Created in COMSOL Multiphysics 6.2

Triaxial Test with Hardening Soil Small Strain Material Model

Introduction

In this example, a triaxial test on a cylindrical soil sample is simulated using a Hardening Soil Small Strain material model and the results are compared with those presented in Ref. 1. Two versions of the Hardening Soil Small Strain model, differing by the selected failure criterion, are tested and compared with each other.

This example consists of a monotonic triaxial compression test and a cyclic triaxial test. In the monotonic triaxial compression test, the hyperbolic stress–strain relation is expected to be recovered by the model. In the cyclic triaxial test, the small strain stiffness and the hysteresis effects are recovered.

Model Definition

In both triaxial tests, 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* stress is applied to create a state of isotropic compression. Thereafter, the soil sample is compressed axially in the monotonic test, while axially compressed and extended repeatedly in the cyclic test.

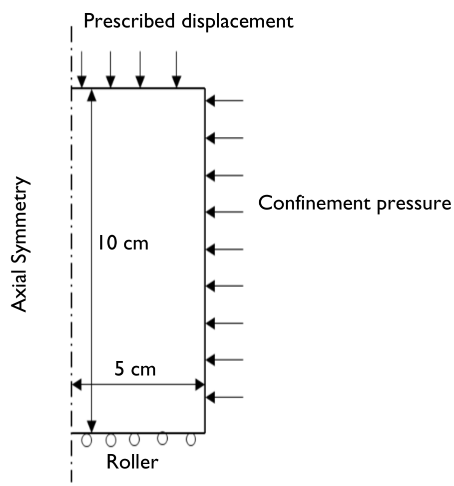

Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

Soil Properties

For the monotonic triaxial compression test, three different soils are used in Ref. 1: Hostun dense soil, Hostun loose soil, and Kaoln clay. The soil properties are presented in Table 1.

TABLE 1: MATERIAL PROPERTIES FOR THE SOIL MODELS.

Property	**Variable**	**Hostun dense**	**Hostun loose**	**Kaolin clay**
Poisson's ratio	ν	0.25	0.2	0.2
Density	ρ	2400 kg/m^3	2000 kg/m^3	1700 kg/m^3
Reference initial stiffness for primary loading	${E_\mathrm{i}}^\mathrm{ref}$	65.488 MPa	23.8 MPa	14.05 MPa
Reference stiffness for unloading and reloading	${E_\mathrm{ur}}^\mathrm{ref}$	90 MPa	60 MPa	11.5 MPa
Initial Young's modulus at reference pressure	${E_0}^\mathrm{ref}$	270 MPa	168 MPa	80 MPa
Stress exponent	m	0.55	0.75	0.8
Swelling to compression ratio	$K_\mathrm{s}/K_\mathrm{c}$	1.84	2.01	4.76
Cohesion	c	0 kPa	0 kPa	0 kPa
Angle of internal friction	ϕ	42°	34°	20°
Dilatation angle	tψ	16°	0°	0°
Ellipse aspect ratio	R_c	0.68027	0.64103	1.2821
Failure ratio	R_f	0.9	0.9	0.9
Initial void ratio	e_0	0.65	0.85	0.9
Reference pressure	p_ref	300 kPa	300 kPa	300 kPa
Initial consolidation pressure	p_c0	200 MPa	200 MPa	200 MPa

The cohesion is considered as zero in Ref. 1, but for better numerical convergence we consider it as small nonzero number. The density and initial consolidation pressure is not given in the Ref. 1 so we make some assumptions.

For the cyclic triaxial test, soil properties are taken from the Hostun dense soil except the initial shear modulus, which is assumed to be constant (G_0 = 190 MPa) and not derived from the initial shear modulus at reference pressure. This change is consistent with the cyclic test of the Small Strain Overlay model presented in Ref. 1, where the initial shear modulus is considered as constant.

