Created in COMSOL Multiphysics 6.1

Shallow Foundation on Unsaturated Soil

Introduction

The hydromechanical behavior of unsaturated soil, such as settlement and heave, is an important topic in geotechnical engineering. Settlement and heave largely depends on the saturation and suction in the soil and on the loading conditions. To study these phenomena, this example looks at a shallow foundation resting on an unsaturated soil. The problem setting and geometry is inspired by the example presented in Ref. 1. In order to demonstrate the settlement and heave of the unsaturated soil, the Modified Cam-Clay model (MCC) and Extended Barcelona Basic model (BBMx) are used. Of these two constitutive models, the latter is more suitable to model unsaturated soils since it includes the effect of suction in its constitutive relationship.

Model Definition

In this example, a 10 m wide and 5 m deep soil stratum is studied. A 1 m wide footing is placed on top of the layer, which applies an incrementally increasing footing pressure. Initially, the ground water level is 3 m below surface. This level then defines the phreatic line under which the soil is saturated, and above which it is unsaturated. In order to study the effects of wetting, the ground water level is increased until the soil is fully saturated after the full footing pressure is applied.

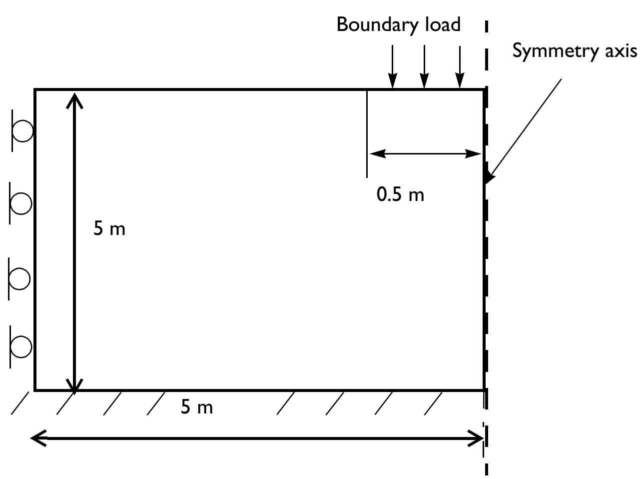

Figure 1: Dimensions, boundary conditions, and pressure load for the unsaturated soil.

SOIL PROPERTIES

The properties of the soil stratum are given in Table 1.

TABLE 1: MATERIAL PROPERTIES.

Property	Variable	Value
Poisson's ratio	ν	0.2
Soil density	ρ_s	1743 kg/m^3
Water density	ρ_w	1000 kg/m^3
Dynamic viscosity of water	μ_w	0.001 Pa.s
Swelling index	κ	0.006
Swelling index for changes in suction	κ_s	0.008
Compression index at saturation	λ	0.22
Compression index for changes in suction	λ_s	0.123
Slope of critical state line	M	1.24
Weight parameter	w	0.4
Soil stiffness parameter	m	50 kPa
Plastic potential smoothing parameter	b_s	10
Tension to suction ratio	k_s	0.6
Initial yield value for suction	s_y	100 kPa
Initial void ratio	e_0	1.8
Reference pressure	p_{ref}	18 kPa
Initial consolidation pressure	p_{c0}	80 kPa

CONSTRAINTS AND LOADS

- The soil stratum is supported by a rigid and perfectly rough base. Apply a fixed constraint on the lower horizontal boundary.
- Use roller boundary conditions on the left vertical boundary, symmetry to the right.
- A pressure head of 3 m is assigned in the Richard's Equation interface. This pressure head is afterwards increased to 5 m.
- The gravity load is applied using a **Gravity** node. The pore pressure in the saturated region of the soil sample is applied using an **External Stress** node.
- A boundary load is applied on top of the soil layer to model the weight of the footing, see Figure 1.

Results and Discussion

Figure 2 shows the pore pressure at different ground water levels in the soil. Negative values of the pore pressure indicate regions that are unsaturated.

