



Model created in COMSOL Multiphysics 6.4

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](#). 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.

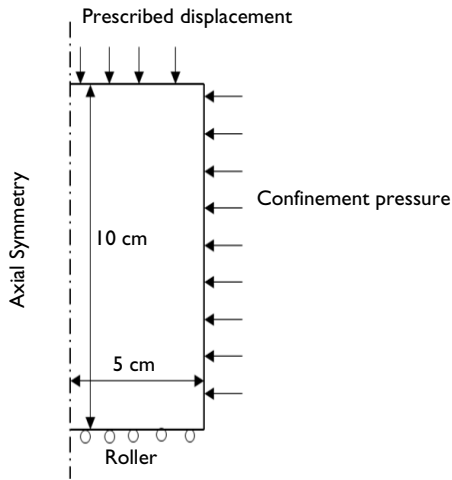


Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

SOIL PROPERTIES

For the monotonic triaxial compression test, three different soils are used in [Ref. 1](#): Hostun dense soil, Hostun loose soil, and Kaolin 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 failure stiffness	E_{50}^{ref}	30MPa	15MPa	3 MPa
Reference stiffness for unloading and reloading	$E_{\text{ur}}^{\text{ref}}$	90 MPa	60 MPa	11.5 MPa
Initial Young's modulus at reference pressure	E_0^{ref}	270 MPa	168 MPa	80 MPa
Stress exponent	m	0.55	0.75	0.8
Cohesion	c	1 kPa	1 kPa	1 kPa
Angle of internal friction	ϕ	42°	34°	20°
Dilatation angle	$\tau\psi$	16°	0°	0°
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}	100 kPa	100 kPa	100 kPa

The reference failure stiffness and cohesion values are slightly altered compared to the [Ref. 1](#) for better numerical convergence. The density is not given in the [Ref. 1](#) so we make some assumption.

For the cyclic triaxial test, soil properties are taken from the Hostun dense soil except the initial shear modulus and reference pressure. The initial shear modulus 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. The reference pressure is considered as 300 kPa same as the *in situ* stresses.

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 Matsuoka–Nakai octahedral shape matches well 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 Matsuoka–Nakai octahedral shape matches well with results presented in the [Ref. 1](#) for Hostun dense and loose sand, but results differ for Kaolin clay. 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.

[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 in [Ref. 1](#). Note that in this example, a different initial shear moduli as well the reference pressure is used than [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.

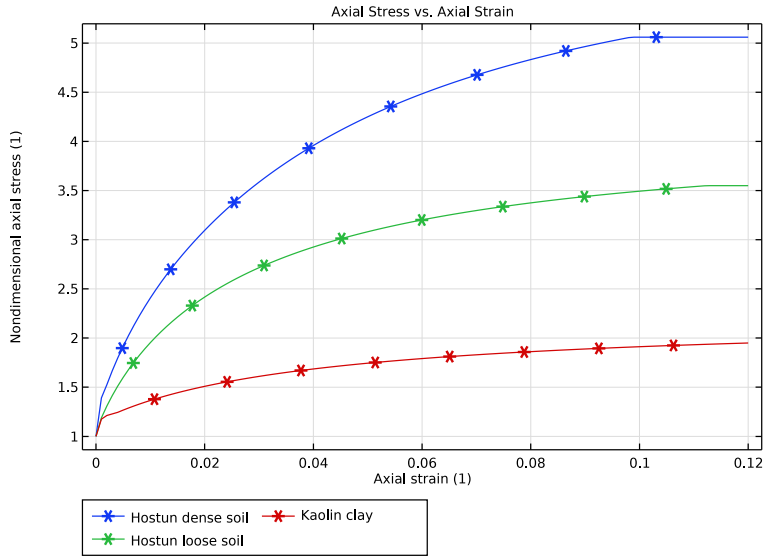


Figure 2: Axial stress versus axial strain in the monotonic triaxial test.

