Model created in COMSOL Multiphysics 6.4

Mechanical Modeling of Bentonite Clay

Introduction

The Barcelona Basic Model (BBM) is one of the most widely used constitutive models for representing hydromechanical behavior in partially saturated soils. In COMSOL Multiphysics, a modified version of the original model is implemented, the so-called Extended Barcelona Basic Model (BBMx) as described in Ref. 1. By including the suction as a state variable in the constitutive relationship, the BBM and BBMx models can describe partially saturated soils. As opposed to fully saturated soils, partially saturated soils can be loaded either hydraulically or mechanically, or by combination of both mechanisms.

This example validates the BBMx model implemented in COMSOL Multiphysics (Ref. 1) by qualitatively matching the results of mechanical modeling to results of the oedometer, triaxial and suction tests presented in Ref. 2 for bentonite clays. However, the experiments in Ref. 2 are set up to determine suitable material parameters to characterize the BBM model, and while the fundamental principles are the same, there are some differences between the BBM and BBMx models. In addition to the tests described in Ref. 2, an additional test of constrained swelling is carried out to compare the results with analytical formulas.

The Cam-Clay family of soil models, like the BBMx model, do not define any stiffness at zero stress, hence numerical simulations that use these soil models always prescribe an initial mean stress equal to the reference pressure at zero strain.

Model Definition

A cylindrical soil specimen of 3.91 cm in diameter and 8 cm in height is used for all tests. The specimen is represented by a 2D axisymmetric geometry due to the intrinsic symmetry of the problem.

For the oedometer test, initial stresses in the radial, circumferential and axial directions are applied. In the first stage, the bottom and side boundaries are constrained in the normal direction. In the second stage, an axial load is applied on the top boundary.

For the uniaxial swelling test, initial stresses in the radial, circumferential and axial directions are applied together with a constant load on the top boundary. The sample is then subjected to changes in suction.

For the triaxial test (Figure 1), an initial stress is applied to create a state of isotropic compression, the soil sample is then compressed further in the axial direction.

In the constrained swelling test, all boundaries are mechanically constrained, and the sample is then subjected to changes in suction.

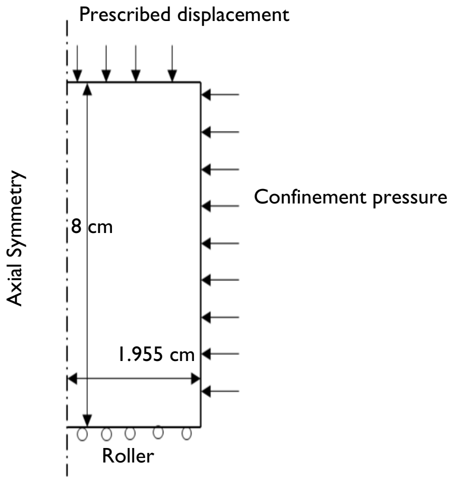

Figure 1: Dimensions, boundary conditions, and boundary loads for the triaxial test.

SOIL PROPERTIES

- Soil properties from Ref. 2 that are used for the BBMx model are presented in Table 1.

TABLE 1: MATERIAL PROPERTIES FOR THE SOIL MODEL.

Property	Variable	Oedometer test	Uniaxial swelling test/ constrained swelling test	Triaxial test
Poisson's ratio	ν	0.224	0.2	0.3
Density	ρ	2400 kg/m^3	2400 kg/m^3	2400 kg/m^3
Swelling index	κ	0.057	0.06	0.1
Swelling index for changes in suction	κ_s	0	0.3/0.03	0
Compression index at saturation	λ	0.101	0.9	0.135
Compression index for changes in suction	λ_s	0.01	0.8	0.1
Angle of internal friction	ϕ	25.37°	25.37°	13.34°
Weight parameter	w	1	0.75	1
Soil stiffness parameter	m	1 MPa	33.33 MPa	1 MPa
Plastic potential smoothing parameter	b_s	100	100	100
Tension to suction ratio	k_s	0.1	0.1	0.1
Initial yield value for suction	s_y	50 MPa	150 MPa	1 MPa
Initial void ratio	e_0	0.772	0.579	1.212
Initial suction	s_0	28 MPa	101.5 MPa	0 MPa
Reference pressure	p_{ref}	0.1 MPa	0.2 MPa	0.1 MPa
Initial consolidation pressure	p_{c0}	7.7 MPa	3.5 MPa	1.5 MPa

- The compression index for changes in current suction, the plastic potential smoothing parameter, and the initial yield value for suction are not specified in Ref. 2.
- The angle of internal friction ϕ is computed from the slope of the critical state line, M, using the formula

$$M = \frac{6\sin\phi}{3 - \sin\phi}$$

- In COMSOL Multiphysics, either the void ratio at reference pressure together with the saturation e_{ref0}, or the initial void ratio e_0 are needed as a material property. For this example, an initial void ratio is provided as an input material property.
- For the oedometer and triaxial tests, the compression index at current suction is independent of the suction, since suction values are held constant in Ref. 2. To achieve this, the weight parameter is set to zero. In COMSOL Multiphysics, the formula for the compression index at current suction is implemented in a different way, therefore, the weight parameter is set to one in order to achieve the same effect as shown in Ref. 2. In these cases the choice of the soil stiffness parameter does not matter.
- The yield function and the plastic potential used in COMSOL Multiphysics is different than the expressions given in Ref. 2. The nonassociative parameter for the plastic potential is always set to one in COMSOL Multiphysics as compared to Ref. 2.

