Model created in COMSOL Multiphysics 6.4

Triaxial and Oedometer Test with Hardening Soil Material Model

Introduction

In this example, triaxial and oedometer tests on a cylindrical soil sample are simulated using a Hardening Soil material model.

The triaxial test consists of two stages: an initial isotropic compression is followed by an axial compression. The oedometer test only have an axial compression stage. The analysis can be simplified by taking the axial symmetry of the specimen into account. The results are compared with those presented in Ref. 1 and Ref. 2.

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. In the oedometer test, loading and unloading cycles are conducted to assess the robustness of the numerical implementation.

Model Definition

In this example, a cylindrical soil specimen of 10 cm in diameter and height is loaded as shown in Figure 1.

For the triaxial test, a confinement pressure is applied to create a state of isotropic compression. Thereafter, the soil sample is compressed axially.

For the oedometer test, the soil sample is compressed axially and constrained on the side.

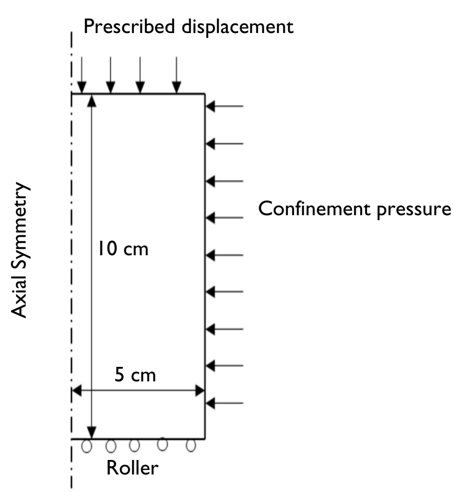

Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

SOIL PROPERTIES

The soil properties for loose Hostun sand are inspired from Ref. 1 and Ref. 2. Along with some properties given in the references, the material parameters are presented in Table 1.

TABLE 1: MATERIAL PROPERTIES FOR THE SOIL MODEL.

Property	Variable	Value in Ref. 1	Value in Ref. 2	Used value
Poisson's ratio	ν	0.2	0.2	0.2
Density	ρ	NA	NA	2400 kg/m^3
Reference initial stiffness for primary loading	$E_{\mathrm{i}}^{\mathrm{ref}}$	NA	68.913 MPa	68.913 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
Cohesion	c	0 kPa	0 kPa	0 kPa
Angle of internal friction	ϕ	34°	34°	34°
Dilatation angle	ψ	0°	1.5°	1.5°
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
Hardening modulus	K_{c}	NA	16.5 MPa	16.5 MPa
Initial ellipse centroid	p_{cc0}	NA	NA	2 kPa
Preconsolidation pressure	p_{c0}	NA	NA	5 kPa

CONSTRAINTS AND LOADS

- For the triaxial test, a confinement pressure of 300 kPa is applied using the **In situ stress** option in the **External Stress** node to model the isotropic compression step. For axial compression, the soil sample is compressed by applying a prescribed displacement on the top boundary. Allow the top boundary to expand freely in the radial direction and apply a roller boundary condition on the bottom boundary.
- For the oedometer test, the soil sample is compressed by applying a prescribed displacement on the top boundary. Apply roller boundary conditions at the vertical and bottom boundary.

The analytical solution to the axial failure stress for the triaxial test 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

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

The values of the axial failure stress are shown in Table 2.

TABLE 2: FAILURE STRESS VALUES.

Failure stress	**Axial compression**
σ_1	-1061.1 kPa

For the sake of consistency with the geomechanical convention, the compressive axial stress and axial strain are plotted along the positive axis in all figures.

The axial stress versus axial strain curves for the triaxial test are shown in Figure 2. The stress–strain curve has an hyperbolic shape, 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 therein).

The same behavior is observed for the von Mises stress in axial compression; it asymptotically matches the ultimate deviatoric stress 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 therein). The shear failure stress is reached at a compressive axial strain of 0.11, which is similar to the value reported in Ref. 2.