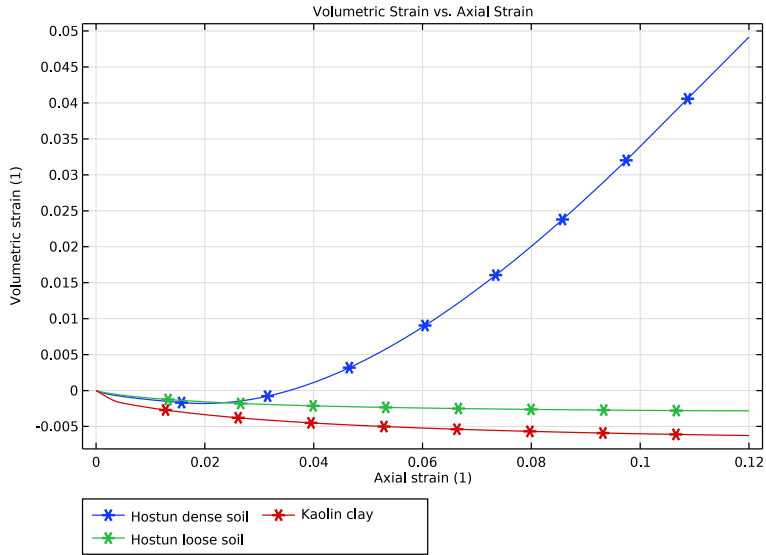


Figure 3: Volumetric strain versus axial strain in the monotonic triaxial test.

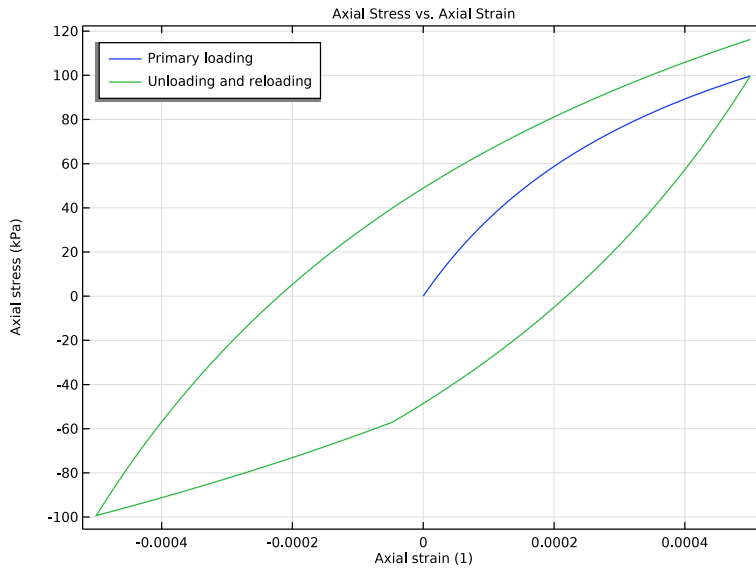


Figure 4: Axial stress versus axial strain in the cyclic triaxial test.

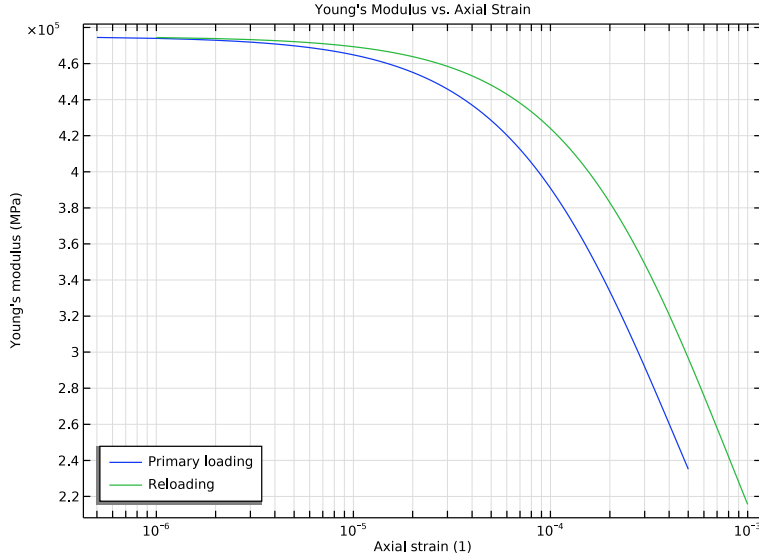


Figure 5: Young's modulus versus axial strain in the cyclic triaxial test.

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 Hardening Soil model in COMSOL Multiphysics is based on Ref. 3.

The Hardening Soil Small Strain model is a combination of the Hardening Soil model and Small-Strain Overlay model, first proposed in Ref. 1. The Hardening Soil and Hardening Soil Small Strain models in the COMSOL Multiphysics software are differentiated based

on the octahedral shape used. Three different octahedral shapes that can be used are Circular or Drucker–Prager, Mohr–Coulomb, and Matsuoka–Nakai.

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.
4. 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_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 I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 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 **Definitions** toolbar, click  **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
<code>appliedDisp</code>	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**.

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

- 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 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 $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.


- 6 From the ψ_m list, choose **Modified Rowe**.

For the monotonic triaxial test, set the **Load Reversal Points** to **None** to simplify the numerical computations and reduce computation time.