CONSTRAINTS AND LOADS

- It is sufficient to model the right half of the domain due to axial symmetry.
- For the oedometer test, an initial stress of –2.97 MPa is applied in the radial and circumferential directions, while –0.18 MPa is applied in the axial direction.
- Note that the reference pressure p_{ref} acts as an initial stress, therefore the values of the diagonal components of the *in situ* stress tensor defined in the **External Stress** node are –2.87 MPa, –2.87 MPa, and –0.08 MPa.
- During the loading stage of the oedometer test, the axial compressive stress is increased from 0.18 MPa to 19.77 MPa, and then decreased from 19.77 MPa to 1.00 MPa. Roller boundary conditions are applied on the bottom and side boundaries. The suction value is kept constant throughout the analysis.
- For the uniaxial swelling test, an initial stress of –2.54 MPa is applied in the radial and circumferential directions, while –8.90 MPa is applied in the axial direction. The reference pressure p_{ref} acts as an initial stress, so the values of the diagonal components of the *in situ* stress tensor in the **External Stress** node are -2.34 MPa, –2.34 MPa, and –8.70 MPa.
- In the loading step of the uniaxial swelling test, the axial compressive stress is maintained constant at –8.90 MPa, while the suction is reduced from 101.5 MPa to 12.6 MPa. Roller boundary conditions are applied on the bottom and side boundaries.
- For the triaxial test in isotropic compression, an initial hydrostatic stress of 1.1 MPa is applied. As the reference pressure p_{ref} acts as an initial stress, the values of the diagonal components of the *in situ* stress tensor defined in the **External Stress** node are –1 MPa.

- During the axial compression stage of the triaxial test, the soil sample is compressed by a prescribed displacement on the top boundary. The top-right corner is allowed to expand freely in the radial direction, and a roller boundary condition is applied at the bottom boundary.
- For the constrained swelling test, all boundaries are constrained in the normal direction and the suction is reduced from 101.5 MPa to 12.6 MPa.

Results and Discussion

Note that for consistency with the geomechanics sign convention, compressive stress and strain is plotted along the positive axis in all figures, while tensile stress and strain is plotted along the negative axis.

The evolution of both the void ratio and the radial stress versus axial stress during the loading and unloading phases in the oedometer test are shown in Figure 2, and the evolution of both the deviatoric stress and the mean stress in Figure 3. The results of both phases of the test resembles the results shown in Ref. 2, but there are some differences mainly in the unloading phase. These differences may be attributed to the implementation of different yield functions and plastic potentials.

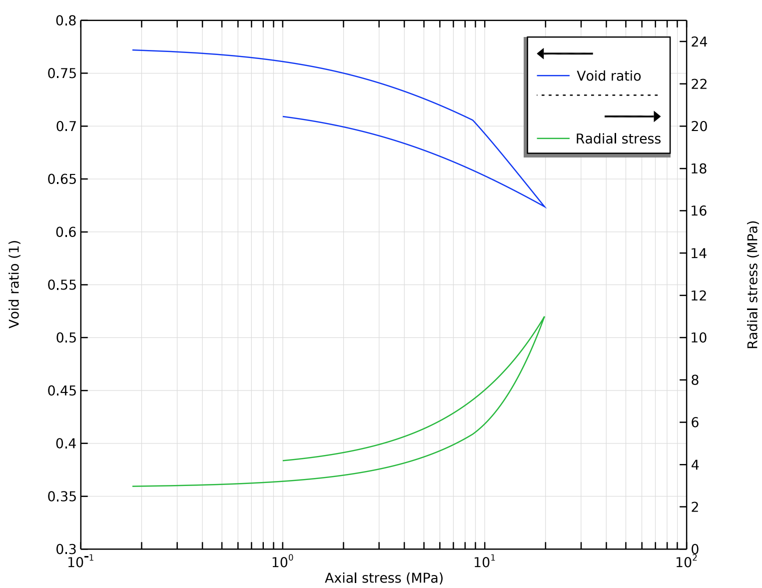

Figure 2: Evolution of void ratio and radial stress versus axial stress for the oedometer test.

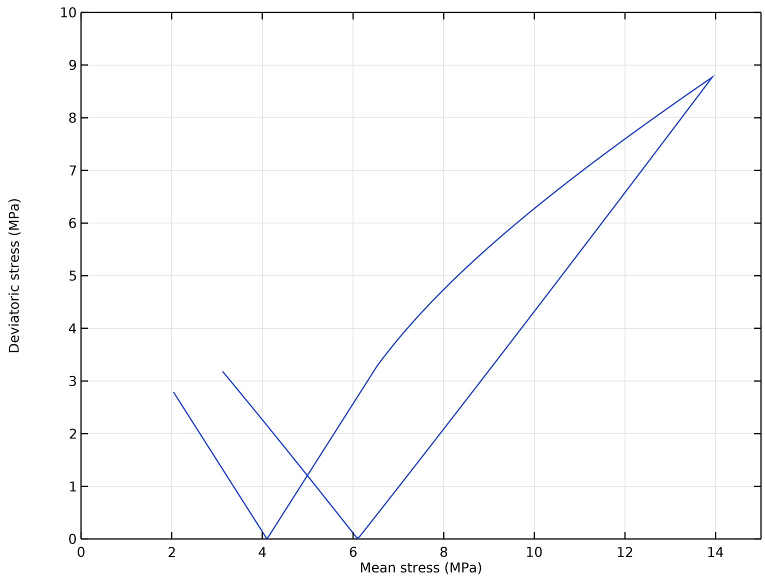

Figure 3: Stress path in the p-q plane for the oedometer test.

The evolution of both the void ratio and the radial stress versus suction in the uniaxial swelling test are shown in Figure 4. Both curves match the numerical results given in Ref. 2. The evolution of the mean stress is shown in Figure 5. The void ratio, the radial stress, and the mean stress all level out at a suction around 80 MPa, which is also reported in Ref. 2. This behavior is due to the accumulation of plastic strains. Moreover, at the end of the suction cycle, the void ratio shows an increasing trend, and this behavior is also reported in Ref. 2.