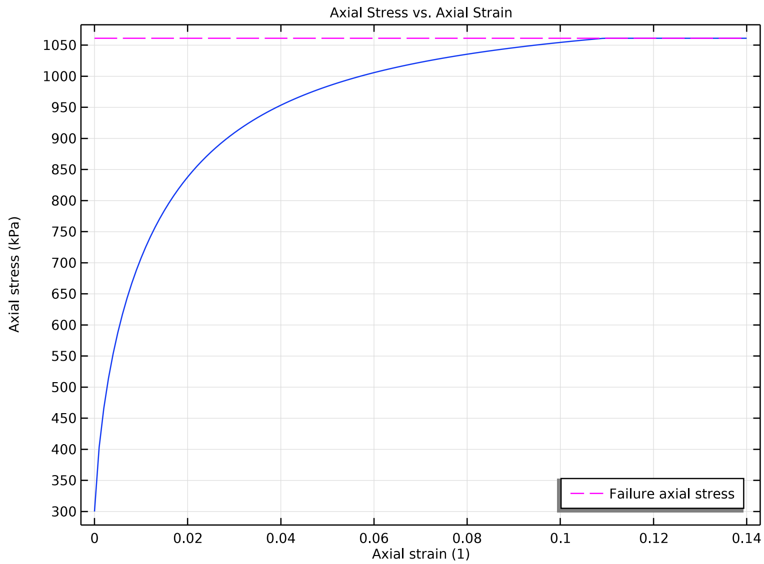

Figure 2: Axial stress versus axial strain for the triaxial test.

.

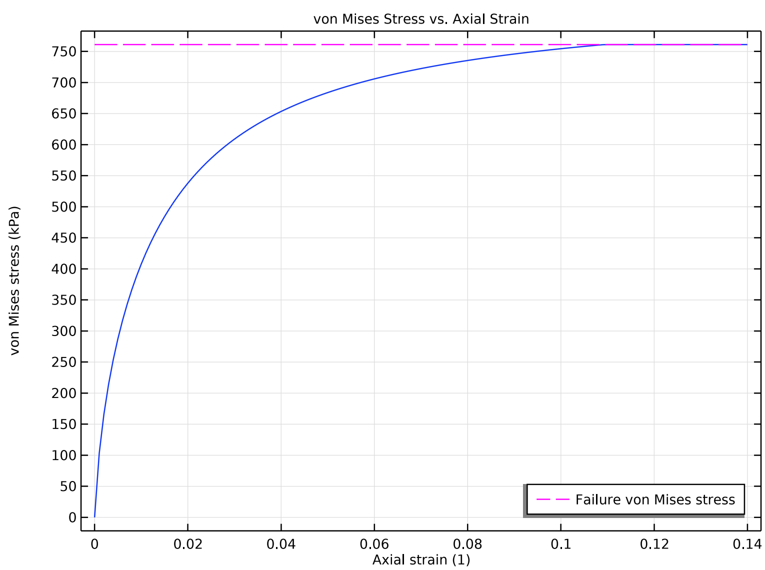

Figure 3: von Mises stress versus axial strain for the triaxial test.

Figure 4 shows the variations in volumetric strain versus applied axial strain. The volumetric strain shows parabolic behavior with respect to the axial strain, as shown in Ref. 2 (see Figure 13 therein). The volumetric strain matches well with the result presented in the Ref. 1. The volumetric strain is highly dependent on the mobilized dilatation angle formulation.

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:

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

The authors in Ref. 2 use the modified Rowe's formulation which is written as:

$$\sin(\psi_m) = \max\left(0, \frac{\sin(\phi_m) - \sin(\phi_c)}{1 - \sin(\phi_m)\sin(\phi_c)}\right) \tag{3}$$

This example uses Rowe's formulation (similar to Ref. 1), which gives a linear distribution with respect to the mobilized friction angle. The result is shown in Figure 6.

