the **Method** list, choose **Solution**.
- 7 From the **Study** list, choose **Study 2: Heat Transfer, Time Dependent**.
- 8 From the **Time (s)** list, choose **Automatic (all solutions)**.
- 9 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 10 Click  **Add**.
- 11 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|-----------------------------------|----------------|
| t (Time variable; used for stationary analysis) | range(0,0.2,2) range(60, 60,1800) | s |

- 12 In the table, click to select the cell at row number 1 and column number 3.
- 13 In the **Model Builder** window, click **Study 3**.
- 14 In the **Settings** window for **Study**, type Study 3: Plasticity in the **Label** text field.

Solution 3 (sol3)


- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.
- 3 In the **Model Builder** window, expand the **Study 3: Plasticity > Solver Configurations > Solution 3 (sol3) > Stationary Solver 1** node, then click **Fully Coupled 1**.
- 4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
Use a double dogleg solver in order to improve the convergence.
- 5 From the **Nonlinear method** list, choose **Double dogleg**.
- 6 In the **Study** toolbar, click  **Compute**.

RESULTS

Stress (solid)

Add a plot of the von Mises stress from **Result Templates**.

- 1 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
- 2 From the **View** list, choose **New view**.

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

Volume 1

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

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


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


Deformation

1 In the **Model Builder** window, expand the **Volume 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 0.

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

5 Click the  **Show Grid** button in the **Graphics** toolbar.

Add a plot of the equivalent plastic strain from **Result Templates**.

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 3: Plasticity/Solution 3 (sol3) > Solid Mechanics > Equivalent Plastic Strain (solid)**.

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

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

RESULTS

Equivalent Plastic Strain (solid)

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

2 From the **View** list, choose **View 3D 4**.

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

4 Locate the **Data** section. From the **Parameter value (t (s))** list, choose **0**.



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

6 From the **Parameter value (t (s))** list, choose **1200**.

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


Add a plot of the temperature in the vessel.

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 3: Plasticity/Solution 3 (sol3) > Heat Transfer in Solids > Temperature (ht)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS

Volume 1


- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 In the **Unit** field, type degC.
- 4 In the **Temperature (ht)** toolbar, click  **Plot**.

Temperature (ht)

- 1 In the **Model Builder** window, click **Temperature (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (t (s))** list, choose **0**.
- 4 In the **Temperature (ht)** toolbar, click  **Plot**.

The steps below investigate the bending and membrane stress in the vessel. First evaluate the maximum of the stress intensity for all time steps.

Stress Intensity, Maximum

- 1 In the **Results** toolbar, click  **Evaluation Group**.
- 2 In the **Settings** window for **Evaluation Group**, type Stress Intensity, Maximum in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Plasticity/Solution 3 (sol3)**.

Surface Maximum 1

- 1 Right-click **Stress Intensity, Maximum** and choose **Maximum > Surface Maximum**.
- 2 Select Boundary 9 only.
- 3 In the **Settings** window for **Surface Maximum**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1) >**

Solid Mechanics > Stress linearization > solid.Slmb - Stress intensity, membrane plus bending - N/m².

4 Click to expand the **Configuration** section. Select the **Include position** checkbox.

5 In the **Stress Intensity, Maximum** toolbar, click  **Evaluate**.

The evaluation of the stress intensity over the whole surface for all time steps can take a few minutes.

STRESS INTENSITY, MAXIMUM

1 Go to the **Stress Intensity, Maximum** window.

2 Click the **Table Graph** button in the window toolbar.

RESULTS

Stress Intensity, Maximum

1 In the **Model Builder** window, under **Results** click **ID Plot Group 5**.

2 In the **Settings** window for **ID Plot Group**, type Stress Intensity, Maximum in the **Label** text field.

Table Graph 1

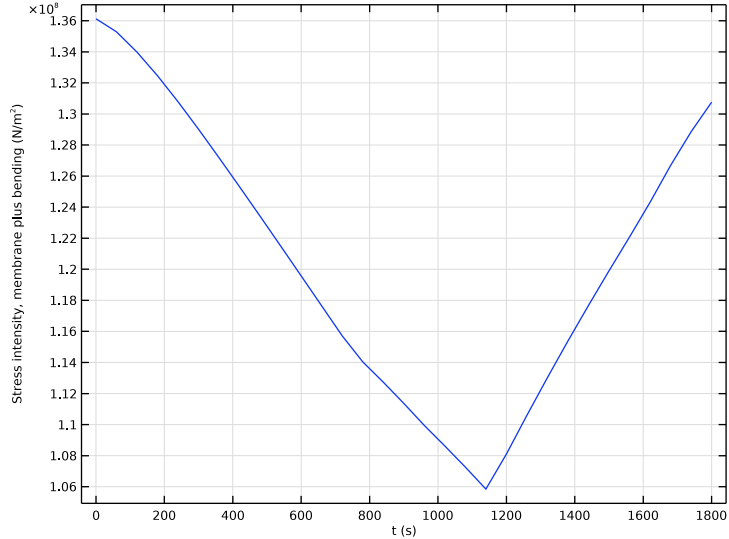
1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Plot columns** list, choose **Manual**.