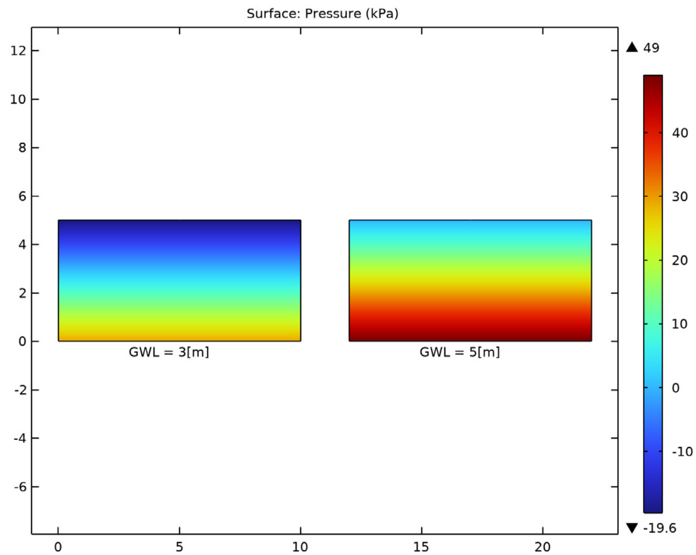

Figure 2: Pore pressure at different ground water levels.

For saturated soils, both the MCC and the BBMx models use the *effective stress principle* to account for the pore pressure; while for unsaturated soils, the BBMx model accounts for the negative pore pressure by including suction as a model parameter in the yield function and plastic potential. Unlike the BBMx model, suction is not included in the MCC model, which makes it less suitable for modeling unsaturated soils.

The distribution of volumetric plastic strain obtained with both material models, at a ground water level equal to 3 m, is shown in Figure 3. With the MCC model, a larger region of the soil underneath the footing is subjected to plastic deformation, as compared to the results from the BBMx model. The same observation is made in Ref. 1. However, when the ground water level increases, the reduction in suction results in additional plastic strains with the BBMx model. In contrast, as shown in Figure 4, no significant additional plastic strains are observed when the MCC model is used; negligible changes in plastic strain can, however, be observed due to changes in the effective stress. The distribution of von Mises stress for the fully saturated soil is shown in Figure 5 and Figure 6. It is distinctively different when comparing the two soil models. These results emphasize the importance of including suction as a constitutive parameter in soil models intended for modeling partially saturated soils.

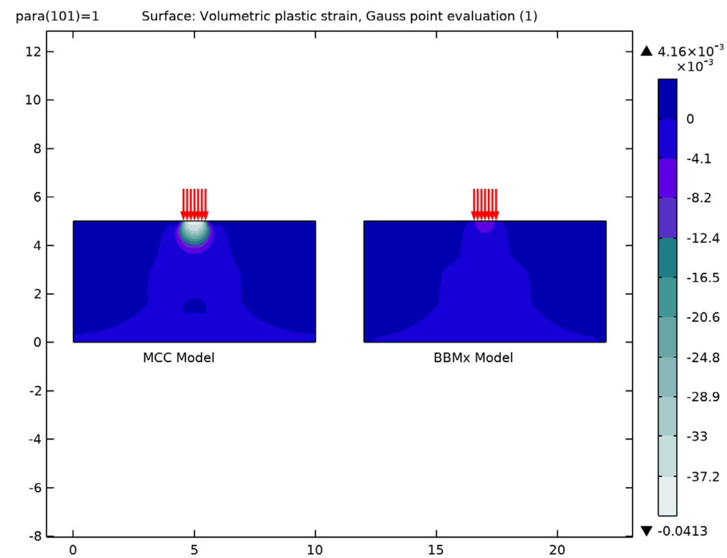

Figure 3: Volumetric plastic strain for a ground water level equal to 3 m.