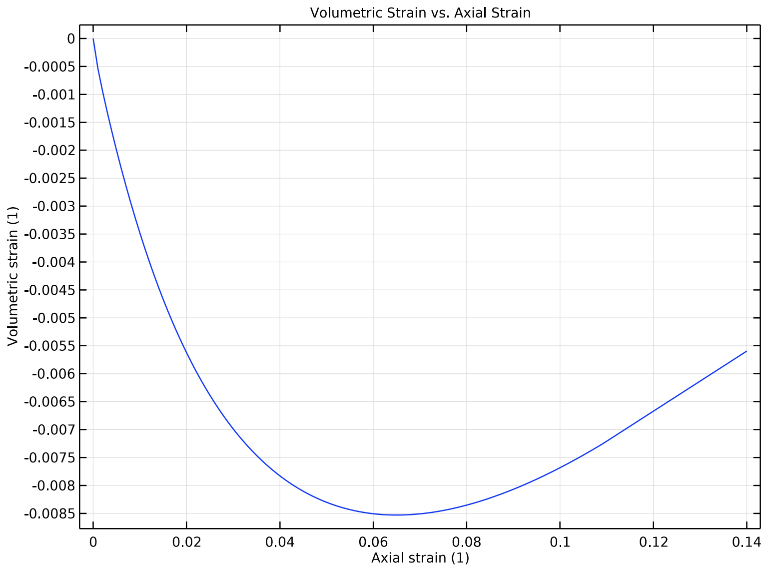

Figure 4: Volumetric strain versus axial strain in the triaxial test.

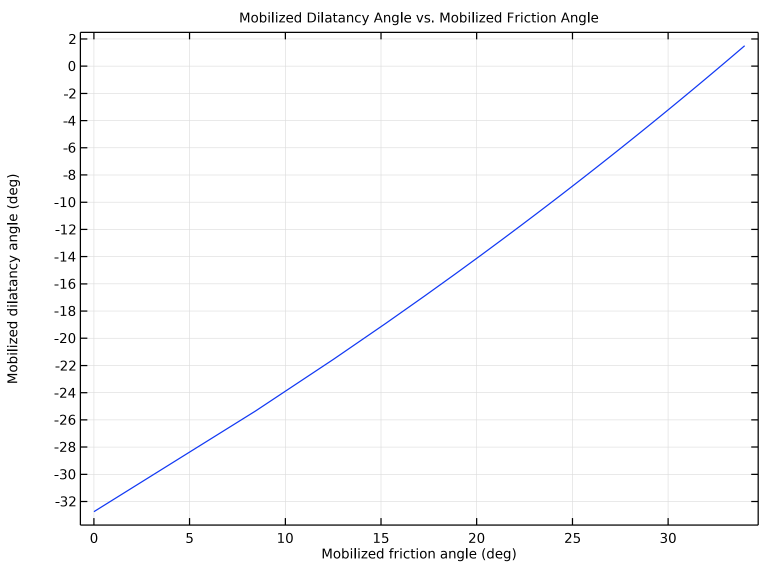

Figure 5: Mobilized dilatancy angle versus mobilized friction angle in the triaxial test.

The axial stress versus axial strain curves in the cyclic oedometer test are shown in Figure 6. The stress–strain curves for four different loading-unloading cycles presented in Figure 6 also match the results presented in Ref. 2 (see Figure 7 therein). Figure 7 shows the linear variation in volumetric strain with applied axial strain.

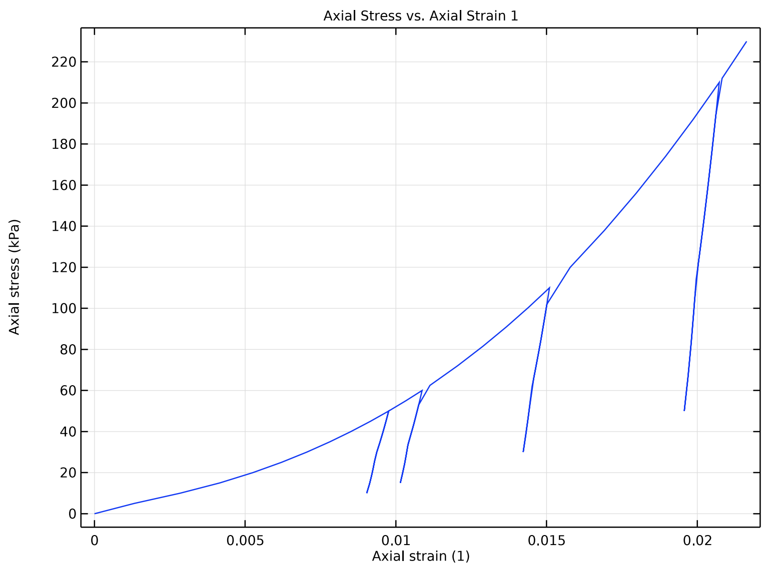

Figure 6: Axial stress versus axial strain in the oedometer test.