CONSTRAINTS AND LOADS

- The stress resulting from the isotropic compression is considered as an *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 in the monotonic triaxial compression test, the soil sample is compressed by applying a prescribed displacement on 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.
- For axial compression and extension in the cyclic triaxial compression test, the soil sample is compressed or extended by applying a prescribed displacement on 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

In Ref. 1, the monotonic triaxial tests are carried out with three different confinement pressures: 100 kPa, 300 kPa, and 600 kPa. In the current example, the monotonic and cyclic triaxial tests are carried out with a confinement pressure of 300 kPa.

Figure 2 shows the axial stress versus axial strain in the monotonic triaxial test for three different soils. The stress-strain curve is hyperbolic, which is a characteristic of the Hardening Soil Small Strain material; as the axial displacement increases, the axial stress increases hyperbolically and approaches the failure stress. The results of the Hardening Soil Small Strain model with Mohr–Coulomb and Matsuoka–Nakai failure criteria match each other for all soil samples. Overall, the results match closely with the results presented in Ref. 1.

Figure 3 shows the variations in volumetric strain with applied axial strain for the monotonic triaxial test. The volumetric strain shows nonlinear behavior with respect to the axial strain. The volumetric strain of Hardening Soil model with the Mohr–Coulomb and Matsuoka–Nakai failure criteria match each other as the plastic potential and the mobilized dilatancy angle formulation are the same. The results show a similar trend as the numerical results presented in the Ref. 1, but the values differ slightly. The reason is that a different mobilized dilatation angle formulation is used compared to Ref. 1, which affects the plastic potential and the computed volumetric plastic strains. The Soreide mobilized dilatancy angle formulation is used in this example, because it is numerically more stable compared to the Rowe, modified Rowe, and Row–Li–Dafalias formulations used for both versions of the Hardening Soil Small Strain model in Ref. 1.

Figure 4 shows the axial stress versus axial strain in the cyclic triaxial test for the dense Hostun soil. The hysteresis and stiffness degradation effect is visible in the cyclic loading. The results are consistent with the results presented for the Hardening Soil Small Strain model in Ref. 1. The reason for the small difference in the results could be the use of different initial shear moduli as well the constant initial shear modulus throughout the cyclic test, which is consistent with the Small Strain Overlay cyclic test in Ref. 1. The stress–strain path in cyclic loading with the Small Strain Overlay model remains closed, but with the Hardening Soil Small strain model it does not close due to the occurrence of plastic strain, a behavior that is consistent with that shown in Ref. 1.

Figure 5 shows the variation of Young's modulus versus axial strain. The stiffness during primary loading and reloading are different and degrade with an increase in axial strain. This behavior is consistent with the behavior presented in Ref. 1.

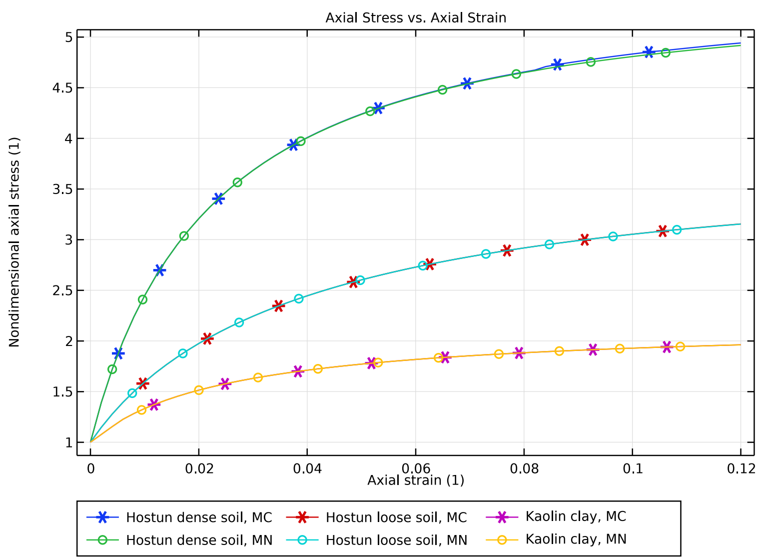

Figure 2: Axial stress versus axial strain in the monotonic triaxial test.

.