4 In the **Columns** list box, select **Stress intensity, membrane plus bending (N/m²)**.

5 In the **Stress Intensity, Maximum** toolbar, click  **Plot**.



The plot shows that the maximum of the stress intensity has a dip just before 1200 s. Look at the distribution of the stress intensity at different times: start, end, and at the dip.

Stress Intensity


- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type **Stress Intensity** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Plasticity/ Solution 3 (sol3)**.
- 4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.
- 5 In the **Stress Intensity** toolbar, click  **Plot**.

Surface 1



- 1 Right-click **Stress Intensity** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Stress linearization > solid.Slmb - Stress intensity, membrane plus bending - N/m²**.

Marker 1

- 1 Right-click **Surface 1** and choose **Marker**.
- 2 In the **Settings** window for **Marker**, locate the **Display** section.

- 3 From the **Display** list, choose **Max**.
- 4 In the **Stress Intensity** toolbar, click  **Plot**.

Stress Intensity

- 1 In the **Model Builder** window, under **Results** click **Stress Intensity**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (t (s))** list, choose **1140**.
- 4 In the **Stress Intensity** toolbar, click  **Plot**.
- 5 From the **Parameter value (t (s))** list, choose **0**.
- 6 In the **Stress Intensity** toolbar, click  **Plot**.
Next, plot the stress linearization line at the critical location.
- 7 In the **Graphics** window, click on the maximum value marker. This automatically populates the **Evaluation 3D** table with the coordinates and the value of the plotted expression.
- 8 In the **Evaluation 3D** table, right-click on the newly added values and select **Copy Selection to Clipboard**.

SOLID MECHANICS (SOLID)

Stress Linearization I

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Solid Mechanics (solid)** click **Stress Linearization I**.
- 2 In the **Settings** window for **Stress Linearization**, click to expand the **Results** section.
- 3 Select the **Linearization line, starting point** table and press CTRL+V. This inserts the coordinates into the table.

The coordinates in the table should now be approximately as shown below.

| | |
|-------|---|
| 0.778 | X |
| 0.778 | Y |
| 0.298 | Z |

STUDY 3: PLASTICITY

Solution 3 (sol3)


- In the **Model Builder** window, under **Study 3: Plasticity > Solver Configurations** right-click **Solution 3 (sol3)** and choose **Solution > Update**.

RESULTS

Graph Plot Style 1

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results** and choose **Configurations > Graph Plot Style**.
- 3 In the **Settings** window for **Graph Plot Style**, locate the **Coloring and Style** section.
- 4 Find the **Line style** subsection. From the **Width** list, choose **2**.
- 5 Locate the **Legends** section. Find the **Include in automatic mode** subsection. Clear the **Point** checkbox.
- 6 Select the **Label** checkbox.


Stress Linearization 1 (solid)

- 1 In the **Model Builder** window, under **Results** click **Stress Linearization 1 (solid)**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Data** section.
- 3 From the **Time selection** list, choose **From list**.
- 4 In the **Parameter values (t (s))** list, choose **0** and **1800**.
- 5 Click to expand the **Style Configuration** section. From the **Configuration** list, choose **Graph Plot Style 1**.
- 6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.
- 7 In the **Stress Linearization 1 (solid)** toolbar, click  **Plot**.




Appendix — Geometry 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 > Thermal–Structure Interaction > Thermal Stress, Solid**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 Click  **Done**.

GEOMETRY I


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** list, choose **xz-plane**.
- 4 Locate the **Unite Objects** section. Clear the **Unite objects** checkbox.
- 5 Click  **Go to Plane Geometry**.

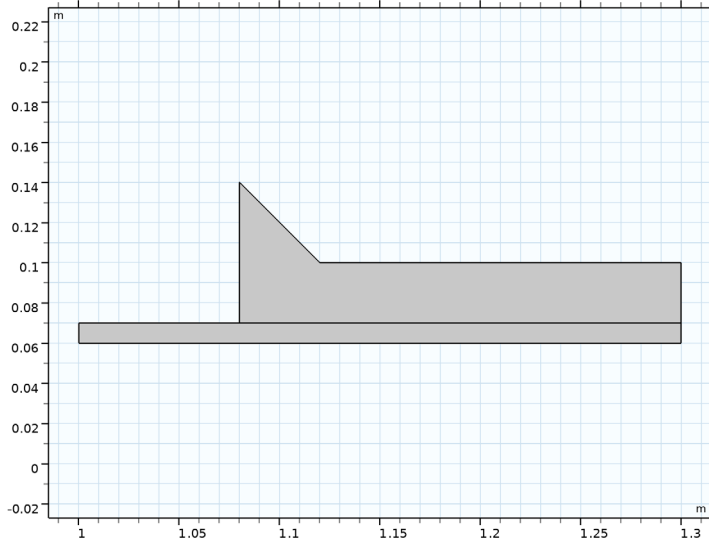
Work Plane 1 (wp1) > Polygon 1 (pol1)