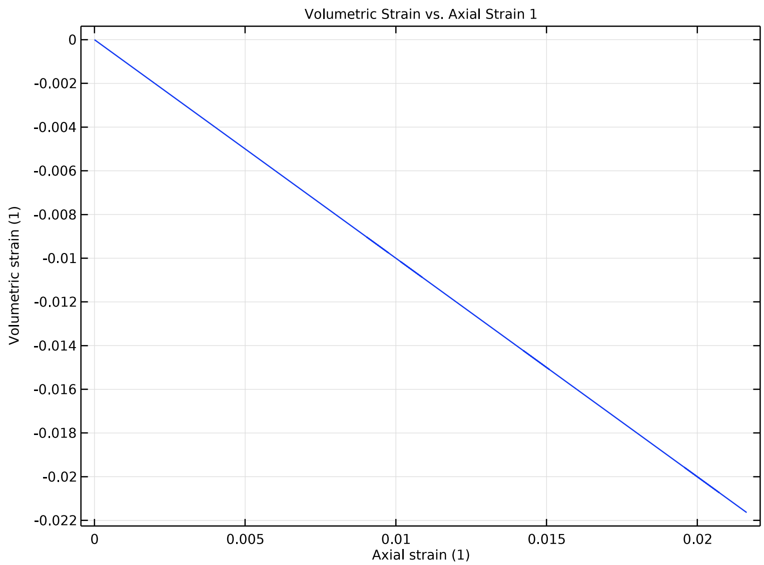

Figure 7: Volumetric strain versus axial strain in the oedometer test.

Notes About the COMSOL Implementation

The *in situ* stress is the stress in the soil sample represented 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, 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 samples.

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.

3. T. Forrister, "Analyzing Triaxial Testing Methods for Geomechanics," COMSOL Blog, 5 Mar. 2018, www.comsol.com/blogs/analyzing-triaxial-testing-methods-for-geomechanics/.

Application Library path: `Geomechanics_Module/Verification_Examples/triaxial_and_oedometer_test_hs`

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_and_oedometer_test_hs_parameters.txt`.

Pressure

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global** > **Interpolation**.
2 In the **Settings** window for **Interpolation**, type `Pressure` in the **Label** text field.
3 Locate the **Definition** section. In the **Function name** text field, type `P`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	50
2	10
3	60
4	15
5	110
6	30
7	210
8	50
9	230

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

Argument	Unit
t	1

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

Function	Unit
P	kPa

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**.

6 Click **Build Selected**.

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

SOLID MECHANICS (SOLID)

Hardening Soil

1 In the **Physics** toolbar, click **Domains** and choose **Elastoplastic Soil Material**.

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil` 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 ψ_m list, choose **Rowe**.

6 Find the **Parameters** subsection. From the E_i^{ref} list, choose **From material**.

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

Apply a confinement pressure of 300 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`.

Add a **Cap and Cutoff** node.

Hardening Soil

In the **Model Builder** window, click **Hardening Soil**.

Cap and Cutoff 1

In the **Physics** toolbar, click **Attributes** and choose **Cap and Cutoff**.

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

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

Roller 2

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

2 Select Boundaries 2 and 4 only.

Boundary Load 1

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

2 Select Boundary 3 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 `P(para)`.

Prescribed Displacement 1, Roller 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)**, Ctrl-click to select **Roller 1** and **Prescribed Displacement 1**.

2 Right-click and choose **Group**.

Triaxial Test

In the **Settings** window for **Group**, type `Triaxial Test` in the **Label** text field.

Boundary Load 1, Roller 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)**, Ctrl-click to select **Roller 2** and **Boundary Load 1**.

2 Right-click and choose **Group**.

Oedometer Test

In the **Settings** window for **Group**, type `Oedometer Test` in the **Label** text field.

