Model created in COMSOL Multiphysics 6.4

Triaxial and Oedometer Test with Modified Cam-Clay Material Model

Introduction

The Modified Cam-Clay (MCC) model is a popular constitutive model for soft soils and clays. The MCC model has a nonlinear relation between stress and strain with a smooth yield surface. In this example, simulations of triaxial tests are carried out to examine the constitutive relation of the MCC model, as it is originally developed for triaxial loading conditions. The oedometer test is also an important test in the field of geomechanics, which is used frequently to determine the material parameters of soils. In this example, the drained triaxial compression test and the oedometer test presented in Ref. 1 are simulated.

The Cam-Clay family of models do not have any stiffness at zero stress; hence, it always starts with an initial mean stress. In COMSOL Multiphysics, the initial mean stress of the MCC model is equal to the reference pressure. The MCC model comes in two flavors: it either requires the specification of a constant shear modulus or a constant Poisson's ratio. In this example, the constant Poisson's ratio formulation is used in order to match the results in Ref. 1. Although the analysis presented in Ref. 1 is transient, a stationary analysis is sufficient to predict the behavior.

Model Definition

In both the triaxial and oedometer tests, a cylindrical soil specimen of 3.91 cm in diameter and 8 cm in height is used. For the triaxial test (see Figure 1), a confinement pressure is applied to create a state of isotropic compression, and later the soil sample is compressed axially. For the oedometer test, the bottom and side boundaries of the cylindrical specimen are constrained in the normal direction and an axial load is applied on the top boundary.

SOIL PROPERTIES

The soil properties for the MCC material model as given in Ref. 1 are

- Density $\rho = 2400\ \mathrm{kg/m^3}$, Poisson's ratio $\nu = 0.35$, slope of critical state line $M = 1.2$, swelling index $\kappa = 0.02$, compression index $\lambda = 0.1$, void ratio $e_{ref} = 1$ at a reference pressure $p_{ref} = 98$ kPa, and initial consolidation pressure $p_{c0} = 100$ kPa or 500 kPa.

.

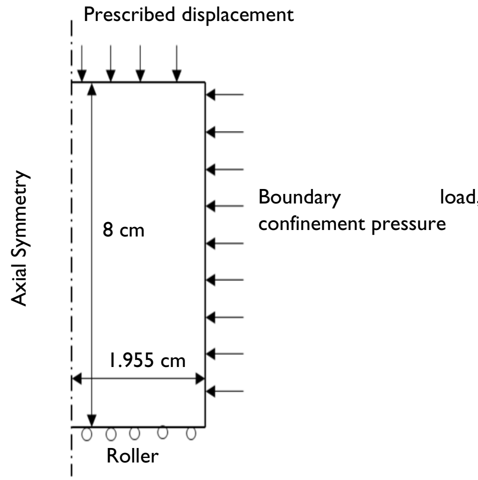

Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

CONSTRAINTS AND LOADS

- For the isotropic compression stage in the triaxial test, a mean stress of 100 kPa is maintained throughout the test. As the MCC model has an initial mean stress equal to the reference pressure, an additional pressure of 2 kPa is applied using an **External Stress** node with the **In situ stress** option.
- For the axial compression stage in the triaxial test, the soil sample is compressed by applying a prescribed displacement on the top boundary. Allow the right boundary to expand freely in the radial direction, and apply a roller boundary condition at the bottom boundary.
- For the axial compression stage in the oedometer test, the soil sample is compressed using a boundary load on the top boundary. A roller boundary condition is applied on the bottom and right vertical boundaries in order simulate confinement (zero radial strain).

Results and Discussion

Note that for the sake of consistency with geomechanics conventions, the compressive axial stress and strain are plotted along the positive axis, whereas the tensile stress and strain are plotted along the negative axis in all the figures. The response of the MCC model with different overconsolidation ratios (OCRs) are plotted in the same figures for comparison purposes.

