Created in COMSOL Multiphysics 6.2

Triaxial Test with Hardening Soil Material Model

Introduction

In this example, a triaxial test on a cylindrical soil sample is simulated using a Hardening Soil material model and the results are compared with those presented in Ref. 1 and Ref. 2. The test consists of two stages: an initial isotropic compression is followed by an axial compression or extension. The analysis can be simplified by taking the axial symmetry of the specimen into account.

The expected hyperbolic stress–strain relation is recovered by the model. 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 as shown in Figure 1. First, the confinement pressure in terms of in situ stresses is applied to create a state of isotropic compression. Thereafter, the soil sample is either compressed or extended axially.

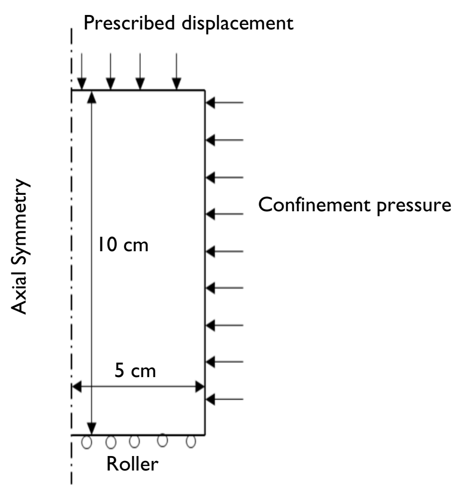

Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

SOIL PROPERTIES

Soil properties for loose Hostun sand are taken from Ref. 1 and Ref. 2. Note that some properties are different in Ref. 1 and Ref. 2. Along with properties given in the references, the properties used in the COMSOL Multiphysics model are presented in Table 1.

TABLE 1: MATERIAL PROPERTIES FOR THE SOIL MODEL.

Property	Variable	Value in Ref. 1	Value in Ref. 2	Value used in COMSOL
Poisson's ratio	ν	0.2	0.2	0.2
Density	ρ	NA	NA	2400 kg/m^3
Reference stiffness for primary loading	$E_{50}{}^{\mathrm{ref}}$	20 MPa	23.89 MPa	20 MPa
Reference stiffness for unloading and reloading	$E_{\mathrm{ur}}{}^{\mathrm{ref}}$	60 MPa	60 MPa	60 MPa
Stress exponent	m	0.65	0.65	0.65
Swelling to compression ratio	$K_{\mathrm{s}}/K_{\mathrm{c}}$	NA	1.75	1.75
Cohesion	c	0 kPa	0 kPa	1 kPa
Angle of internal friction	ϕ	34°	34°	34°
Dilatation angle	ψ	0°	1.5°	1.5°
Ellipse aspect ratio	R_{c}	NA	1.0428	1.0428
Failure ratio	R_{f}	0.9	0.95	0.95
Initial void ratio	e_0	0.89	NA	0.89
Reference pressure	p_{ref}	100 kPa	100 kPa	100 kPa
Initial consolidation pressure	p_{c0}	NA	NA	1000 kPa

CONSTRAINTS AND LOADS

- The stress resulting from isotropic compression is considered as *in situ* stress; therefore, there is no need to model this stage explicitly. Instead, a confinement pressure of 300 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

$$f = \frac{|\sigma_1 - \sigma_3|}{2} + \frac{|\sigma_1 + \sigma_3|}{2}\sin(\phi) - c\cos\phi = 0 \qquad (1)$$

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 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 2.

TABLE 2: FAILURE STRESS VALUES.

Failure stress	**Axial compression**	**Axial extension**
σ_1	-1064.9 kPa	-83.7 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 axial stress versus axial strain curves in compression and extension are shown 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 approaches the failure stress given in Table 2. The results presented for the axial compression case in Figure 2 also closely match the results presented in Ref. 1 (see Figure 6).

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. The results presented for the axial compression case in Figure 3 also closely match the results presented in Ref. 2 (see Figure 12).