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	Critical state model
Friction angle	internalphi	`Phi`	rad	Critical state model
Dilatation angle	psid	`Psi`	rad	Critical state model
Density	rho	`Rho`	kg/m³	Basic
Reference initial stiffness for primary loading	EiRef	`Eiref`	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	Hardening soil
Initial void ratio	evoid0	`e0`	1	Critical state model
Preconsolidation pressure	pcp0	`Pcp0`	Pa	Cap and cutoff
Initial ellipse centroid	pcc0	`Pcc0`	Pa	Cap and cutoff
Hardening modulus	Kc	`Kcc`	Pa	Cap and cutoff

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: TRIAXIAL TEST

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** checkbox.

4 In the **Label** text field, type `Study: Triaxial Test`.

Disable the features that are not needed in this study.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Hardening Soil** > **Cap and Cutoff 1** and **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Oedometer Test**.

5 Right-click and choose **Disable**.

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Axial displacement)	`range(0,-0.0001,-0.014)`	m

9 In the **Study** toolbar, click **Compute**.

Add a second study for the axial compression step in the oedometer test.

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 the **Add Study** button in the window toolbar.

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

Disable the features that are not needed in this study.

STUDY: OEDOMETER TEST

Disable the default plots for this study.

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

2 Clear the **Generate default plots** checkbox.

3 In the **Label** text field, type `Study: Oedometer Test`.

Disable the features that are not needed in this study.

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

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Hardening Soil** > **External Stress 1** and **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Triaxial Test**.

5 Right-click and choose **Disable**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.1,9)`	1

9 In the **Study** toolbar, click **Compute**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	kPa

8 Click **Apply**.

Axial Stress vs. Axial Strain

1 In the **Results** toolbar, click **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** checkbox. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** checkbox. 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 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 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

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

Point Graph 2

1 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 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

8 In the table, enter the following settings:

Legends
`Failure axial stress`

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

von Mises Stress vs. Axial Strain

1 In the **Results** toolbar, click **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** checkbox. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** checkbox. 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 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 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

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

Point Graph 2

1 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-p0`.

4 Locate 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. Select the **Show legends** checkbox.

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

8 In the table, enter the following settings:

Legends
`Failure von Mises stress`

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

Volumetric Strain vs. Axial Strain

1 In the **Results** toolbar, click **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** checkbox. In the associated text field, type `Axial strain (1)`.

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

Point Graph 1

1 Right-click **Volumetric 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.evol`.

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

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

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

Mobilized Dilatancy Angle vs. Mobilized Friction Angle

1 In the **Results** toolbar, click **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** checkbox. In the associated text field, type `Mobilized friction angle (deg)`.

6 Select the **y-axis label** checkbox. 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 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.epm1.psim - Mobilized dilatancy angle - rad**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.epm1.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.epm1.phim - Mobilized friction angle - rad**.

6 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.epm1.phim*180/pi`.

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

Axial Stress vs. Axial Strain 1

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

Point Graph 2

1 In the **Model Builder** window, expand the **Axial Stress vs. Axial Strain 1** node.

2 Right-click **Point Graph 2** and choose **Delete**.

Axial Stress vs. Axial Strain 1

1 In the **Model Builder** window, under **Results** click **Axial Stress vs. Axial Strain 1**.

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

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

4 In the **Axial Stress vs. Axial Strain 1** toolbar, click **Plot**.

Volumetric Strain vs. Axial Strain 1

1. In the **Model Builder** window, right-click **Volumetric Strain vs. Axial Strain** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study: Oedometer Test/Solution 2 (sol2)**.
4. In the **Volumetric Strain vs. Axial Strain 1** toolbar, click **Plot**.

Axial Stress vs. Axial Strain, Mobilized Dilatancy Angle vs. Mobilized Friction Angle, Volumetric Strain vs. Axial Strain, von Mises Stress vs. Axial Strain

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Axial Stress vs. Axial Strain**, **von Mises Stress vs. Axial Strain**, **Volumetric Strain vs. Axial Strain**, and **Mobilized Dilatancy Angle vs. Mobilized Friction Angle**.
2. Right-click and choose **Group**.

Triaxial Test

In the **Settings** window for **Group**, type `Triaxial Test` in the **Label** text field.

Axial Stress vs. Axial Strain 1, Volumetric Strain vs. Axial Strain 1

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Axial Stress vs. Axial Strain 1** and **Volumetric Strain vs. Axial Strain 1**.
2. Right-click and choose **Group**.

Oedometer Test

In the **Settings** window for **Group**, type `Oedometer Test` in the **Label** text field.