The OCR is a ratio between the initial consolidation pressure to the initial mean effective stress. For the oedometer test in Ref. 1, the OCR is the ratio of the initial consolidation pressure to the initial vertical load. A soil with an OCR equal to 1 is referred to as normally consolidated. When the OCR is equal to 5 or 50, the soil is instead referred to as highly overconsolidated.

The variation in von Mises stress versus axial strain with different OCRs is shown in Figure 2. The stress-strain curve is nonlinear, which is a characteristic of the MCC model. As the axial displacement increases, the von Mises stress increases hyperbolically and approaches a critical state asymptotically. When the soil attains the critical state, additional loading does not produce any volume changes or hardening. The response of the soil in Figure 2 matches very closely with the numerical results given in Ref. 1 (see Figure 7 in Ref. 1).

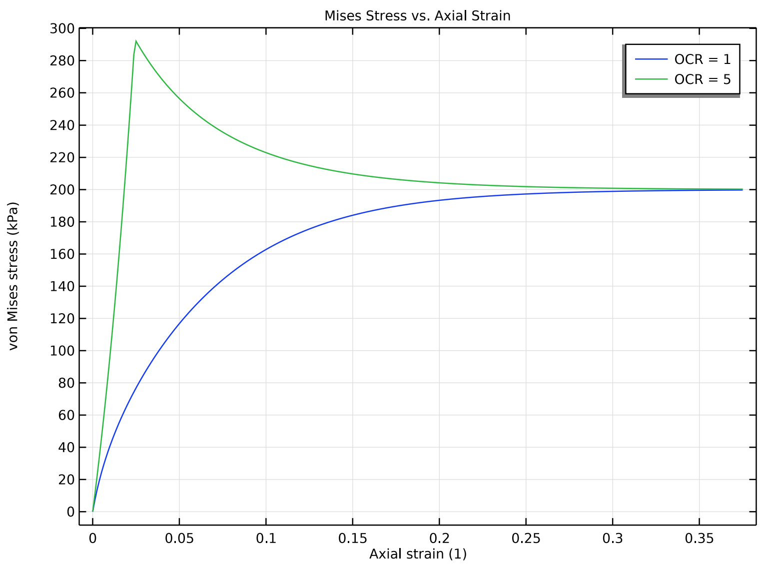

Figure 2: von Mises stress versus axial strain.

The variation in the total volumetric strain versus the axial strain with different OCRs can be seen in Figure 3, which matches very closely with the numerical results given in Ref. 1 (see Figure 8 in Ref. 1). For normally consolidated soils (OCR = 1), the total volumetric strain remains compressive. In contrast, the volumetric response for highly overconsolidated soils turns tensile after an initial compressive phase. This counterintuitive behavior can be further explained by Figure 4 and Figure 5.

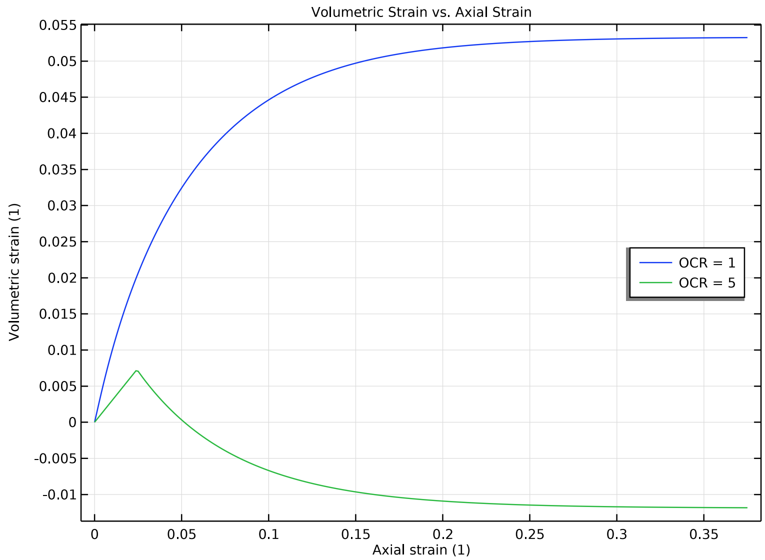

Figure 3: Volumetric strain versus axial strain.