The evolution of the volumetric strain and the deviatoric stress versus axial strain are shown in Figure 6 and Figure 7, respectively. These results show good qualitative agreement with the results presented in Ref. 2. The slight mismatch is anticipated due to the different implementation of the material models, given by the different definitions for the yield function and the plastic potential.

The stress path in the p-q plane for the triaxial test is shown in Figure 8. The deviatoric stress equals 0.3 MPa when the mean stress is 1.2 MPa. this stress state coincides with the initial yield surface. These are the same value as reported in Ref. 2.

For the constrained swelling stress, evolution of the mean stress versus suction is shown in Figure 9. This result matches exactly the analytical expression given by Equation 1 in the pure elastic range.

For the BBMx model, the evolution of the volumetric strain is given by

$$d\varepsilon_v = \frac{dp}{K} + \frac{ds}{K_s}$$

$$\text{with } K = \frac{(1+e_0)p}{\kappa} \text{ and } K_s = \frac{(1+e_0)(s+p_{\text{atm}})}{\kappa_s}$$

For constrained swelling there is no change in volumetric strain, hence

$$\kappa\frac{dp}{p} = -\kappa_s\frac{ds}{s}$$

Integrating this equation, and finding the integration constant based on the initial values, gives

$$\ln\left(\frac{p}{p_{\text{ref}}}\right) = -\frac{\kappa}{\kappa_s}\ln\left(\frac{s+p_{\text{atm}}}{s_0+p_{\text{atm}}}\right)$$

$$p = p_{\text{ref}}e^{-A} \text{ with } A = \frac{\kappa}{\kappa_s}\ln\left(\frac{s+p_{\text{atm}}}{s_0+p_{\text{atm}}}\right) \quad (1)$$

For the constrained swelling test, both the volumetric strain and the void ratio remain constant as suction changes; this is portrayed in Figure 10.

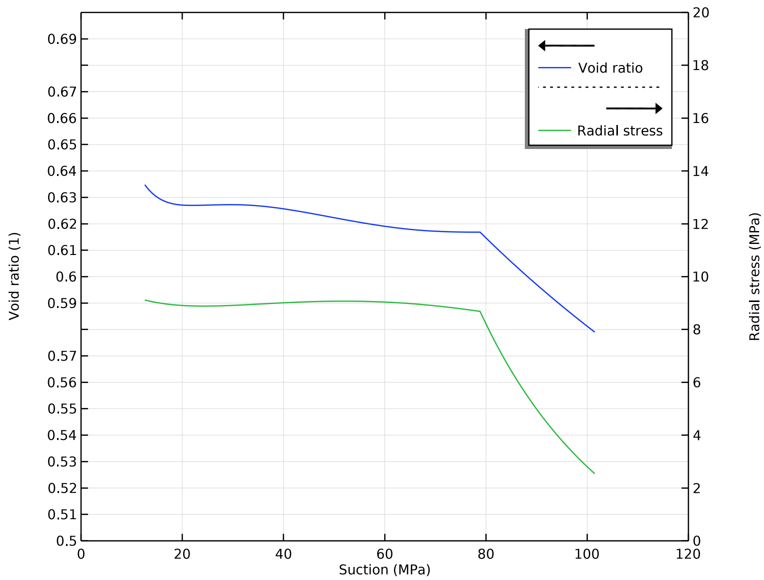

Figure 4: Evolution of void ratio and radial stress versus suction in the uniaxial swelling test.

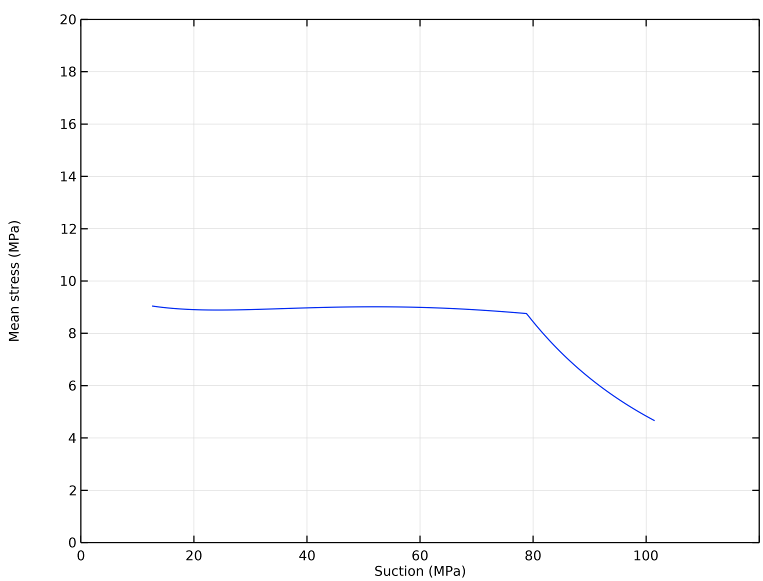

Figure 5: Evolution of mean stress versus suction in the uniaxial swelling test.