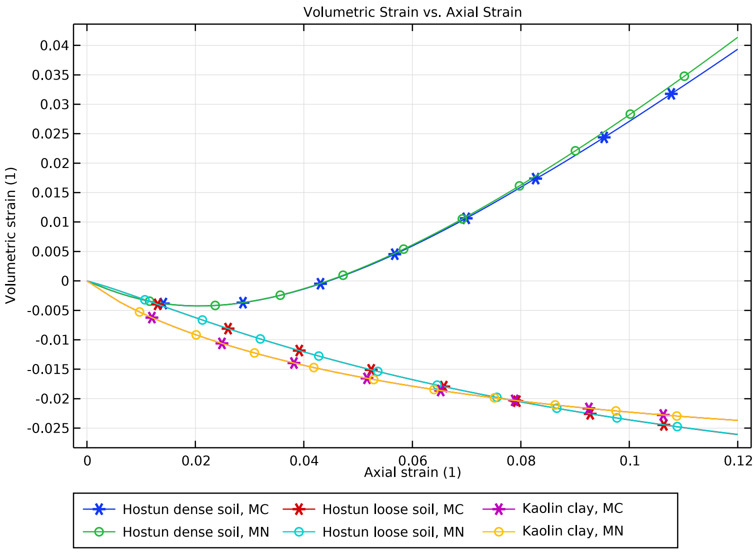

Figure 3: Volumetric strain versus axial strain in the monotonic triaxial test.

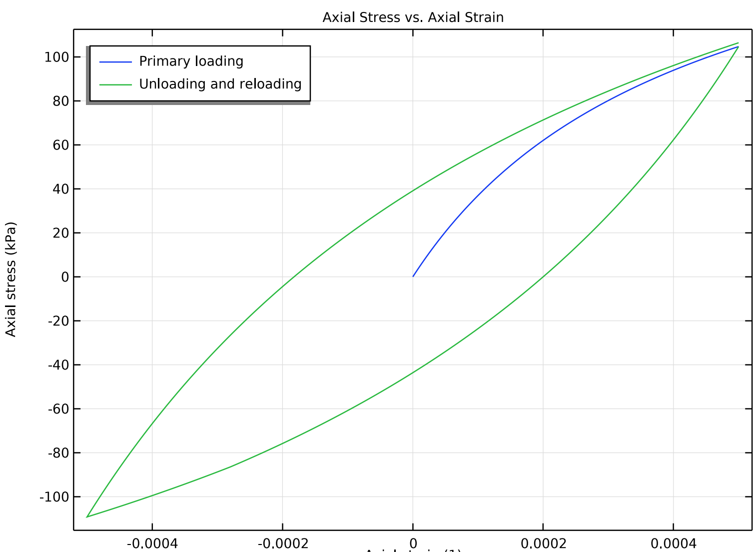

Figure 4: Axial stress versus axial strain in the cyclic triaxial test.

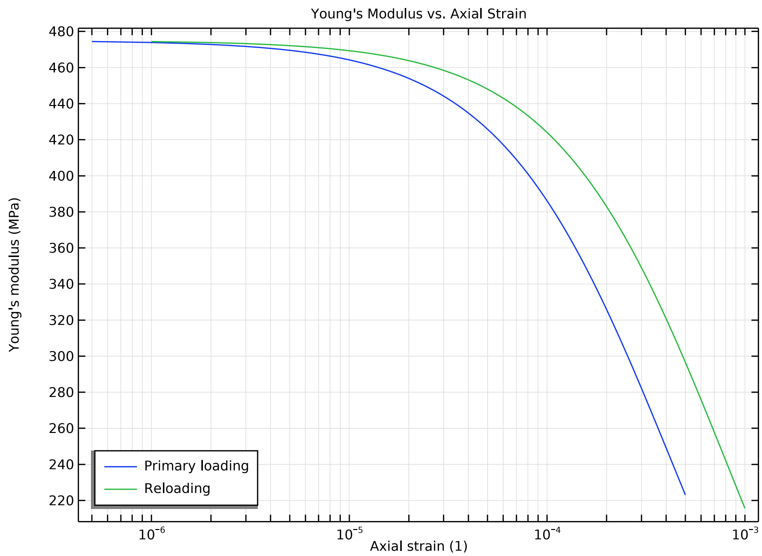

Figure 5: Young's modulus versus axial strain in the cyclic triaxial test.

