Created in COMSOL Multiphysics 5.6

# Triaxial Test with Hardening Soil Material Model

## *Introduction*

In this example a triaxial soil test is simulated using a Hardening Soil material model. The test consists of two main stages: an initial isotropic compression followed by an axial compression or extension. The analysis can be simplified by considering the axial symmetry of the specimen.

The expected hyperbolic stress-strain relation is recovered by the simulation. It is also verified that the asymptotic value of the axial stress is equal to the analytical value of the failure stress, both in extension and in compression.

## *Model Definition*

In this triaxial test, a cylindrical soil specimen of 10 cm in diameter and height is loaded. Initially, a confinement pressure is applied to create a state of isotropic compression, and later the soil sample is compressed and extended axially to simulate either axial compression or extension.

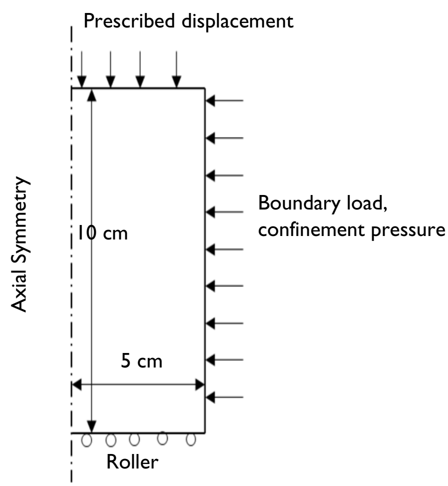


*Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.*

#### SOIL PROPERTIES

The soil properties for the Hardening Soil material model are

- Density $\rho = 2400\ \mathrm{kg/m^3}$, reference stiffness for primary loading, $E_{50}^{\mathrm{ref}} = 25$ MPa, reference stiffness for unloading and reloading $E_{\mathrm{ur}}^{\mathrm{ref}} = 75$ MPa, bulk modulus in compression $K_{\mathrm{c}} = 20$ MPa, stress exponent $m = 0.5$, and Poisson's ratio $\nu = 0.2$.
- Cohesion $c = 1$ kPa, angle of internal friction $\phi = 30°$, and dilatation angle $\psi = 10°$.
- Void ratio at reference pressure $e_{\mathrm{ref}} = 0.8$, reference pressure $p_{\mathrm{ref}} = 100$ kPa, and initial consolidation pressure $p_{\mathrm{c0}} = 1$ MPa.

#### CONSTRAINTS AND LOADS

- It is sufficient to only model the right half of the domain due to the axial symmetry.
- The stresses from the isotropic compression stage are considered as in-situ stresses, hence there is no need to model this stage explicitly. Instead a confinement pressure of 100 kPa is applied using the *in-situ stress* option in the **External Stress** node. Note that no boundary load is applied in this example.
- For axial compression or extension, the soil sample is compressed by applying a prescribed displacement to the top boundary. Allow the top-right corner to expand freely in the radial direction, and apply a roller boundary condition at the bottom boundary.

## *Results and Discussion*

The analytical solution to the axial failure stress for the soil specimen is given by the Mohr-Coulomb criterion

$$f = \frac{|\sigma_1 - \sigma_3|}{2} + \frac{|\sigma_1 + \sigma_3|}{2}\sin(\phi) - c\cos\phi \tag{1}$$

The axial failure stress in compression is given by

$$\sigma_1 = \sigma_3\frac{1+\sin\phi}{1-\sin\phi} - 2c\frac{(\cos\phi)}{1-\sin\phi}$$

Similarly, the axial failure stress in the extension is given by

$$\sigma_1 = \sigma_3\frac{1-\sin\phi}{1+\sin\phi} + 2c\frac{\cos\phi}{1+\sin\phi}$$

The values of the axial failure stress in compression and extension are shown in Table 1.