- 1 In the **Work Plane** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 From the **Data source** list, choose **Vectors**.
- 4 In the **xw** text field, type 1.3 1.08 1.08 1.12 1.3.
- 5 In the **yw** text field, type 0.07 0.07 0.14 0.1 0.1.
- 6 Click  **Build Selected**.


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

- 1 In the **Work Plane** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.3.
- 4 In the **Height** text field, type 0.01.
- 5 Locate the **Position** section. In the **xw** text field, type 1.
- 6 In the **yw** text field, type 0.06.



7 Click  **Build Selected**.




Revolve 1 (rev1)


- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Axis** section.
- 3 Find the **Direction of revolution axis** subsection. In the **xw** text field, type 1.
- 4 In the **yw** text field, type 0.
- 5 Locate the **Revolution Angles** section. Click the **Angles** button.
- 6 In the **End angle** text field, type -90.
- 7 Click  **Build Selected**.

Work Plane 2 (wp2)



- 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** list, choose **xz-plane**.
- 4 Locate the **Unite Objects** section. Clear the **Unite objects** checkbox.
- 5 Click  **Go to Plane Geometry**.

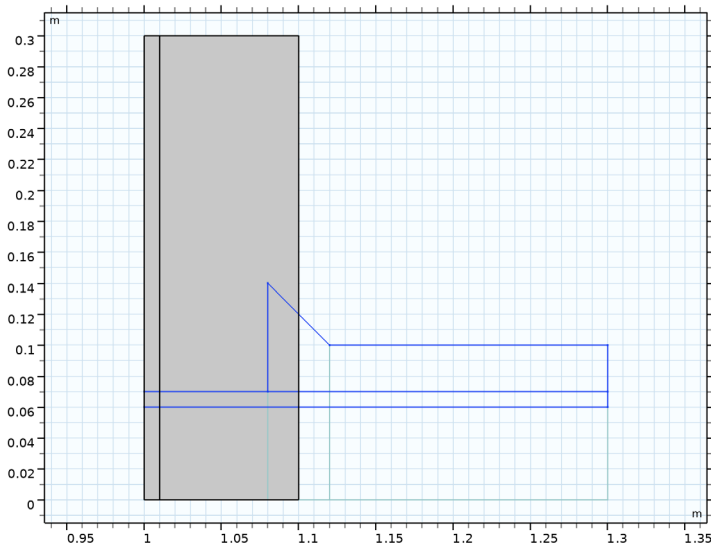
Work Plane 2 (wp2) > Rectangle 1 (r1)

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

- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.09.
- 4 In the **Height** text field, type 0.3.
- 5 Locate the **Position** section. In the **xw** text field, type 1.01.
- 6 Click  **Build Selected**.

Work Plane 2 (wp2) > Rectangle 2 (r2)


- 1 In the **Work Plane** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.01.
- 4 In the **Height** text field, type 0.3.
- 5 Locate the **Position** section. In the **xw** text field, type 1.
- 6 Click  **Build Selected**.



Revolve 2 (rev2)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
- 3 Click the **Angles** button.
- 4 In the **End angle** text field, type 45.

5 Click  **Build Selected**.

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

Difference 1 (dif1)

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

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

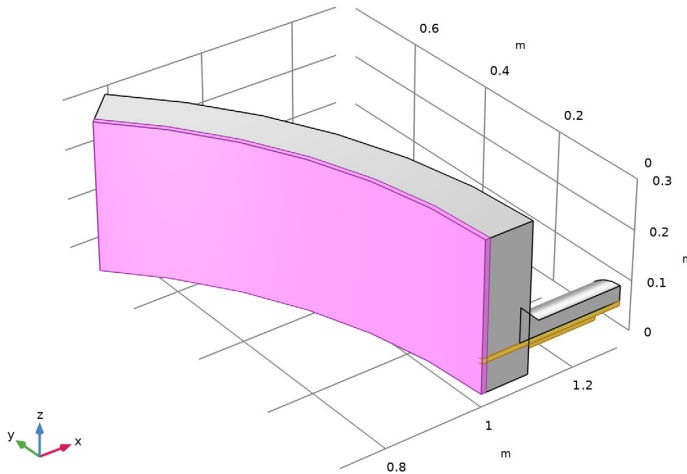
3 Select the **Keep objects to subtract** checkbox.