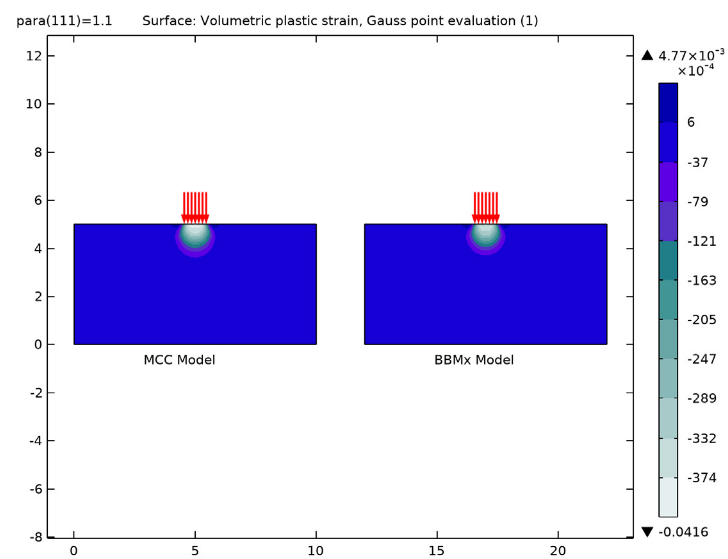

Figure 4: Volumetric plastic strain for a ground water level equal to 5 m.

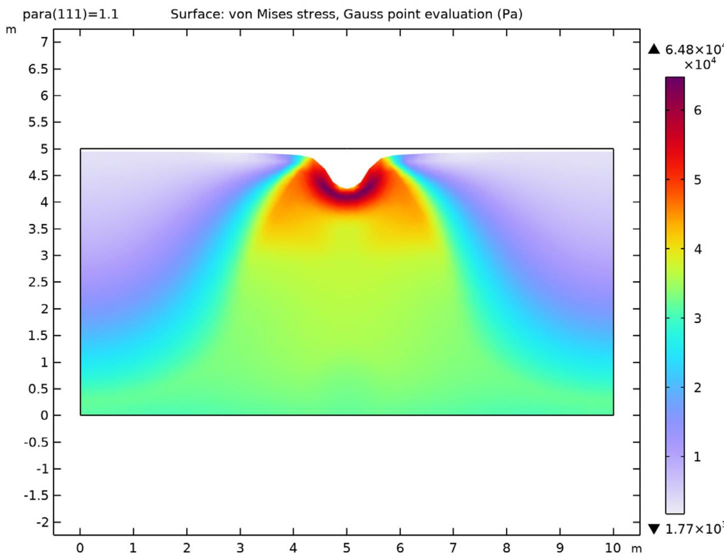

Figure 5: Distribution of von Mises stress with the MCC model for a ground water level equal to 5 m.

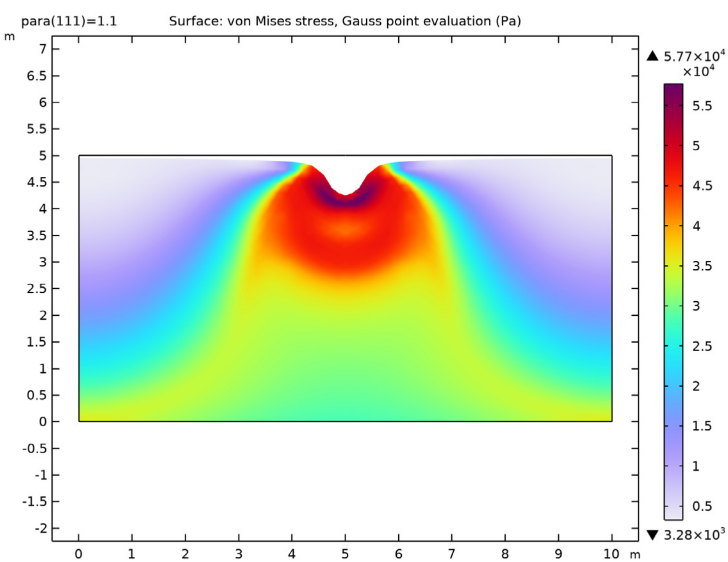

Figure 6: Distribution of von Mises stress with the BBMx model for a ground water level equal to 5 m.