TABLE 1: FAILURE STRESS VALUES.

| Failure stress | Axial compression | Axial extension |
|---|---|---|
| $\sigma_1$ | -303.46 kPa | -32.18 kPa |

Note that for the sake of consistency with geomechanical convention, the compressive axial stress and strain are plotted along the positive axis, while the tensile stress and strain are plotted along the negative axis in all the figures.

The variation of axial stress versus axial strain in compression and extension can be seen in Figure 2. The stress-strain curve is hyperbolic, which is a characteristic of the Hardening Soil material model; as the axial displacement increases, the axial stress increases hyperbolically, and then asymptotically matches the corresponding failure stress given in the Table 1. The same behavior is observed for the von Mises stress in axial compression and extension; it asymptotically matches the ultimate deviatoric stress computed internally based on the Mohr-Coulomb criterion, see Figure 3.

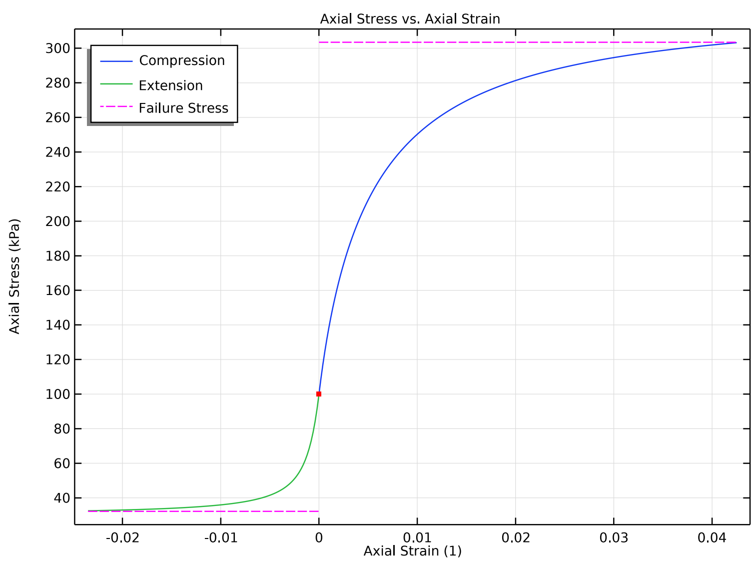


*Figure 2: Axial stress versus axial strain.*

.

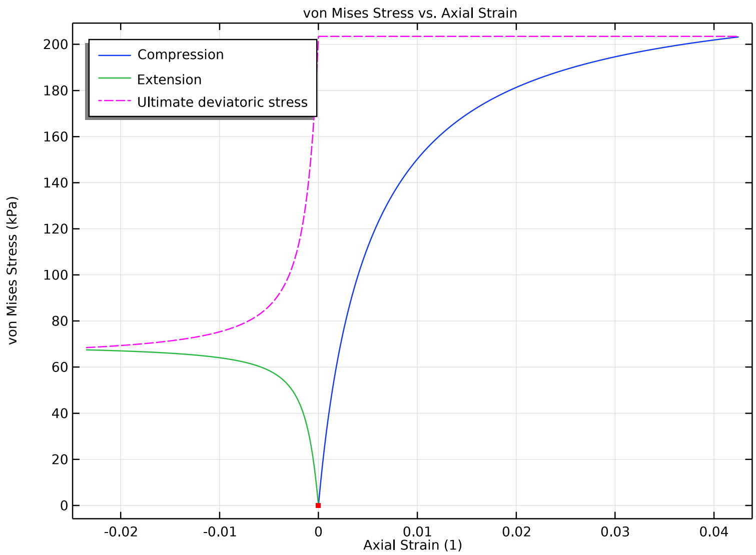


*Figure 3: von Mises stress versus axial strain.*

The variations of equivalent plastic strain and volumetric plastic strain are shown in Figure 4 and Figure 5, respectively. The equivalent plastic strain increases monotonically with the axial strain in the case of axial compression and extension, while the volumetric plastic strain first gradually decreases and then shows an increasing trend in both cases.