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* stresses in the soil sample.

The Hardening Soil model comes with different flavors or versions; the model presented in Ref. 2 is known as the Hardening Soil-Original model, whereas the improved version presented in Ref. 1 is known as the Hardening Soil-Smooth model, and the model presented in Ref. 3 is known as the Hardening Soil-Lusas–Cardiff. The same models in COMSOL Multiphysics are differentiated based on the failure criterion used and known as the Mohr–Coulomb, Matsuoka–Nakai, and Panteghini–Lagioia failure criterion, respectively. Note that apart from the failure criterion these versions have other differences, too.

The Hardening Soil Small Strain model is a combination of the Hardening Soil model and Small Strain Overlay model, first proposed in Ref. 1. Similar to the Hardening Soil model, the Hardening Soil Small Strain model can be classified into three versions as Hardening Soil Small Strain-Original model, Hardening Soil Small Strain-Smooth model, and Hardening Soil Small Strain-Lusas–Cardiff. The same models in the COMSOL Multiphysics software are differentiated based on the failure criterion used and known as the Mohr–Coulomb, Matsuoka–Nakai, and Panteghini–Lagioia failure criterion, respectively. Note that apart from the failure criterion these versions have other differences, too.

References

1. T. Benz, *Small-Strain Stiffness of Soils and its Numerical Consequences*, PhD Dissertation, Stuttgart University, 2006.

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

3. 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_small_strain`

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. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 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 **Clear Table**.

4 In the table, enter the following settings:

Name	Expression	Value	Description
disp	0[cm]	0 m	Displacement parameter
para	0	0	Parameter
G_0	190[MPa]	1.9E8 Pa	Initial shear modulus for Hostun dense soil
a	0.385	0.385	Material parameter

Soil Properties

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Soil Properties` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `triaxial_test_hardening_soil_small_strain_dense_soil_parameters.txt`.

5 In the **Home** toolbar, click **Parameter Case**.

6 In the **Settings** window for **Case**, type `Hostun Dense Soil Properties` in the **Label** text field.

7 In the **Home** toolbar, click **Parameter Case**.

8 In the **Settings** window for **Case**, type `Hostun Loose Sand Properties` in the **Label** text field.

9 Locate the **Parameters** section. Click **Load from File**.

10 Browse to the model's Application Libraries folder and double-click the file `triaxial_test_hardening_soil_small_strain_loose_soil_parameters.txt`.

11 In the **Home** toolbar, click **Parameter Case**.

12 In the **Settings** window for **Case**, type `Kaolin Clay Properties` in the **Label** text field.

13 Locate the **Parameters** section. Click **Load from File**.

14 Browse to the model's Application Libraries folder and double-click the file `triaxial_test_hardening_soil_small_strain_clay_parameters.txt`.

DEFINITIONS

Interpolation 1 (int1)

1 In the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

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

3 In the **Function name** text field, type `appliedDisp`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	5e-3
2	-5e-3
3	5e-3

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

Function	Unit
appliedDisp	cm

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

Argument	Unit
t	1

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 `2`.

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.

Set up the first physics interface for the triaxial compression test and the second interface for the cyclic triaxial test.

SOLID MECHANICS [MONOTONIC]

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

2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics [Monotonic]` in the **Label** text field.

3 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Hardening Soil Small Strain: Mohr–Coulomb

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

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil Small Strain: 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 Small Strain**.

5 From the E_i^{ref} list, choose **From material**.

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

7 In the p_{ref} text field, type `p0`.

8 In the p_{c0} text field, type `200[MPa]`.

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`.

Hardening Soil Small Strain: Mohr–Coulomb

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

Hardening Soil Small Strain: Matsuoka–Nakai

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

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil Small Strain: 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**.