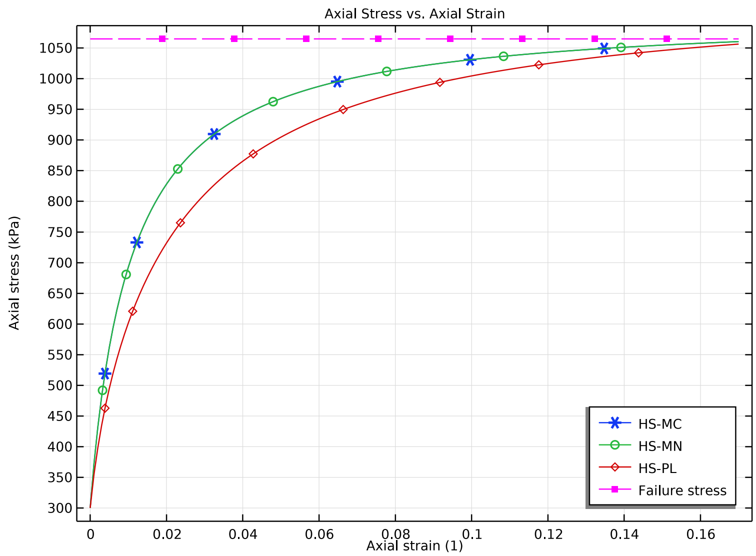

Figure 2: Axial stress versus axial strain.

.

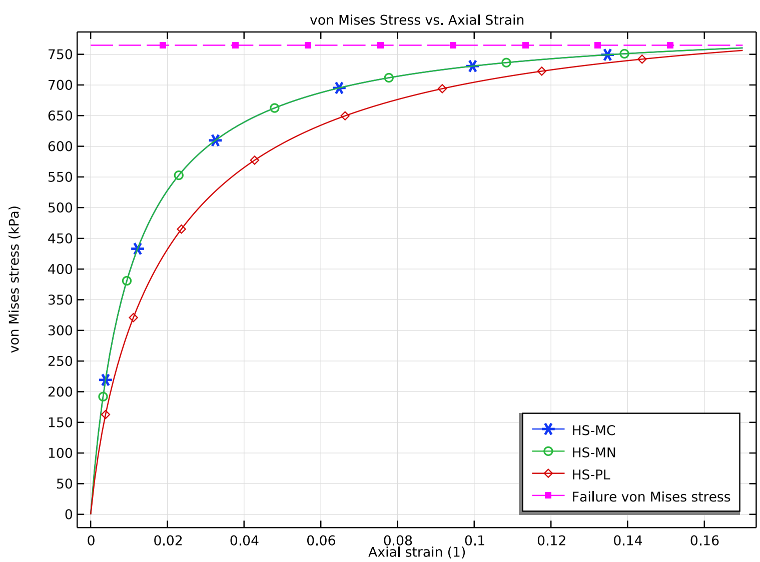

Figure 3: von Mises stress versus axial strain.

Figure 4 shows the variations in volumetric strain with applied axial strain. The volumetric strain shows parabolic behavior with respect to the axial strain in the case of axial compression, as shown in Ref. 2 (see Figure 13). The volumetric strain matches well with one of the experimental curves presented in the reference, but shows larger values than the theoretical results. The reason could be a different dilation angle (compared to Ref. 1) and a different mobilized dilatation angle formulation.

Figure 5 shows the variations in volumetric plastic strain with applied axial strain. The behavior is the same as for the volumetric strain presented in Figure 4.

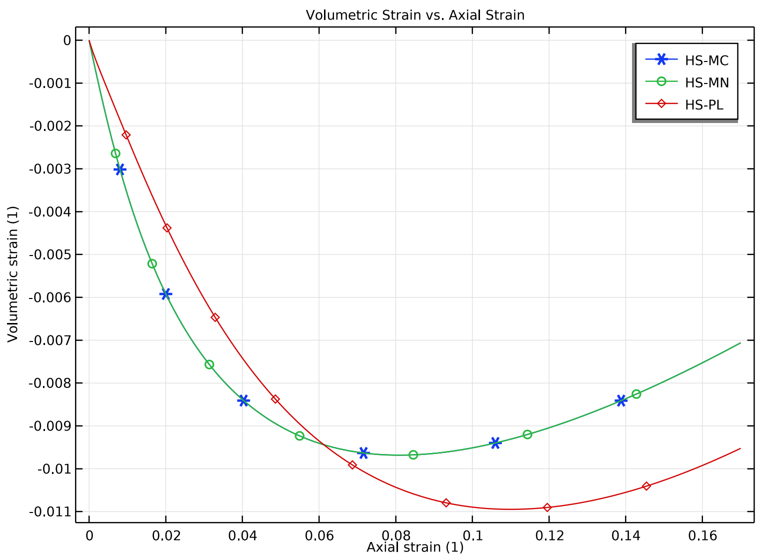

Figure 4: Volumetric strain versus axial strain.