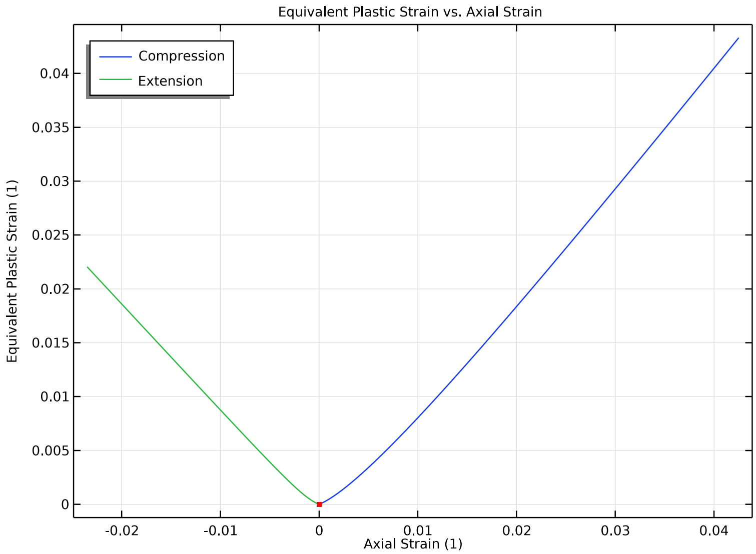


*Figure 4: Equivalent plastic strain versus axial strain.*

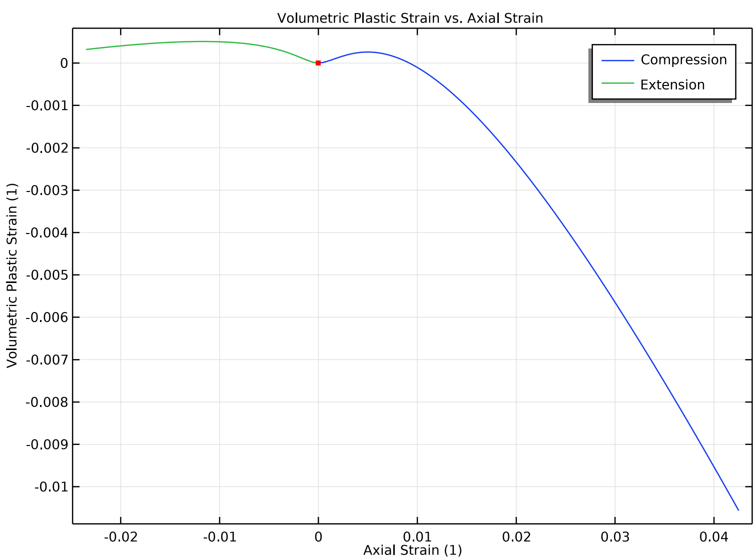


*Figure 5: Volumetric plastic strain versus axial strain.*

The dilatancy characteristics of soil describes its volumetric response to shearing. In the Hardening Soil model, Rowe's stress dilatancy theory is used, where the mobilized dilatancy angle $(\psi_m)$ is related to the critical state friction angle $(\phi_c)$ and the mobilized friction angle $(\phi_m)$:

$$\sin(\psi_m) = \frac{\sin(\phi_m) - \sin(\phi_c)}{1 - \sin(\phi_m)\sin(\phi_c)} \tag{2}$$

Many implementations of the Hardening Soil model use Equation 2 with a lower cutoff, but such an approach gives a bilinear function, see Ref. 1. In COMSOL Multiphysics, a scaled approach is used that provides a single nonlinear function instead:

$$\sin(\psi_m) = \left(\frac{\sin(\phi_m) - \sin(\phi_c)}{1 - \sin(\phi_m)\sin(\phi_c)}\right)\frac{\sin(\phi_m)}{\sin(\phi)} \tag{3}$$

When the mobilized dilatancy angle is plotted against the mobilized friction angle it gives a parabolic distribution in both cases; this result, shown in Figure 6, matches results shown in Ref. 1.