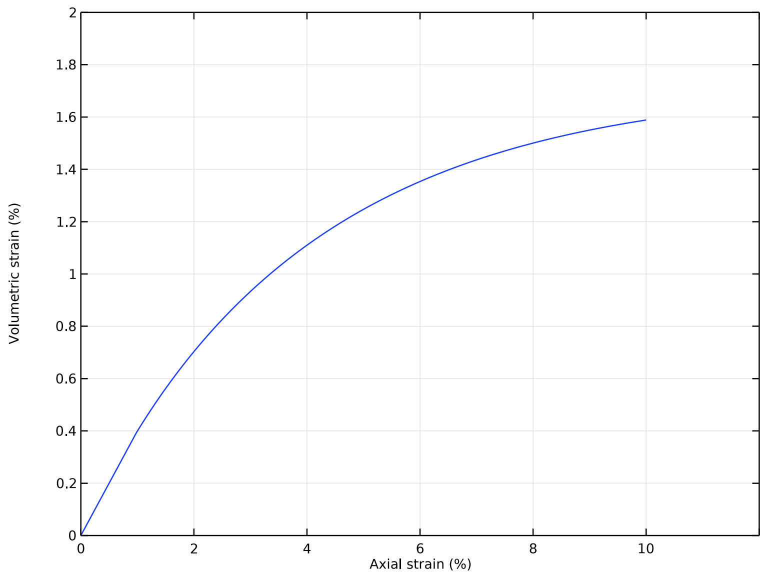

Figure 6: Evolution of volumetric strain versus axial strain in the triaxial test.

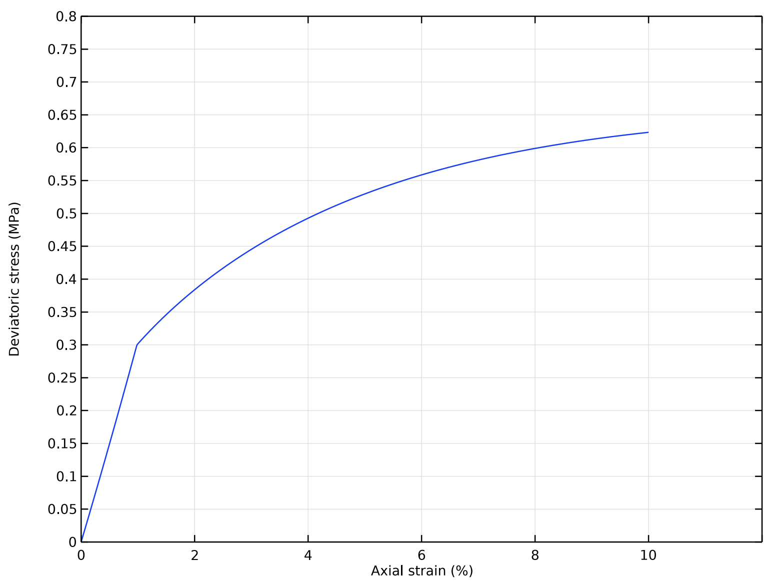

Figure 7: Evolution of deviatoric stress versus axial strain in the triaxial test.

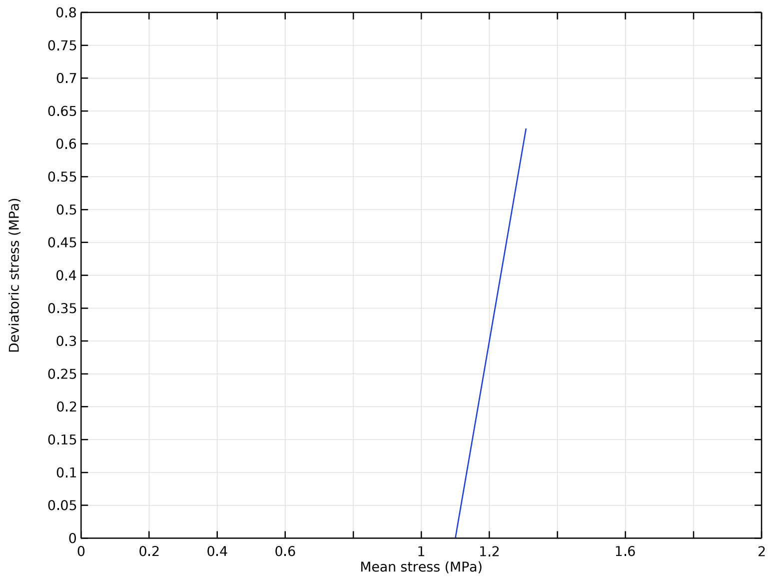

Figure 8: Stress path in the p-q plane for the triaxial test.

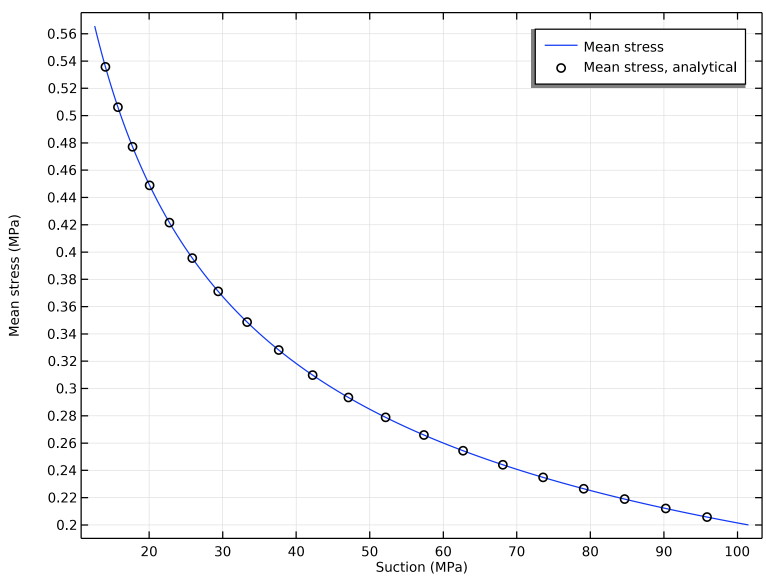

Figure 9: Evolution of mean stress versus suction in the constrained swelling test.