The evolution of the consolidation pressure and the volumetric plastic strains is shown in Figure 4 and Figure 5, respectively. For normally consolidated soils, the consolidation pressure increases, indicating that the final yield envelope is expanding. This, in turn, gives compressive volumetric plastic strains, see Figure 5. This behavior is called isotropic hardening. For highly overconsolidated soils, the consolidation pressure is decreasing, indicating the shrinking of the final yield envelope, which in turn develops tensile volumetric plastic strains, see Figure 5. This behavior called isotropic softening

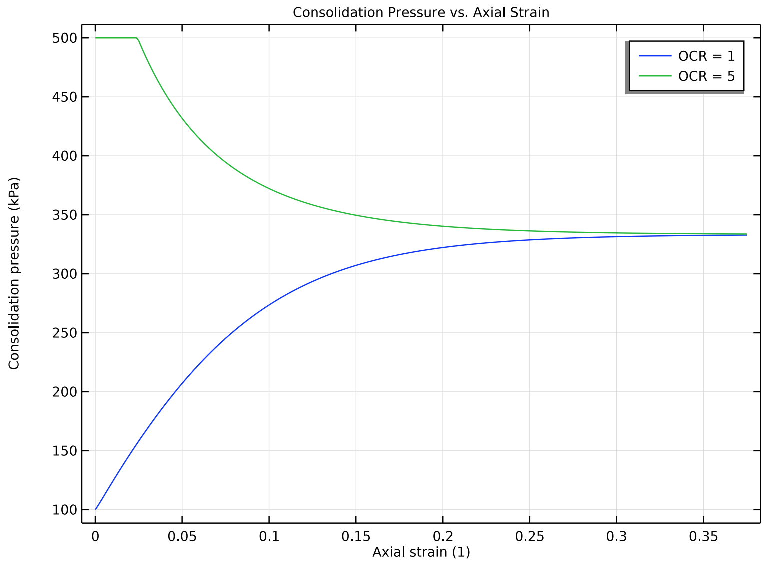

Figure 4: Consolidation pressure versus axial strain.

In COMSOL Multiphysics, the reference pressure acts as an initial stress and needs to be nonzero. For the oedometer test in Ref. 1, there seems to be no initial stress. Hence, for the corresponding test in COMSOL Multiphysics, set the reference pressure to 1 kPa. The void ratio at the reference pressure is calculated based on its value at 98 kPa as

$$e = e_{\mathrm{ref}} - \lambda \ln \frac{p}{p_{\mathrm{ref}}}$$

$$e_{\mathrm{ref}} = 1 + 0.1\ln(98) = 1.4584$$

The variation in the void ratio versus the logarithm of the vertical load for the highly overconsolidated soil in the oedometer test is plotted in Figure 6, which matches qualitatively with numerical results given in Ref. 1 (see Figure 6 in Ref. 1). The slight difference in the results can be due to the different initial conditions.

The variation in the total volumetric strain versus the axial load for the highly overconsolidated soil in the oedometer test can be seen in Figure 7. When the yield limit is reached, the response is nonsmooth.