- 7 Locate the **Load Reversal Points** section. From the list, choose **None**.

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

Hardening Soil Small Strain


- 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 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 $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.
- 6 From the ψ_m list, choose **Modified Rowe**.
- 7 Find the **Parameters** subsection. From the G_0 list, choose **From material**.
- 8 In the p_{ref} text field, type p0.

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 I

- 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	l	Critical state model
Friction angle	internalphi	Phi	rad	Critical state model
Dilatation angle	psid	Psi	rad	Critical state model
Density	rho	Rho	kg/m ³	Basic
Reference failure stiffness	E50Ref	E50ref	Pa	Hardening soil
Reference stiffness for unloading and reloading	EurRef	Eurref	Pa	Hardening soil
Reference initial shear modulus	G0Ref	G0ref	N/m ²	Hardening soil
Reference shear strain	gammaRef	gammaR	l	Hardening soil
Stress exponent	mh	m	l	Hardening soil

Property	Variable	Value	Unit	Property group
Cohesion	cohesion	c	Pa	Hardening soil
Initial void ratio	evoid0	e0	l	Critical state model
Initial shear modulus	G0	G_0	N/m ²	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** checkbox.

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 **Study Settings** section.
- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type 0.0001.
- 5 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics [Cyclic] (solid2)**.
- 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 (Displacement parameter)	range(0,0.01,1.2)	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 **Model Builder** window, under **Study: Monotonic Triaxial Loading > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1** click **Fully Coupled 1**.
- 7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 8 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 9 In the **Study** toolbar, click  **Compute**.

Add a second study for the cyclic triaxial 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.

STUDY: CYCLIC TRIAXIAL LOADING

Disable the default plots also for this study.


- 1 In the **Settings** window for **Study**, type **Study: Cyclic Triaxial Loading** in the **Label** text field.
- 2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.


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 **Study Settings** section.
- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type **0.0001**.
- 5 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics [Monotonic] (solid)**.
- 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 (0,0.001,3)	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 **Model Builder** window, under **Study: Cyclic Triaxial Loading > Solver Configurations > Solution 6 (sol6) > Stationary Solver 1** click **Fully Coupled 1**.
- 7 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 8 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 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 > Young's modulus (Pa)** 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
Young's modulus	Pa	MPa

- 8 Click  **Apply**.

Axial Stress vs. Axial Strain (Monotonic)

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type Axial strain (1).
- 7 Select the **y-axis label** checkbox. 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 box, select **Hostun Dense Soil Properties**.
- 6 Select Point 4 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** checkbox.
- 13 From the **Legends** list, choose **Manual**.
- 14 In the table, enter the following settings:

Legends

Hostun dense soil

Point Graph 2

- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 In the **Soil Properties** list box, select **Hostun Loose Sand Properties**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

Legends

Hostun loose soil

Point Graph 3


- 1 Right-click **Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 In the **Soil Properties** list box, select **Kaolin Clay Properties**.

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

Legends

Kaolin clay

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**.

Volumetric Strain vs. Axial Strain (Monotonic)

- 1 Right-click **Axial Stress vs. Axial Strain (Monotonic)** and choose **Duplicate**.
- 2 In the **Settings** window for **ID 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.ev01`.


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.ev01`.

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.ev01`.
- 4 In the **Volumetric Strain vs. Axial Strain (Monotonic)** toolbar, click  **Plot**.

Axial Stress vs. Axial Strain (Cyclic)

- 1 In the **Results** toolbar, click  **ID Plot Group**.

- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type Axial strain (1).
- 8 Select the **y-axis label** checkbox. 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 (I-300I)** text field, type range (1, 1, 1001).
- 6 Select Point 4 only.
- 7 Locate the **y-Axis Data** section. In the **Expression** text field, type - (solid2.S133+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.e133.
- 11 Locate the **Legends** section. Select the **Show legends** checkbox.
- 12 From the **Legends** list, choose **Manual**.
- 13 In the table, enter the following settings:

Legends

Primary loading

Point Graph 2


- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 In the **Parameter indices (I-300I)** 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**.

Young's Modulus vs. Axial Strain (Cyclic)

- 1 Right-click **Axial Stress vs. Axial Strain (Cyclic)** and choose **Duplicate**.
- 2 In the **Settings** window for **ID 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 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid2.e133)`.


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 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid2.e133-withsol('sol6',solid2.e133,setval(para,2)))`.
- 6 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 **ID Plot Group**, locate the **Axis** section.
- 3** Select the **x-axis log scale** checkbox.
- 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**.