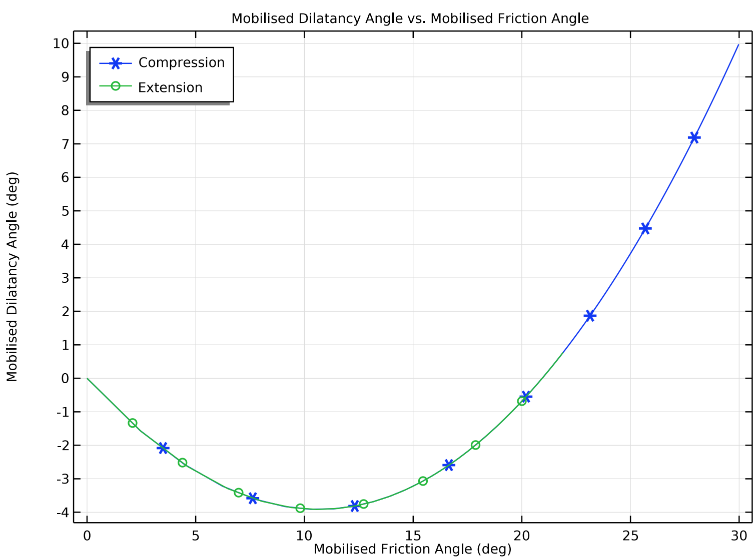


*Figure 6: Mobilized dilatancy angle versus mobilized friction angle.*

Figure 7 shows the variation of the Lode angle versus the axial strain. For axial compression the Lode angle is $\pi/3$, while for axial extension it is zero radians. This result verifies the established convention in COMSOL Multiphysics, stating that the Lode angle is $\pi/3$ when the stress state is on the compressive meridian and zero radians when the stress state is on the tensile meridian.

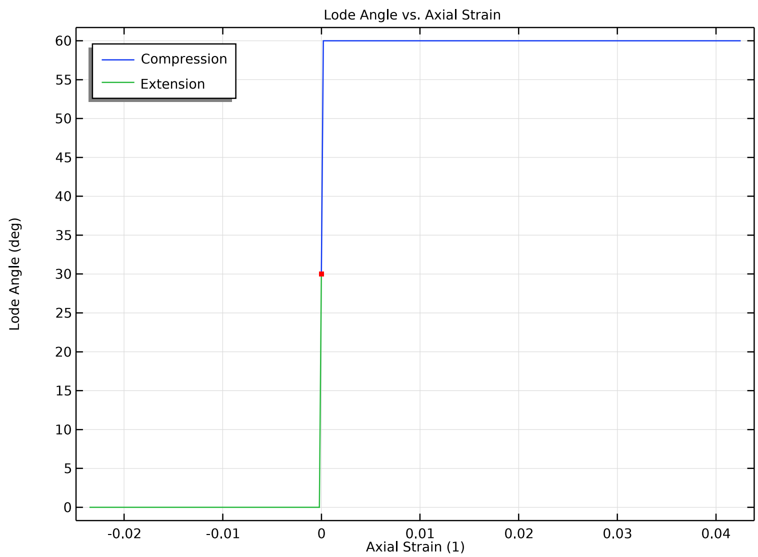


*Figure 7: Lode angle versus axial strain.*

## Notes About the COMSOL Implementation

The in-situ stresses are the stresses in the soil sample in strain-free configuration. There are two methods to account for in-situ stresses in COMSOL Multiphysics. One method is to create two stationary study steps or studies, with combination of **Initial Stress and Strain** and **External Stress** nodes. The second method is to use the *in-situ stress* option in the **External Stress** node with single study, which gives initial stresses in the soil sample without any strain. In this example, the second method is used to model the initial/in-situ stresses in the soil sample.