The curve of footing pressure versus settlement is shown in Figure 7. The first stage of the analysis is done at a water level of 3 m, then, the footing pressure is incrementally increased up to 130 kPa. In the elastic regime both models give the same response, but as plastic strains develop, the two models differ significantly. The BBMx model shows smaller settlement as compared to the MCC model; this indicates that if suction is not taken in to account, the settlement is overestimated.

When the ground water level gradually increases to the surface level, the two material models react differently, see Figure 7. The MCC model shows a reduction in the displacement of the footing, while the BBMx model instead shows a large increase in the footing displacement. With the MCC model, the rise in ground water level causes a reduction in effective stress, which gives an overall positive displacement of the soil surface called *heave*, see Figure 8. Both the footing and the adjacent soil display heaving with the MCC model. With the BBMx model, the footing shows additional settlement due to the reduction in both suction and effective stress. Note that Figure 8 shows the vertical displacement due to wetting only. The behavior shown in Figure 7 and Figure 8 is also reported in Ref. 1. For the BBMx model, the additional footing settlement due to wetting is referred to as *soil collapse due to loss of capillary cohesion*.

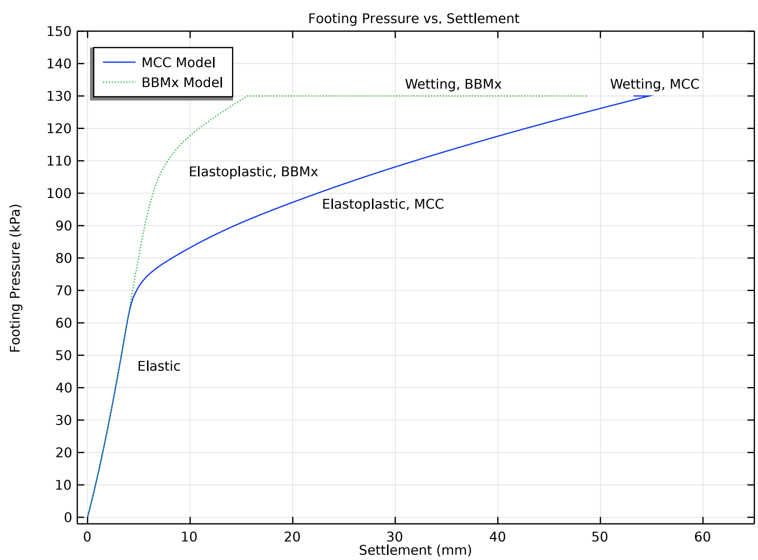

Figure 7: Footing pressure versus settlement.

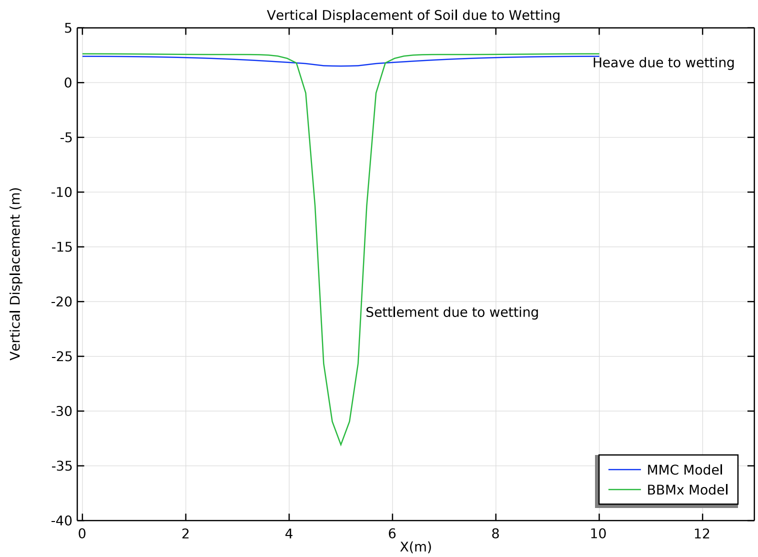

Figure 8: Vertical displacement of the stratum surface due to wetting.