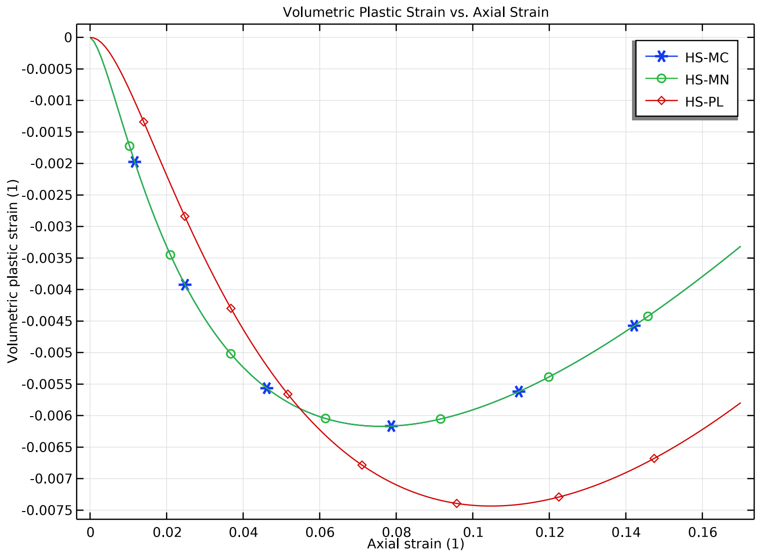

Figure 5: Volumetric plastic strain versus axial strain.

The dilatancy characteristics of a soil describes its volumetric response to shearing. For the Hardening Soil model presented in Ref. 1, Rowe's stress dilatancy theory is used, where the mobilized dilatancy angle (ψ_m) is related to the critical state friction angle (ϕ_c) and the mobilized friction angle (ϕ_m):

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

This formulation comes with a lower cutoff of zero, which gives a bilinear function; see Ref. 1. This COMSOL Multiphysics model uses a scaled approach (Soreide), which provides a single nonlinear function:

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

When the mobilized dilatancy angle is plotted against the mobilized friction angle it gives a parabolic distribution in both cases; the result is shown in Figure 6.

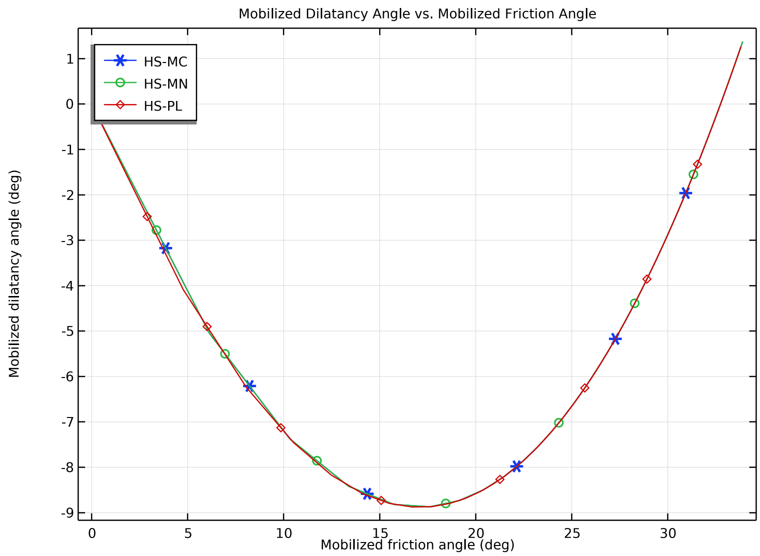

Figure 6: Mobilized dilatancy angle versus mobilized friction angle.

Notes About the COMSOL Implementation

The *in situ* stress is the stress in the soil sample in the 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, using a 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 a single study, which gives initial stresses in the soil sample without any strain. In this example, the second method is used to model the *in situ* stress in the soil sample.

References

1. T. Schanz, P.A. Vermeer, and P.G. Bonnier, "The Hardening Soil Model: Formulation and Verification," *Beyond 2000 in Computational Geotechnics*, Rotterdam, 1999.

2. T.A. Bower, P.J. Cleall, and A.D. Jefferson, "A Reformulated Hardening Soil Model," *Proceedings of the Institution of Civil Engineers — Engineering and Computational Mechanics*, vol. 173, no. 1, pp. 11–29, 2020.

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 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

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**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **z size** text field, type `3`.

5 Locate the **Displacement** section. In the **z** text field, type `20[cm]`.

6 Click **Build Selected**.

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

SOLID MECHANICS (SOLID)

Hardening Soil: Mohr–Coulomb

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

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil: Mohr-Coulomb` in the **Label** text field.

3 Select Domain 1 only.

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

5 From the K_c list, choose **From swelling to compression ratio**.

6 In the R_f text field, type `0.95`.

7 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 σ_{ins} text field, type `-p0`.