4 Clear the **Keep interior boundaries** checkbox.

5 Select the object **rev1(2)** only.

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

7 Select the object **rev2(2)** only.



8 Click  **Build Selected**.

Difference 2 (dif2)

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

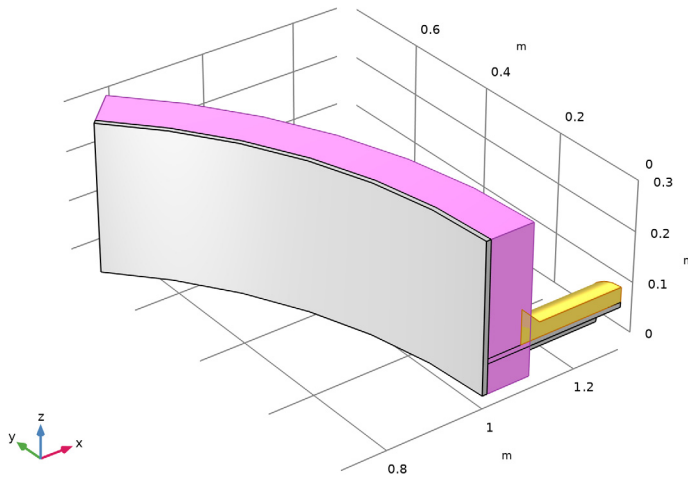
2 In the **Settings** window for **Difference**, locate the **Difference** section.

3 Select the **Keep objects to subtract** checkbox.

4 Select the object **rev1(1)** only.



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

6 Select the object **rev2(1)** only.





7 Click  **Build Selected**.

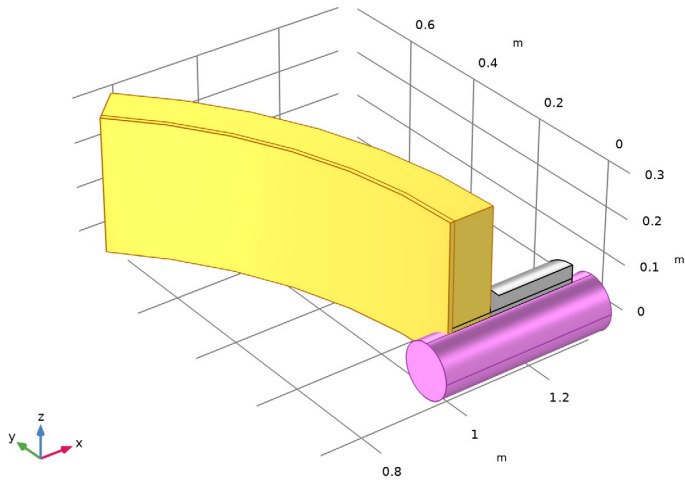
Cylinder 1 (cyl1)

- 1 In the **Geometry** toolbar, click  **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.06.
- 4 In the **Height** text field, type 0.4.
- 5 Locate the **Position** section. In the **x** text field, type 0.95.
- 6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
- 7 In the **x** text field, type 1.
- 8 In the **z** text field, type 0.
- 9 Click  **Build Selected**.

Difference 3 (dif3)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the objects **rev2(1)** and **rev2(2)** 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 **cyll** only.



6 Click  **Build Selected**.

Difference 4 (dif4)

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

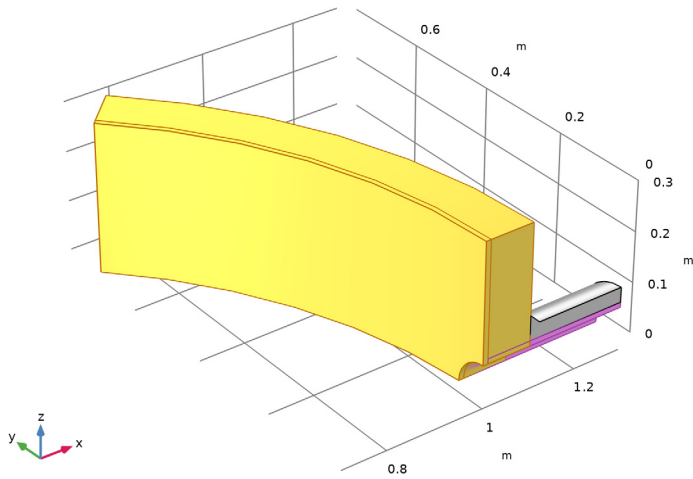
2 Select the object **dif3** only.

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 **dif1** only.

6 Select the **Keep objects to subtract** checkbox.



7 Click  **Build Selected**.

8 In the **Geometry** toolbar, click  **Build All**.

9 In the **Model Builder** window, click **Geometry 1**.