Notes About the COMSOL Implementation

The model setup neither includes transient phenomena nor the effect that the deformation has on the pore pressure. This means that it would be possible to model this problem without the Richards' Equation interface by defining the pore pressure in the saturated region using an analytical function. However, the current setup can be easily extended to more complex multiphysics scenarios.

A linear discretization is used for the Solid Mechanics interface to achieve numerical stability for the nonlinear plasticity problem. A dense mesh is used in the domain to maintain good accuracy.

The Cam-Clay family of soil models, like the MCC or BBMx models, 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.

Reference

1. A.A. Abed and P.A. Vermeer, "Numerical Simulation of Unsaturated Soil Behavior," *International Journal of Computer Applications in Technology*, vol. 34, no. 1, 2009.

Application Library path: `Geomechanics_Module/Soil/settlement_analysis`

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**.
2 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Richards' Equation (dl)**.
3 Right-click and choose **Add Physics**.
4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5 Right-click and choose **Add Physics**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
8 Click **Done**.

GEOMETRY 1

Model parameters are available in the appended text file.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `settlement_analysis_parameters.txt`.

Footing Pressure

1 In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
2 In the **Settings** window for **Interpolation**, type `Footing Pressure` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `F_P`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	130
1.1	130

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
F_P	kPa

Ground Water Level

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Ground Water Level` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `GWL`.

4 In the table, enter the following settings:

t	f(t)
0	3
1	3
1.1	5

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
GWL	m

Initial Suction Profile

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Initial Suction Profile` in the **Label** text field.

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

4 Locate the **Definition** section. In the **Expression** text field, type `rhow*g_const*(Y-3)*(Y>=3)`.

5 In the **Arguments** text field, type `Y`.

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

Argument	Unit
Y	m

7 In the **Function** text field, type `Pa`.

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

Argument	Lower limit	Upper limit	Unit
Y	0	5	m

DEFINITIONS

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

Model variables are available in the appended text file.

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 Click **Load from File**.

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

Create half of the geometry by exploiting symmetry.

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`.

4 In the **Height** text field, type 5.

Add a line segment to represent the foundation.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type 4.5.

5 In the **y** text field, type 5.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type 5.

8 In the **y** text field, type 5.

9 Click **Build Selected**.

Add points on side boundaries to represent the initial ground water level.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

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

3 In the **y** text field, type 3.

MATERIALS

Add a **Porous Material** that contains information about the fluid and porous matrix properties together with the structural properties.

Porous Material 1 (pmat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Porous Material**.

Continue with setting up the physics. After that, the software automatically detects which material properties are required. First, change the discretization to **Linear**.

RICHARDS' EQUATION (DL)

1 In the **Settings** window for **Richards' Equation**, click to expand the **Discretization** section.

2 From the **Pressure** list, choose **Linear**.

Unsaturated Porous Medium 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Richards' Equation (dl)** click **Unsaturated Porous Medium 1**.

2 In the **Settings** window for **Unsaturated Porous Medium**, locate the **Porous Medium** section.

3 From the **Storage model** list, choose **User defined**.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Retention Model** section.

3 From the **Retention model** list, choose **User defined**. In the S_e text field, type `Se`.

4 In the θ_l text field, type `S_res*phi0+Se*(phi0-S_res*phi0)`.

5 In the C_m text field, type `Cm`.

6 In the κ_r text field, type `k_rel`.

7 In the θ_r text field, type `S_res*phi0`.

Pressure Head 1

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

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for **Pressure Head**, locate the **Pressure Head** section.

4 In the H_{p0} text field, type `GWL(para)-Y`.

Symmetry 1

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

2 Select Boundary 6 only.

Continue with setting up the **Solid Mechanics** interface. Change the discretization to **Linear**.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

3 From the **Displacement field** list, choose **Linear**.

Modified Cam-Clay Model (MCC)

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 Model (MCC)` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Elastoplastic Soil Material** section. From the e_0 list, choose **From material**.

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

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

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 **Pore pressure**.

4 In the p_A text field, type `PorePressure`.

5 In the p_{ref} text field, type `0`.

6 From the α_B list, choose **User defined**. In the associated text field, type `1`.

Go to the material node and assign the required material properties.