Roller 1

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

2 Select Boundaries 2 and 5 only.

Prescribed Displacement 1

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

2 Select Boundaries 3 and 6 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`.

SOLID MECHANICS [CYCLIC]

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics [Cyclic]` in the **Label** text field.

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

4 Select Domain 1 only.

5 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Hardening Soil Small Strain: Matsuoka–Nakai

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

For the cyclic test, set the shear modulus at small strains to be constant by applying the initial shear modulus of the domain material, G_0.

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Hardening Soil Small Strain: Matsuoka-Nakai` 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 Small Strain**.

5 From the **Failure criterion** list, choose **Matsuoka–Nakai**.

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

7 From the G_0 list, choose **From material**.

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

9 In the p_{ref} text field, type `p0`.

10 In the p_{c0} text field, type `200[MPa]`.

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`.

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 `-appliedDisp(para)`.

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 initial stiffness for primary loading	EiRef	`Eiref`	Pa	Hardening Soil
Reference stiffness for unloading and reloading	EurRef	`Eurref`	Pa	Hardening Soil
Reference initial shear modulus	G0Ref	`G0ref`	N/m²	Hardening Soil
Reference shear strain	gammaRef	`gammaR`	1	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
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

Property	Variable	Value	Unit	Property group
Density	rho	Rho	kg/m³	Basic
Initial shear modulus	G0	G_0	N/m²	Hardening Soil

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: MONOTONIC TRIAXIAL LOADING

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: Monotonic Triaxial Loading` in the **Label** text field.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 From the **Sweep type** list, choose **Parameter switch**.

4 Click **Add**.

5 In the table, enter the following settings:

Switch	Cases	Case numbers
Soil Properties	All	range(1,1,3)

Step 1: Stationary

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

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

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics [Monotonic] (solid)	√	Automatic (Stationary)
Solid Mechanics [Cyclic] (solid2)		Automatic (Stationary)

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,0.02,1.2)`	`cm`

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study: Monotonic Triaxial Loading>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 From the **Predictor** list, choose **Constant**.

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

Add a second study for the cyclic triaxial test.

ADD STUDY

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

STUDY: CYCLIC TRIAXIAL LOADING

Disable the default plots also for this study.

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

2 In the **Settings** window for **Study**, type `Study: Cyclic Triaxial Loading` 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: Cyclic Triaxial Loading** click **Step 1: Stationary**.

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

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics [Monotonic] (solid)		Automatic (Stationary)
Solid Mechanics [Cyclic] (solid2)	√	Automatic (Stationary)

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.001,3)`	1

Solution 6 (sol6)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Study: Cyclic Triaxial Loading>Solver Configurations>Solution 6 (sol6)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

5 From the **Predictor** list, choose **Constant**.

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

RESULTS

Axial Stress vs. Axial Strain (Monotonic)

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 (Monotonic)` 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.

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

7 Select the **y-axis label** check box. In the associated text field, type `Nondimensional axial stress (1)`.

8 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

9 From the **Position** list, choose **Bottom**.

10 In the **Number of rows** text field, type `2`.

Point Graph 1

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

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

3 From the **Dataset** list, choose **Study: Monotonic Triaxial Loading/ Parametric Solutions 1 (sol2)**.

4 From the **Soil Properties** list, choose **From list**.

5 In the **Soil Properties** list, select **Hostun Dense Soil Properties**.

6 Select Point 6 only.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `-solid.SZZ/p0`.

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

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

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

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

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

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

14 In the table, enter the following settings:

Legends
`Hostun dense soil, MC`

15 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 8 only.

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

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

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

Legends
`Hostun dense soil, MN`

Point Graph 1

In the **Model Builder** window, right-click **Point Graph 1** 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 **Data** section.

3 In the **Soil Properties** list, select **Hostun Loose Sand Properties**.

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

Legends
`Hostun loose soil, MC`

Point Graph 2

In the **Model Builder** window, right-click **Point Graph 2** 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 **Data** section.

3 In the **Soil Properties** list, select **Hostun Loose Sand Properties**.

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

Legends
`Hostun loose soil, MN`

Point Graph 3

In the **Model Builder** window, right-click **Point Graph 3** and choose **Duplicate**.