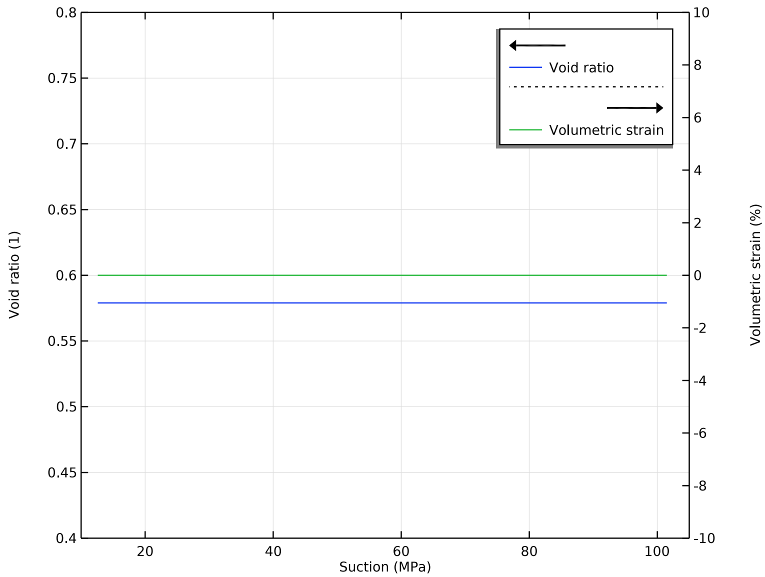

Figure 10: Evolution of void ratio and volumetric strain versus suction in the constrained swelling test.

Notes About the COMSOL Implementation

The *in situ* stresses are the stresses in the soil sample in a strain-free configuration. There are two methods to account for *in situ* stresses in COMSOL Multiphysics. One way is to create two stationary study steps or studies, with a combination of the **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 single study. This option gives initial stresses in the soil sample without imposing any strains. In this example, the second method is used to model the *in situ* stresses in the soil sample.

References

1. D. Pedroso and M. Farias, “Extended Barcelona Basic Model for unsaturated soil under cyclic loadings,” *Computers and Geotechnics*, vol. 38, no. 5, pp. 731–740, 2011.

2. O. Kristensson and M. Åkesson, “Mechanical modeling of MX-80 - Quick tools for BBM parameter analysis,” *Physics and Chemistry of the Earth*, vol. 33, pp. 5508–5515, 2008.

Application Library path: `Geomechanics_Module/Verification_Examples/mechanical_modeling_of_bentonite_clay`

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 Right-click and choose **Add Physics**.

4 Repeat the two previous instructions three times to add four Solid Mechanics interfaces in total.

5 Click **Study**.

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

7 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
para	0	0	Parameter
disp3	0[cm]	0 m	Axial displacement

BBMx Material Parameters

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

First set up a physics interface for each soil test.

2 In the **Settings** window for **Parameters**, type `BBMx Material Parameters` 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 `mechanical_modeling_of_bentonite_clay_oedometer_parameters.txt`.

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

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

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

8 In the **Settings** window for **Case**, type `Uniaxial Swelling Test Parameters` 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 `mechanical_modeling_of_bentonite_clay_uniaxial_swelling_parameters.txt`.

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

12 In the **Settings** window for **Case**, type `Triaxial Test Parameters` 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 `mechanical_modeling_of_bentonite_clay_triaxial_parameters.txt`.

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

16 In the **Settings** window for **Case**, type `Constrained Swelling Test Parameters` in the **Label** text field.

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

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

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 [OEDOMETER TEST]

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 [Oedometer Test]` in the **Label** text field.

Elastoplastic Soil Material 1

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

The provided material data does not include the void ratio at reference pressure and saturation but instead gives the initial void ratio. Hence, change the default option of the initial void ratio property to **From material** instead of deriving it from void ratio at reference pressure and saturation.

2 Select Domain 1 only.

3 In the **Settings** window for **Elastoplastic Soil Material**, locate the **Elastoplastic Soil Material** section.

4 From the **Material model** list, choose **Extended Barcelona basic**.

5 From the $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.

6 Find the **Parameters** subsection. In the s_0 text field, type `s0`.

7 In the s text field, type `s0`.

8 In the p_{ref} text field, type `pref`.

9 In the p_{c0} text field, type `pc0`.

External Stress 1

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

The oedometer 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 a compressive loading. The initial stress state can be modeled using the **In situ stress** option of the **External Stress** node.

2 In the **Settings** window for **External Stress**, locate the **External Stress** section.

3 From the **Stress input** list, choose **In situ stress**.

4 From the list, choose **Symmetric**.

5 Specify the σ_{ins} matrix as

-2.87[MPa]	0	0

0	-2.87[MPa]	0
0	0	-0.08[MPa]

Roller 1

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

2 Select Boundaries 2 and 4 only.

Instead of adding a boundary load on the top boundary, prescribe the displacement as a function of the axial stress.

Elastoplastic Soil Material 1

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics [Oedometer Test] (solid)** > **Elastoplastic Soil Material 1** node.

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 sigmaZ.

4 In the table, enter the following settings:

t	f(t)
0	-0.18
1	-19.77
2	-1

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

Argument	Unit
t	1

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

Function	Unit
sigmaZ	MPa

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 Select Domain 1 only.

3 In the **Settings** window for **Average**, locate the **Advanced** section.

4 From the **Frame** list, choose **Material (R, PHI, Z)**.

5 Clear the **Compute integral in revolved geometry** checkbox.

SOLID MECHANICS [OEDOMETER TEST] (SOLID)

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 `disp1`.

Add a global equation to compute the axial displacement, so that the axial stress equals the reaction force for such a prescribed displacement. Show the equation-based contributions to add a global equation.

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

8 Click **OK**.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

Multiply the global equation by a suitable penalty factor in order to strictly satisfy this equation. For the current model, `1e5` is an appropriate penalty factor.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
disp1	(aveop1(solid.SZZ)-sigmaZ(para))*1e5	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog, type `disp` in the text field.

6 In the tree, select **General** > **Displacement (m)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `pressure` in the text field.