A **Prescribed Displacement** node is used to apply axial displacement on the top boundary in order to compress or extend the soil specimen. The axial compression and extension analysis does not converge for axial displacement values greater than 0.0425 m and

0.0235 m, respectively. These are the values at which the simulation fails as the axial stress approaches the failure stress.

## *Reference*

1. T. Bower, *Constitutive modelling of soils and fibre-reinforced soils*, PhD Thesis, Cardiff University, 2017.

**Application Library path:** `Geomechanics_Module/Verification_Examples/triaxial_test_hardening_soil`

## *Modeling Instructions*

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

### NEW

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

### MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

### GLOBAL DEFINITIONS

#### *Parameters 1*

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `triaxial_test_hardening_soil_parameters.txt`.

Define variables for the failure stress in axial compression and extension based on the Mohr-Coulomb criterion.

## DEFINITIONS

### *Variables 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| sigmafc | -p0*(1+sin(Phi))/(1-sin(Phi))-2*c*cos(Phi)/(1-sin(Phi)) | Pa | Failure stress in compression |
| sigmafe | -p0*(1-sin(Phi))/(1+sin(Phi))+2*c*cos(Phi)/(1+sin(Phi)) | Pa | Failure stress in extension |

## GEOMETRY 1

### *Rectangle 1 (r1)*

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

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

3 In the **Width** text field, type `5[cm]`.

4 In the **Height** text field, type `10[cm]`.

5 Click **Build Selected**.

## SOLID MECHANICS (SOLID)

### *Elastoplastic Soil Material 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Elastoplastic Soil Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Elastoplastic Soil Material**, locate the **Elastoplastic Soil Material** section.

4 From the **Material model** list, choose **Hardening Soil**.

5 From the $K_0^{\mathrm{nc}}$ list, choose **From angle of internal friction**.

**6** In the $p_{c0}$ text field, type `1000[kPa]`.

Apply a confinement pressure of 100 kPa using an **External Stress** node.

*External Stress 1*

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

**2** In the **Settings** window for **External Stress**, locate the **External Stress** section.

**3** From the **Stress input** list, choose **In-situ stress**.

**4** In the $\sigma_{ins}$ text field, type `-p0`.

*Roller 1*

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

**2** Select Boundary 2 only.

*Prescribed Displacement 1*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

**2** Select Boundary 3 only.

**3** In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

**4** Select the **Prescribed in z direction** check box.

**5** In the $u_{0z}$ text field, type `disp`.

## MATERIALS

*Soil Material*

**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 `Soil Material` in the **Label** text field.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Poisson's ratio | nu | `Nu` | 1 | Basic |
| Reference stiffness for primary loading | E50Ref | `E50ref` | Pa | Hardening Soil |
| Reference stiffness for unloading and reloading | EurRef | `Eurref` | Pa | Hardening Soil |

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Stress exponent | mH | `m` | 1 | Hardening Soil |
| Cohesion | cohesion | `c` | Pa | Mohr-Coulomb |
| Dilatation angle | psid | `Psi` | rad | Mohr-Coulomb |
| Bulk modulus in compression | Kc | `kc` | N/m² | Hardening Soil |
| Initial void ratio | evoid0 | `e0` | 1 | Hardening Soil |
| Angle of internal friction | internalphi | `Phi` | rad | Mohr-Coulomb |
| Density | rho | `Rho` | kg/m³ | Basic |

One mesh element is sufficient for this analysis.

### MESH 1

*Mapped 1*

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

*Distribution 1*

1. Right-click **Mapped 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.
4. Locate the **Distribution** section. In the **Number of elements** text field, type `1`.
5. Click **Build Selected**.

### STUDY: AXIAL COMPRESSION

Disable the default plots for this study.

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study: Axial Compression` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

*Step 1: Stationary*

1. In the **Model Builder** window, under **Study: Axial Compression** click **Step 1: Stationary**.

**2** In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

**3** Select the **Auxiliary sweep** check box.

**4** Click **Add**.

**5** In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| disp (Axial displacement) | `range(0,-0.00002,-0.004) range(-0.00402,-0.00001,-0.00425)` | `m` |

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

Add a second study for the axial compression case.