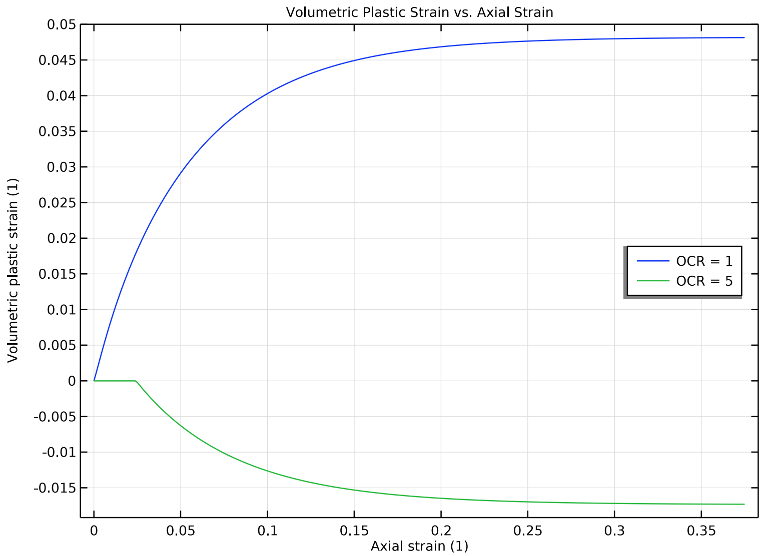

Figure 5: Volumetric plastic strain versus axial strain.

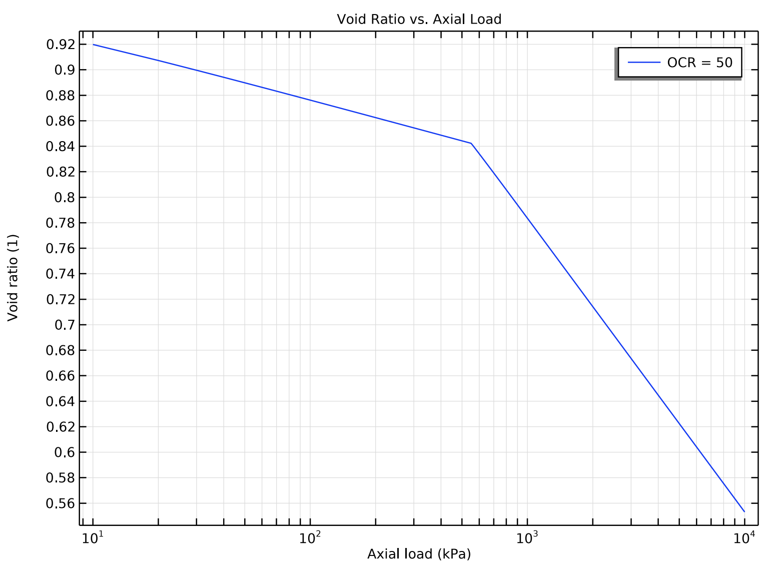

Figure 6: Void ratio versus the axial load (log-scale).

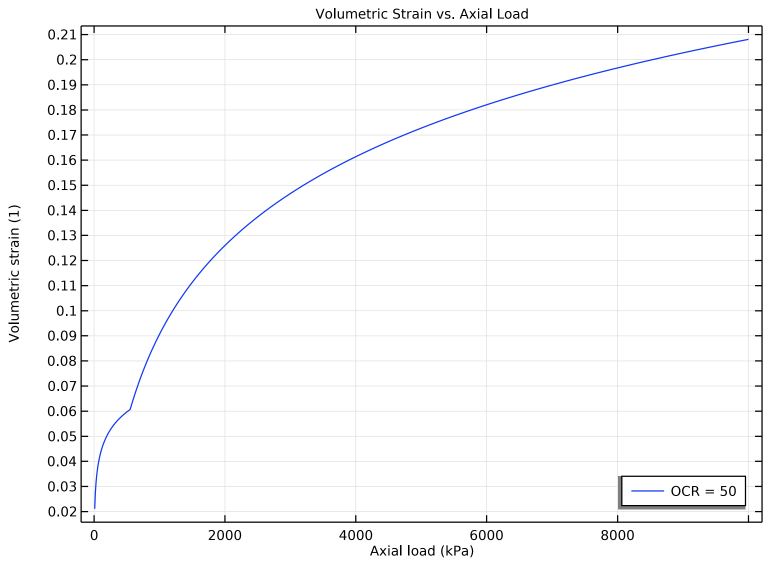

Figure 7: Volumetric strain versus axial load.