11 In the tree, select **General** > **Pressure (Pa)**.

12 Click **OK**.

DEFINITIONS

Interpolation 2 (int2)

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 `suction`.

4 In the table, enter the following settings:

t	f(t)
0	101.5
1	12.6

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

Argument	Unit
t	1

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

Function	Unit
suction	MPa

SOLID MECHANICS [UNIAXIAL SWELLING TEST]

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 [Uniaxial Swelling Test]` in the **Label** text field.

Elastoplastic Soil Material 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Elastoplastic Soil Material**, locate the **Elastoplastic Soil Material** section.

4 From the **Material model** list, choose **Extended Barcelona basic**.

5 From the Γ(θ) list, choose **Matsuoka–Nakai**.

6 Find the **Parameters** subsection. In the s_0 text field, type `s0`.

7 In the s text field, type `suction(para)`.

8 In the p_{ref} text field, type `pref`.

9 In the p_{c0} text field, type `pc0`.

External Stress 1

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

The uniaxial swelling 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 a reduction in the suction while keeping a constant axial stress. The initial stress state can be modeled using the **In situ stress** option of the **External Stress** node.

2 In the **Settings** window for **External Stress**, locate the **External Stress** section.

3 From the **Stress input** list, choose **In situ stress**.

4 From the list, choose **Symmetric**.

5 Specify the σ_{ins} matrix as

-2.34[MPa]	0	0
0	-2.34[MPa]	0
0	0	-8.70[MPa]

Roller 1

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

2 Select Boundaries 2 and 4 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 `disp2`.

Global Equations 1 (ODE2)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
disp2	(aveop1(solid2.SZZ)+8.90[MPa])	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog, type `disp` in the text field.

6 In the tree, select **General** > **Displacement (m)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `pressure` in the text field.

11 In the tree, select **General** > **Pressure (Pa)**.

12 Click **OK**.

SOLID MECHANICS [TRIAXIAL TEST]

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 3 (solid3)**.

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

Elastoplastic Soil Material 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Elastoplastic Soil Material**, locate the **Elastoplastic Soil Material** section.

4 From the **Material model** list, choose **Extended Barcelona basic**.

5 From the $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.

6 Find the **Parameters** subsection. In the s_0 text field, type `s0`.

7 In the s text field, type `s0`.

8 In the p_{ref} text field, type `pref`.

9 In the p_{c0} text field, type `pc0`.

Decrease the relative tolerance in the feature's **Advanced** section. To see this section, activate advanced physics settings as follows.

10 Click the **Show More Options** button in the **Model Builder** toolbar.

11 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

12 Click **OK**.

13 In the **Settings** window for **Elastoplastic Soil Material**, click to expand the **Advanced** section.

14 From the **Local method** list, choose **Backward Euler**.

15 In the **Relative tolerance** text field, type `1e-8`.

External Stress 1

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

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.

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 `-1[MPa]`.

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 `-disp3`.

SOLID MECHANICS [CONSTRAINED SWELLING TEST]

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 4 (solid4)**.

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

Elastoplastic Soil Material 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Elastoplastic Soil Material**, locate the **Elastoplastic Soil Material** section.

4 From the **Material model** list, choose **Extended Barcelona basic**.

5 From the $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.

6 Find the **Parameters** subsection. In the s_0 text field, type `s0`.

7 In the s text field, type `suction(para)`.

8 In the p_{ref} text field, type `pref`.

9 In the p_{c0} text field, type `pc0`.

Roller 1

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

2 Select Boundaries 2–4 only.

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
`P_analytical`	`solid4.epm1.pref*exp(-solid4.epm1.kappaSwellings/solid4.epm1.kappaSwelling0*log((solid4.epm1.ss+solid4.epm1.patm)/(solid4.epm1.ss0+solid4.epm1.patm)))`	N/m²	`Analytical formula for pressure`

MATERIALS

MX-80 Bentonite Clay

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 `MX-80 Bentonite Clay` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	`Nu`	1	Critical state model
Friction angle	internalphi	`Phi`	rad	Critical state model
Density	rho	`Rho`	kg/m³	Basic
Swelling index at saturation	kappaSwelling0	`kappa`	1	Barcelona Basic
Swelling index for changes in suction	kappaSwellingsu	`kappa_s`	1	Barcelona Basic
Compression index at saturation	lambdaComp0	`lambda`	1	Barcelona Basic
Compression index for changes in suction	lambdaCompssu	`lambda_s`	1	Barcelona Basic
Weight parameter	wb	`ws`	1	Barcelona Basic
Soil stiffness parameter	mb	`m`	Pa	Barcelona Basic
Plastic potential smoothing parameter	bb	`bs`	1	Barcelona Basic
Tension to suction ratio	kb	`ks`	1	Barcelona Basic
Initial yield value for suction	sy0	`sy`	Pa	Barcelona Basic
Initial void ratio	evoid0b	`e0`	1	Barcelona Basic

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Only using a few mesh elements is sufficient for this analysis.

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**.

Set up the studies for all tests.

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 Right-click and choose **Add Study**.

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

6 Right-click and choose **Add Study**.

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

8 Right-click and choose **Add Study**.

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

STUDY [OEDOMETER TEST]

1 In the **Settings** window for **Study**, type `Study [Oedometer Test]` in the **Label** text field.

2 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 Click **Add**.

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

5 Click **Add**.

6 In the table, enter the following settings:

Switch	Cases	Case numbers
BBMx Material Parameters	User defined	1

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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics [Uniaxial Swelling Test] (solid2)**, **Solid Mechanics [Triaxial Test] (solid3)**, and **Solid Mechanics [Constrained Swelling Test] (solid4)**.

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.005,2)`	

Use appropriate scaling and the Constant (Newton) nonlinear method to improve convergence.