### ADD STUDY

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

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

**3** Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

### STUDY: AXIAL EXTENSION

Disable the default plots for this study.

**1** In the **Model Builder** window, click **Study 2**.

**2** In the **Settings** window for **Study**, type `Study: Axial Extension` in the **Label** text field.

**3** Locate the **Study Settings** section. Clear the **Generate default plots** check box.

#### *Step 1: Stationary*

**1** In the **Model Builder** window, click **Step 1: Stationary**.

**2** In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

**3** Select the **Auxiliary sweep** check box.

**4** Click **Add**.

**5** In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| disp (Axial displacement) | `range(0,0.00002,0.0015) range(0.00151,0.00001,0.00235)` | `m` |

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

### RESULTS

*Axial Stress vs. Axial Strain*

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

2 In the **Settings** window for **1D Plot Group**, type `Axial Stress vs. Axial Strain` in the **Label** text field.

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

4 In the **Title** text area, type `Axial Stress vs. Axial Strain`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Axial Strain (1)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Axial Stress (kPa)`.

9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

1 Right-click **Axial Stress vs. Axial Strain** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `-solid.SZZ`.

5 From the **Unit** list, choose **kPa**.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `-solid.eZZ`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

| Legends |
|---|
| Compression |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

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

| **Legends** |
|---|
| `Extension` |

*Point Graph 3*

**1** In the **Model Builder** window, under **Results>Axial Stress vs. Axial Strain** right-click **Point Graph 1** and choose **Duplicate**.

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

**3** In the **Expression** text field, type `-sigmafc`.

**4** Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

**5** From the **Color** list, choose **Magenta**.

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

| **Legends** |
|---|
| `Failure Stress` |

*Point Graph 4*

**1** Right-click **Point Graph 3** and choose **Duplicate**.

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

**3** From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

**4** Locate the **y-Axis Data** section. In the **Expression** text field, type `-sigmafe`.

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

*Annotation 1*

**1** In the **Model Builder** window, right-click **Axial Stress vs. Axial Strain** and choose **Annotation**.

**2** In the **Settings** window for **Annotation**, locate the **Position** section.

**3** In the **R** text field, type `0`.

**4** In the **Z** text field, type `100`.

**5** Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

**6** In the **Axial Stress vs. Axial Strain** toolbar, click **Plot**.

*von Mises Stress vs. Axial Strain*

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

2 In the **Settings** window for **1D Plot Group**, type `von Mises Stress vs. Axial Strain` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `von Mises Stress vs. Axial Strain`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Axial Strain (1)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `von Mises Stress (kPa)`.

9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

1 Right-click **von Mises Stress vs. Axial Strain** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.mises`.

5 From the **Unit** list, choose **kPa**.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `-solid.eZZ`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

| Legends |
|---|
| Compression |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

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

| Legends |
|---|
| Extension |

*Point Graph 3*

1 In the **Model Builder** window, under **Results>von Mises Stress vs. Axial Strain** right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Soil material properties>Hardening Soil>solid.epsm1.qf - Ultimate deviatoric stress - Pa**.

3 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 From the **Color** list, choose **Magenta**.

5 Click to expand the **Legends** section. In the table, enter the following settings:

| Legends |
| --- |
| `Ultimate deviatoric stress` |

*Point Graph 4*

1 Right-click **Point Graph 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

4 Locate the **Legends** section. Clear the **Show legends** check box.

*Annotation 1*

1 In the **Model Builder** window, right-click **von Mises Stress vs. Axial Strain** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **R** text field, type `0`.

4 In the **Z** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 In the **von Mises Stress vs. Axial Strain** toolbar, click **Plot**.

*Equivalent Plastic Strain vs. Axial Strain*

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

2 In the **Settings** window for **1D Plot Group**, type `Equivalent Plastic Strain vs. Axial Strain` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Equivalent Plastic Strain vs. Axial Strain`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Axial Strain (1)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Equivalent Plastic Strain (1)`.