Notes About the COMSOL Implementation

In COMSOL Multiphysics, the MCC model comes in two flavors: it either requires specification of constant shear modulus or a constant Poisson's ratio. For the constant shear modulus option, the Poisson's ratio is computed based on the bulk modulus and the given shear modulus. This Poisson's ratio depends on deformation, but does not enter into the constitutive relation and remains only as a postprocessing variable. For the constant Poisson's ratio option, the shear modulus is calculated from the bulk modulus and the Poisson's ratio. This variable shear modulus does enter into the constitutive relation.

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

Reference

1. G. Ye and B. Ye, "Investigation of the Overconsolidation and Structural Behavior of Shanghai Clays by Element Testing and Constitutive Modeling," *Underground Space*, vol. 1, pp. 62-77, 2016.

2. 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_mcc`

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 In the table, enter the following settings:

Name	Expression	Value	Description
disp	0[cm]	0 m	Axial displacement
p0	2[kPa]	2000 Pa	Pressure
OCR	5	5	Overconsolidation ratio
F	10[kPa]	10000 Pa	Axial load
isOedometerTest	0	0	Boolean for oedometer test

Add variables for the reference void ratio and the reference pressure, which are different for the two tests.

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
e_ref	1*(1-isOedometerTest)+1.4584*isOedometerTest		Void ratio at reference pressure
p_ref	98[kPa]*(1-isOedometerTest)+1[kPa]*isOedometerTest	Pa	Reference pressure

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 `1.955[cm]`.

4 In the **Height** text field, type `8[cm]`.

5 Click **Build Selected**.

SOLID MECHANICS (SOLID)

Modified Cam-Clay Material Model

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

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Modified Cam-Clay Material Model` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Elastoplastic Soil Material** section. Find the **Parameters** subsection. From the M list, choose **From material**.

5 From the e_0 list, choose **From void ratio at reference pressure**.

6 From the e_{ref} list, choose **User defined**. In the associated text field, type `e_ref`.

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

8 In the p_{c0} text field, type `100[kPa]*OCR`.

The triaxial test is carried out in two steps. The first step is needed to get the initial stress state of the sample, and the second step is an axial compressive loading. The initial stress state can be modeled using the **In situ stress** option of the **External Stress** node.

External Stress [Triaxial Test]

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

2 In the **Settings** window for **External Stress**, type `External Stress [Triaxial Test]` in the **Label** text field.

3 Locate the **External Stress** section. 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`.

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 `F`.

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	`0.35`	1	Critical state model
Slope of critical state line	M	`1.2`	1	Critical state model
Density	rho	`2400[kg/m^3]`	kg/m³	Basic
Swelling index	kappaSwelling	`0.02`	1	Modified Cam-clay
Compression index	lambdaComp	`0.1`	1	Modified Cam-clay

One 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**.

Disable the features that are not needed in this study.

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

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

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)** > **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,-0.03)	m

Add a **Parametric Sweep** node to study the soil specimen for different OCRs.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
OCR (Overconsolidation ratio)	1 5	1

5 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`.

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

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

2 Click **Add**.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
isOedometerTest (Boolean for oedometer test)	1	1

Disable the features that are not needed in this study.

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

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Modified Cam-Clay Material Model** > **External Stress [Triaxial Test]** 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
F (Axial load)	`range(10,10,10000)`	`kPa`

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 Select the **Apply conversions to expressions with the same dimensions** checkbox.
9 Click **Apply**.

Mises Stress vs. Axial Strain

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Mises Stress vs. Axial Strain` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Mises 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 `von Mises stress (kPa)`.

Point Graph 1

1 Right-click **Mises Stress vs. Axial Strain** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Study: Triaxial Test/Parametric Solutions 1 (sol2)**.
4 From the **Parameter selection (OCR)** list, choose **First**.
5 Select Point 2 only.
6 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.mises`.
7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

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

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

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

11 In the table, enter the following settings:

Legends
OCR = 1

Point Graph 2

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

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

3 From the **Parameter selection (OCR)** list, choose **Last**.

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

Legends
OCR = 5

5 In the **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 **Manual**.

4 In the **Title** text area, type `Volumetric Strain 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 `Volumetric strain (1)`.