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 [Oedometer Test]** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node, then click **Global Equations 1 (comp1.ODE1)**.

4 In the **Settings** window for **State**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `1e-3`.

7 In the **Model Builder** window, expand the **Study [Oedometer Test]** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

8 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

9 From the **Nonlinear method** list, choose **Constant (Newton)**.

10 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

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

STUDY [UNIAXIAL SWELLING TEST]

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

2 In the **Settings** window for **Study**, type `Study [Uniaxial Swelling Test]` 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
BBMx Material Parameters	User defined	2

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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics [Oedometer Test] (solid)**, **Solid Mechanics [Triaxial Test] (solid3)**, and **Solid Mechanics [Constrained Swelling Test] (solid4)**.

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.005,1)`	

Use appropriate scaling and the Constant (Newton) nonlinear method to improve convergence.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

3 In the **Model Builder** window, expand the **Study [Uniaxial Swelling Test]** > **Solver Configurations** > **Solution 4 (sol4)** > **Dependent Variables 1** node, then click **Global Equations 1 (comp1.ODE2)**.

4 In the **Settings** window for **State**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `1e-3`.

7 In the **Model Builder** window, expand the **Study [Uniaxial Swelling Test]** > **Solver Configurations** > **Solution 4 (sol4)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

8 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

9 From the **Nonlinear method** list, choose **Constant (Newton)**.

10 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

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

STUDY [TRIAXIAL TEST]

1 In the **Model Builder** window, click **Study 3**.

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.

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
BBMx Material Parameters	User defined	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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics [Oedometer Test] (solid)**, **Solid Mechanics [Uniaxial Swelling Test] (solid2)**, and **Solid Mechanics [Constrained Swelling Test] (solid4)**.

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

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp3 (Axial displacement)	`range(0,0.001,0.8)`	`cm`

Use the Constant (Newton) nonlinear method to improve convergence.

Solution 7 (sol7)

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

2 In the **Model Builder** window, expand the **Solution 7 (sol7)** node.

3 In the **Model Builder** window, expand the **Study [Triaxial Test]** > **Solver Configurations** > **Solution 7 (sol7)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Constant (Newton)**.

6 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

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

STUDY [CONSTRAINED SWELLING TEST]

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study [Constrained Swelling Test]` 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
BBMx Material Parameters	User defined	4

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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics [Oedometer Test] (solid)**, **Solid Mechanics [Uniaxial Swelling Test] (solid2)**, and **Solid Mechanics [Triaxial Test] (solid3)**.

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

5 Click **Add**.

6 In the table, enter the following settings:

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

Use the Constant (Newton) nonlinear method to improve convergence.

Solution 10 (sol10)

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

2 In the **Model Builder** window, expand the **Solution 10 (sol10)** node.

3 In the **Model Builder** window, expand the **Study [Constrained Swelling Test] > Solver Configurations > Solution 10 (sol10) > Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Constant (Newton)**.

6 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	MPa

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

9 Click **Apply**.

Void Ratio and Radial Stress

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

2 In the **Settings** window for **1D Plot Group**, type `Void Ratio and Radial Stress` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

5 Select the **x-axis label** checkbox. In the associated text field, type `Axial stress (MPa)`.

6 Select the **Secondary y-axis label** checkbox. In the associated text field, type `Radial stress (MPa)`.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

8 In the **x minimum** text field, type `0.1`.

9 In the **x maximum** text field, type `100`.

10 In the **y minimum** text field, type `0.3`.

11 In the **y maximum** text field, type `0.8`.

12 In the **Secondary y minimum** text field, type `0`.

13 In the **Secondary y maximum** text field, type `25`.

14 Select the **x-axis log scale** checkbox.

Point Graph 1

1 Right-click **Void Ratio and Radial Stress** and choose **Point Graph**.

2 Select Point 4 only.

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

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

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

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

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

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

9 In the table, enter the following settings:

Legends
`Void ratio`

Point Graph 2

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

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

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

4 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

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

Legends
`Radial stress`

6 In the **Void Ratio and Radial Stress** toolbar, click **Plot**.

Stress Path

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

2 In the **Settings** window for **1D Plot Group**, type `Stress Path` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Mean stress (MPa)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Deviatoric stress (MPa)`.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

8 In the **x minimum** text field, type `0`.

9 In the **x maximum** text field, type `15`.

10 In the **y minimum** text field, type `0`.

11 In the **y maximum** text field, type `10`.

Point Graph 1

1 Right-click **Stress Path** and choose **Point Graph**.

2 Select Point 4 only.

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

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

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

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

7 In the **Stress Path** toolbar, click **Plot**.

Stress Path, Void Ratio and Radial Stress

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Void Ratio and Radial Stress** and **Stress Path**.

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

Oedometer Test

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

Void Ratio and Radial Stress 1

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

2 In the **Settings** window for **1D Plot Group**, type `Void Ratio and Radial Stress 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Uniaxial Swelling Test]/ Solution 4 (sol4)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

6 Select the **x-axis label** checkbox. In the associated text field, type `Suction (MPa)`.

7 Select the **Secondary y-axis label** checkbox. In the associated text field, type `Radial stress (MPa)`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `120`.

11 In the **y minimum** text field, type `0.5`.

12 In the **y maximum** text field, type `0.7`.

13 In the **Secondary y minimum** text field, type `0`.

14 In the **Secondary y maximum** text field, type `20`.

Point Graph 1

1 Right-click **Void Ratio and Radial Stress 1** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid2.evoid`.

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

6 In the **Expression** text field, type solid2.epm1.ss.

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
Void ratio

Point Graph 2

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

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

3 In the **Expression** text field, type -solid2.SRR.

4 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

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

Legends
Radial stress

6 In the **Void Ratio and Radial Stress 1** toolbar, click **Plot**.

Mean Stress

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

2 In the **Settings** window for **1D Plot Group**, type Mean Stress in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Uniaxial Swelling Test]/ Solution 4 (sol4)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type Suction (MPa).