9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

1 Right-click **Equivalent Plastic Strain vs. Axial Strain** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.epe`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `-solid.eZZ`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

| **Legends** |
| --- |
| `Compression` |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

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

| **Legends** |
| --- |
| `Extension` |

*Annotation 1*

1 In the **Model Builder** window, right-click **Equivalent Plastic Strain vs. Axial Strain** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **R** text field, type `0`.

4 In the **Z** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 In the **Equivalent Plastic Strain vs. Axial Strain** toolbar, click **Plot**.

*Volumetric Plastic Strain vs. Axial Strain*

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

2 In the **Settings** window for **1D Plot Group**, type `Volumetric Plastic Strain vs. Axial Strain` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Volumetric Plastic Strain vs. Axial Strain`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Axial Strain (1)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Volumetric Plastic Strain (1)`.

*Point Graph 1*

1 In the **Model Builder** window, right-click **Volumetric Plastic Strain vs. Axial Strain** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `-solid.epvol`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `-solid.eZZ`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

| Legends |
| --- |
| Compression |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

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

| **Legends** |
|---|
| `Extension` |

*Annotation 1*

**1** In the **Model Builder** window, right-click **Volumetric Plastic Strain vs. Axial Strain** and choose **Annotation**.

**2** In the **Settings** window for **Annotation**, locate the **Position** section.

**3** In the **R** text field, type `0`.

**4** In the **Z** text field, type `0`.

**5** Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

**6** In the **Volumetric Plastic Strain vs. Axial Strain** toolbar, click **Plot**.

*Mobilised Dilatancy Angle vs. Mobilised Friction Angle*

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

**2** In the **Settings** window for **1D Plot Group**, type `Mobilised Dilatancy Angle vs. Mobilised Friction Angle` in the **Label** text field.

**3** Locate the **Title** section. From the **Title type** list, choose **Manual**.

**4** In the **Title** text area, type `Mobilised Dilatancy Angle vs. Mobilised Friction Angle`.

**5** Locate the **Plot Settings** section. Select the **x-axis label** check box.

**6** In the associated text field, type `Mobilised Friction Angle (deg)`.

**7** Select the **y-axis label** check box.

**8** In the associated text field, type `Mobilised Dilatancy Angle (deg)`.

**9** Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

**1** In the **Model Builder** window, right-click **Mobilised Dilatancy Angle vs. Mobilised Friction Angle** and choose **Point Graph**.

**2** Select Point 4 only.

**3** In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Soil material properties>Hardening Soil>solid.epsm1.psim - Mobilized dilatancy angle**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.epsm1.psim*180/pi`.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Soil material properties>Hardening Soil>solid.epsm1.phim - Mobilized friction angle**.

6 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.epsm1.phim*180/pi`.

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

| **Legends** |
| --- |
| `Compression` |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

| **Legends** |
| --- |
| `Extension` |

6 In the **Mobilised Dilatancy Angle vs. Mobilised Friction Angle** toolbar, click **Plot**.

*Lode Angle vs. Axial Strain*

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

2 In the **Settings** window for **1D Plot Group**, type `Lode Angle vs. Axial Strain` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Lode Angle vs. Axial Strain`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Axial Strain (1)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Lode Angle (deg)`.

9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

1 In the **Model Builder** window, right-click **Lode Angle vs. Axial Strain** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.thetaL`.

5 From the **Unit** list, choose **°**.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `-solid.eZZ`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

| Legends |
|---|
| Compression |

*Point Graph 2*

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Axial Extension/Solution 2 (sol2)**.

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

| Legends |
|---|
| Extension |

*Annotation 1*

1 In the **Model Builder** window, right-click **Lode Angle vs. Axial Strain** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **R** text field, type `0`.

4 In the **Z** text field, type `30`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 In the **Lode Angle vs. Axial Strain** toolbar, click **Plot**.