Hardening Soil: Mohr–Coulomb

In the **Model Builder** window, right-click **Hardening Soil: Mohr–Coulomb** and choose **Duplicate**.

Hardening Soil: Matsuoka–Nakai

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Hardening Soil: Mohr–Coulomb 1**.

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil: Matsuoka-Nakai` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domain 2 only.

5 Locate the **Elastoplastic Soil Material** section. From the **Failure criterion** list, choose **Matsuoka–Nakai**.

Hardening Soil: Mohr–Coulomb

In the **Model Builder** window, right-click **Hardening Soil: Mohr–Coulomb** and choose **Duplicate**.

Hardening Soil: Panteghini–Lagioia

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Hardening Soil: Mohr–Coulomb 1**.

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil: Panteghini-Lagioia` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domain 3 only.

5 Locate the **Elastoplastic Soil Material** section. From the **Failure criterion** list, choose **Panteghini–Lagioia**.

Roller 1

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

2 Select Boundaries 2, 5, and 8 only.

Prescribed Displacement 1

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

2 Select Boundaries 3, 6, and 9 only.

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

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	Young's modulus and Poisson's ratio
Reference failure stiffness	E50Ref	`E50ref`	Pa	Hardening Soil
Reference stiffness for unloading and reloading	EurRef	`Eurref`	Pa	Hardening Soil
Stress exponent	mH	`m`	1	Hardening Soil
Cohesion	cohesion	`c`	Pa	Mohr-Coulomb
Dilatation angle	psid	`Psi`	rad	Mohr-Coulomb
Swelling to compression ratio	rsc	`rc`	1	Hardening Soil

Property	Variable	Value	Unit	Property group
Ellipse aspect ratio	Rcap	Rc	1	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 1

Disable the default plots for this study.

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

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

3 Clear the **Generate default plots** check box.

Step 1: Stationary

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

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

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

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
disp (Axial displacement)	`range(0,-0.0001,-0.017)`	`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 **Label**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** check box. In the associated text field, type `Axial stress (kPa)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

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

2 Select Point 8 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 **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Number** text field, type `6`.

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

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

13 In the table, enter the following settings:

Legends
HS-MC

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 10 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 7.

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

Legends
HS-MN

8 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 12 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 8.

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

Legends
HS-PL

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

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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`

7 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 **Label**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** check box. In the associated text field, type `von Mises stress (kPa)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

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

2 Select Point 8 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 **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Number** text field, type `6`.

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

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

13 In the table, enter the following settings:

Legends
HS-MC

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 10 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 7.

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

Legends
HS-MN

8 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 12 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 8.

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

Legends
HS-PL

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

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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

3 In the **Expression** text field, type `-sigmafc-p0`.

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 von Mises stress`

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

Volumetric 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 Strain vs. Axial Strain` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** check box. In the associated text field, type `Volumetric strain (1)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Point Graph 1

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

2 Select Point 8 only.

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

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

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

8 From the **Positioning** list, choose **Interpolated**.

9 In the **Number** text field, type 6.

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

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

12 In the table, enter the following settings:

Legends
HS-MC

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 10 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 7.

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

Legends
HS-MN

8 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 12 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 8.

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

Legends
HS-PL

8 In the **Volumetric 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 **Label**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** check box. In the associated text field, type `Volumetric plastic strain (1)`.

Point Graph 1

1 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 **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

9 In the **Number** text field, type `6`.

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

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

12 In the table, enter the following settings:

Legends
HS-MC

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 10 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `7`.

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

Legends
HS-MN

8 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 12 only.

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `8`.

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

Legends
HS-PL

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

Mobilized Dilatancy Angle vs. Mobilized 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 `Mobilized Dilatancy Angle vs. Mobilized Friction Angle` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Mobilized friction angle (deg)`.

6 Select the **y-axis label** check box. In the associated text field, type `Mobilized dilatancy angle (deg)`.

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

Point Graph 1

1 Right-click **Mobilized Dilatancy Angle vs. Mobilized Friction Angle** and choose **Point Graph**.

2 Select Point 8 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 - rad**.

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 - rad**.

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

7 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

9 In the **Number** text field, type `6`.

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

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

12 In the table, enter the following settings:

Legends
HS-MC

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 10 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.epsm2.psim*180/pi`.

7 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.epsm2.phim*180/pi`.

8 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `7`.

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

Legends
HS-MN

10 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 12 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.epsm3.psim*180/pi`.

7 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.epsm3.phim*180/pi`.

8 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `8`.

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

Legends
HS-PL

10 In the **Mobilized Dilatancy Angle vs. Mobilized Friction Angle** toolbar, click **Plot**.