7 Select the **y-axis label** checkbox. In the associated text field, type Mean stress (MPa).

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type 0.

10 In the **x maximum** text field, type 120.

11 In the **y minimum** text field, type 0.

12 In the **y maximum** text field, type 20.

Point Graph 1

1 Right-click **Mean Stress** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid2.pm`.

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

6 In the **Expression** text field, type `solid2.epm1.ss`.

7 In the **Mean Stress** toolbar, click **Plot**.

Stress Path 1

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

2 In the **Settings** window for **1D Plot Group**, type `Stress Path 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Uniaxial Swelling Test]/ Solution 4 (sol4)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Mean stress (MPa)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Deviatoric stress (MPa)`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `10`.

11 In the **y minimum** text field, type `0`.

12 In the **y maximum** text field, type `8`.

Point Graph 1

1 Right-click **Stress Path 1** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid2.mises`.

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

6 In the **Expression** text field, type `solid2.pm`.

7 In the **Stress Path 1** toolbar, click **Plot**.

Mean Stress, Stress Path 1, Void Ratio and Radial Stress 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Void Ratio and Radial Stress 1**, **Mean Stress**, and **Stress Path 1**.

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

Uniaxial Swelling Test

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

Volumetric Strain

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Triaxial Test]/ Solution 7 (sol7)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

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

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

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `12`.

11 In the **y minimum** text field, type `0`.

12 In the **y maximum** text field, type `2`.

Point Graph 1

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

2 Select Point 4 only.

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

4 In the **Expression** text field, type `-solid3.evol`.

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

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

7 In the **Expression** text field, type `-solid3.eZZ`.

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

9 In the **Volumetric Strain** toolbar, click **Plot**.

Deviatoric Stress

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

2 In the **Settings** window for **1D Plot Group**, type `Deviatoric Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Triaxial Test]/ Solution 7 (sol7)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

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

7 Select the **y-axis label** checkbox. In the associated text field, type `Deviatoric stress (MPa)`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `12`.

11 In the **y minimum** text field, type `0`.

12 In the **y maximum** text field, type `0.8`.

Point Graph 1

1 Right-click **Deviatoric Stress** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid3.mises`.

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

6 In the **Expression** text field, type `-solid3.eZZ`.

7 From the **Unit** list, choose **%**.

8 In the **Deviatoric Stress** toolbar, click **Plot**.

Stress Path 2

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

2 In the **Settings** window for **1D Plot Group**, type `Stress Path 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Triaxial Test]/ Solution 7 (sol7)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Mean stress (MPa)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Deviatoric stress (MPa)`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `2`.

11 In the **y minimum** text field, type `0`.

12 In the **y maximum** text field, type `0.8`.

Point Graph 1

1 Right-click **Stress Path 2** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid3.mises`.

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

6 In the **Expression** text field, type `solid3.pm`.

7 In the **Stress Path 2** toolbar, click **Plot**.

Deviatoric Stress, Stress Path 2, Volumetric Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Volumetric Strain**, **Deviatoric Stress**, and **Stress Path 2**.

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

Triaxial Test

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

Mean Stress 1

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

2 In the **Settings** window for **1D Plot Group**, type `Mean Stress 1` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Data** section. From the **Dataset** list, choose **Study [Constrained Swelling Test]/ Solution 10 (sol10)**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Suction (MPa)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Mean stress (MPa)`.

Point Graph 1

1 Right-click **Mean Stress 1** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `solid4.pm`.

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

6 In the **Expression** text field, type `solid4.epm1.ss`.

7 Click to expand the **Coloring and Style** section. 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
`Mean stress`

Point Graph 2

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

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

3 In the **Expression** text field, type `P_analytical`.

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

Legends
`Mean stress, analytical`

5 Locate the **Coloring and Style** section. From the **Color** list, choose **From theme**.

6 Find the **Line style** subsection. From the **Line** list, choose **None**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

9 In the **Number** text field, type `20`.

10 In the **Mean Stress 1** toolbar, click **Plot**.

Void Ratio and Volumetric Strain

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

2 In the **Settings** window for **1D Plot Group**, type `Void Ratio and Volumetric Strain` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Constrained Swelling Test]/ Solution 10 (sol10)**.
4 Locate the **Title** section. From the **Title type** list, choose **None**.
5 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.
6 Select the **x-axis label** checkbox. In the associated text field, type `Suction (MPa)`.
7 Select the **y-axis label** checkbox. In the associated text field, type `Void ratio (1)`.
8 Select the **Secondary y-axis label** checkbox. In the associated text field, type `Volumetric strain (%)`.
9 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
10 In the **x minimum** text field, type `10`.
11 In the **x maximum** text field, type `105`.
12 In the **y minimum** text field, type `0.4`.
13 In the **y maximum** text field, type `0.8`.
14 In the **Secondary y minimum** text field, type `-10`.
15 In the **Secondary y maximum** text field, type `10`.

Point Graph 1

1 Right-click **Void Ratio and Volumetric Strain** and choose **Point Graph**.
2 Select Point 4 only.
3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `solid4.evoid`.
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `solid4.epm1.ss`.
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
`Void ratio`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.
2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `solid.evol`.

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

5 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

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

Legends
Volumetric strain

Void Ratio and Volumetric Strain

1 In the **Model Builder** window, collapse the **Results** > **Void Ratio and Volumetric Strain** node.

2 In the **Model Builder** window, click **Void Ratio and Volumetric Strain**.

3 In the **Void Ratio and Volumetric Strain** toolbar, click **Plot**.

Mean Stress 1, Void Ratio and Volumetric Strain

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

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

Constrained Swelling Test

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