Point Graph 5

1 In the **Model Builder** window, click **Point Graph 5**.

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

3 In the **Soil Properties** list, select **Kaolin Clay Properties**.

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

Legends
`Kaolin clay, MC`

Point Graph 4

In the **Model Builder** window, right-click **Point Graph 4** and choose **Duplicate**.

Point Graph 6

1 In the **Model Builder** window, click **Point Graph 6**.

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

3 In the **Soil Properties** list, select **Kaolin Clay Properties**.

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

Legends
`Kaolin clay, MN`

Axial Stress vs. Axial Strain (Monotonic)

1 In the **Model Builder** window, click **Axial Stress vs. Axial Strain (Monotonic)**.

2 In the **Axial Stress vs. Axial Strain (Monotonic)** toolbar, click **Plot**.

3 Right-click **Axial Stress vs. Axial Strain (Monotonic)** and choose **Duplicate**.

Volumetric Strain vs. Axial Strain (Monotonic)

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

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

3 Locate the **Title** section. In the **Title** text area, type `Volumetric Strain vs. Axial Strain`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Volumetric strain (1)`.

Point Graph 1

1 In the **Model Builder** window, expand the **Volumetric Strain vs. Axial Strain (Monotonic)** node, then click **Point Graph 1**.

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

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

Point Graph 2

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

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

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

Point Graph 3

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

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

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

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 `solid.evol`.

Point Graph 5

1 In the **Model Builder** window, click **Point Graph 5**.

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

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

Point Graph 6

1 In the **Model Builder** window, click **Point Graph 6**.

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

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

4 In the **Volumetric Strain vs. Axial Strain (Monotonic)** toolbar, click **Plot**.

Axial Stress vs. Axial Strain (Cyclic)

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 (Cyclic)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Cyclic Triaxial Loading/ Solution 6 (sol6)**.

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

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

6 Locate the **Plot Settings** section.

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

8 Select the **y-axis label** check box. 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 (Cyclic)** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Study: Cyclic Triaxial Loading/Solution 6 (sol6)**.

4 From the **Parameter selection (para)** list, choose **Manual**.

5 In the **Parameter indices (1-3001)** text field, type `range(1,1,1001)`.

6 Select Point 6 only.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `-(solid2.Sl33+p0)`.

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

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

10 In the **Expression** text field, type `-solid2.el33`.

11 Locate 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
Primary loading

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 **Data** section.

3 In the **Parameter indices (1-3001)** text field, type `range(1001,1,3001)`.

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

Legends
Unloading and reloading

Axial Stress vs. Axial Strain (Cyclic)

1 In the **Model Builder** window, click **Axial Stress vs. Axial Strain (Cyclic)**.

2 In the **Axial Stress vs. Axial Strain (Cyclic)** toolbar, click **Plot**.

3 Right-click **Axial Stress vs. Axial Strain (Cyclic)** and choose **Duplicate**.

Young's Modulus vs. Axial Strain (Cyclic)

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

2 In the **Settings** window for **1D Plot Group**, type `Young's Modulus vs. Axial Strain (Cyclic)` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Young's Modulus vs. Axial Strain`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Young's modulus (MPa)`.

Point Graph 1

1 In the **Model Builder** window, expand the **Young's Modulus vs. Axial Strain (Cyclic)** node, then click **Point Graph 1**.

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

3 In the **Expression** text field, type `solid2.E`.

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

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid2.el33)`.

Point Graph 2

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

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

3 In the **Parameter indices (1-3001)** text field, type `range(2002,1,3001)`.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid2.E`.

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

6 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid2.el33-withsol('sol6',solid2.el33,setval(para,2)))`.

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

Legends
Reloading

Young's Modulus vs. Axial Strain (Cyclic)

1 In the **Model Builder** window, click **Young's Modulus vs. Axial Strain (Cyclic)**.

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

3 Select the **x-axis log scale** check box.

4 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

5 In the **Young's Modulus vs. Axial Strain (Cyclic)** toolbar, click **Plot**.