8 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Point Graph 1

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

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

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

4 From the **Parameter selection (OCR)** list, choose **First**.

5 Select Point 2 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `-solid.evol`.

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

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

9 Locate the **Legends** section. Select the **Show legends** checkbox.

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

11 In the table, enter the following settings:

Legends
OCR = 1

Point Graph 2

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

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

3 From the **Parameter selection (OCR)** list, choose **Last**.

4 Select Point 4 only.

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

Legends
OCR = 5

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

Consolidation Pressure vs. Axial Strain

1 In the **Results** toolbar, click **1D Plot Group**.

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

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

4 In the **Title** text area, type `Consolidation Pressure 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 `Consolidation pressure (kPa)`.

Point Graph 1

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

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

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

4 From the **Parameter selection (OCR)** list, choose **First**.

5 Select Point 2 only.

6 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** > **Modified Cam-clay** > **solid.epm1.pc - Consolidation pressure - Pa**.

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

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

9 Locate the **Legends** section. Select the **Show legends** checkbox.

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

11 In the table, enter the following settings:

Legends
OCR = 1

Point Graph 2

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

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

3 From the **Parameter selection (OCR)** list, choose **Last**.

4 Select Point 4 only.

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

Legends
OCR = 5

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

Volumetric Plastic Strain vs. Axial Strain

1 In the **Results** toolbar, click **1D Plot Group**.

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

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

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

5 Locate the **Plot Settings** section.

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 `Volumetric plastic strain (1)`.

8 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Point Graph 1

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

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

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

4 From the **Parameter selection (OCR)** list, choose **First**.

5 Select Point 2 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `-solid.epvol`.

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

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

9 Locate the **Legends** section. Select the **Show legends** checkbox.

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

11 In the table, enter the following settings:

Legends
OCR = 1

Point Graph 2

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

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

3 From the **Parameter selection (OCR)** list, choose **Last**.

4 Select Point 4 only.

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

Legends
OCR = 5

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

Void Ratio vs. Axial Load

1 In the **Results** toolbar, click **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Oedometer Test/ Parametric Solutions 2 (sol6)**.

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

5 In the **Title** text area, type `Void Ratio vs. Axial Load`.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** checkbox. In the associated text field, type `Axial load (kPa)`.

8 Select the **y-axis label** checkbox. In the associated text field, type `Void ratio (1)`.

9 Locate the **Axis** section. Select the **x-axis log scale** checkbox.

Point Graph 1

1 Right-click **Void Ratio vs. Axial Load** and choose **Point Graph**.

2 Select Point 2 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** > **Modified Cam-clay** > **solid.epm1.evoid - Void ratio - 1**.

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

5 In the **Expression** text field, type `F`.

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
OCR = 50

9 In the **Void Ratio vs. Axial Load** toolbar, click **Plot**.

Volumetric Strain vs. Axial Load

1 In the **Results** toolbar, click **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Oedometer Test/ Parametric Solutions 2 (sol6)**.

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

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** checkbox. In the associated text field, type `Axial load (kPa)`.

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

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

Point Graph 1

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

2 Select Point 2 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 `F`.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

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

9 In the table, enter the following settings:

Legends
`OCR = 50`

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

Consolidation Pressure vs. Axial Strain, Mises Stress vs. Axial Strain, Volumetric Plastic Strain vs. Axial Strain, Volumetric Strain vs. Axial Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mises Stress vs. Axial Strain**, **Volumetric Strain vs. Axial Strain**, **Consolidation Pressure vs. Axial Strain**, and **Volumetric Plastic Strain vs. Axial Strain**.

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

Triaxial Test

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

Void Ratio vs. Axial Load, Volumetric Strain vs. Axial Load

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Void Ratio vs. Axial Load** and **Volumetric Strain vs. Axial Load**.

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

Oedometer Test

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