MATERIALS

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rhos`	kg/m³	Basic
Young's modulus	E		Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`Nu`	1	Young's modulus and Poisson's ratio
Swelling index	kappaSwelling	`kappa`	1	Cam-Clay
Initial void ratio	evoid0	`e0`	1	Cam-Clay
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`k`	m²	Basic
Slope of critical state line	M	`Mb`	1	Cam-Clay
Compression index	lambdaComp	`lambda`	1	Cam-Clay

SOLID MECHANICS (SOLID)

Extended Barcelona Basic Model (BBMx)

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

2 In the **Settings** window for **Elastoplastic Soil Material**, type `Extended Barcelona Basic Model (BBMx)` in the **Label** text field.

3 Locate the **Elastoplastic Soil Material** section. From the **Material model** list, choose **Extended Barcelona Basic**.

4 From the M list, choose **From material**.

5 From the e_0 list, choose **From material**.

6 In the s_0 text field, type `InitSuction(Y)`.

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

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

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

10 Select Domain 1 only.

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 **Pore pressure**.

4 In the p_{A} text field, type `PorePressure`.

5 In the p_{ref} text field, type `0`.

6 From the α_{B} list, choose **User defined**. In the associated text field, type `1`.

Go to the material node and assign the required material properties.

MATERIALS

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E		Pa	Young's modulus and Poisson's ratio
Void ratio at reference pressure	evoidref		1	Cam-Clay
Plastic potential smoothing parameter	bB	`bb`	1	Barcelona Basic
Compression index at saturation	lambdaComp0	`lambda`	1	Barcelona Basic
Weight parameter	wB	`wb`	1	Barcelona Basic
Initial yield value for suction	sy0	`sy`	Pa	Barcelona Basic
Slope of critical state line	M	`Mb`	1	Barcelona Basic
Compression index for changes in suction	lambdaCompss	`lambda_s`	1	Barcelona Basic
Swelling index for changes in suction	kappaSwellings	`kappa`	1	Barcelona Basic
Soil stiffness parameter	mB	`mb`	Pa	Barcelona Basic
Tension to suction ratio	kB	`kb`	1	Barcelona Basic
Swelling index	kappaSwelling	`kappa`	1	Barcelona Basic
Initial void ratio	evoid0	`e0`	1	Barcelona Basic
Density	rho	`rhos`	kg/m³	Basic

4 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Fluid 1 (pmat1.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat1.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rhow	kg/m³	Basic
Dynamic viscosity	mu	muw	Pa·s	Basic
Porosity	epsilon	1	1	Porous model

SOLID MECHANICS (SOLID)

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

Fixed Constraint 1

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

2 Select Boundary 2 only.

Roller 1

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

2 Select Boundaries 1 and 3 only.

Symmetry 1

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

2 Select Boundary 6 only.

Boundary Load 1

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

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
-F_P(para)	y

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

Use finer mesh in the mesh control domain.

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Free Triangular 1

In the **Model Builder** window, right-click **Free Triangular 1** and choose **Build All**.

STUDY: MCC

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

2 In the **Settings** window for **Study**, type `Study: MCC` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: MCC** 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** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)> Extended Barcelona Basic Model (BBMx)**.

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

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

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.01,1.1)`	

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

10 Click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study: MCC/Solution 1 (sol1)>Solid Mechanics> Volumetric Plastic Strain (solid)**, **Study: MCC/Solution 1 (sol1)>Solid Mechanics> Current Void Volume Fraction (solid)**, and **Study: MCC/Solution 1 (sol1)>Solid Mechanics> Applied Loads (solid)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

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

STUDY: BBMX

1 In the **Settings** window for **Study**, type `Study: BBMx` in the **Label** text field.

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

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.01,1.1)`	

Customize the solver settings in order to achieve better convergence.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study: BBMx>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Parametric 1**.

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

5 Select the **Tuning of step size** check box.

6 In the **Minimum step size** text field, type `0.0001`.

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

RESULTS

Mirror 2D 1

1. In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.
2. In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.
3. In row **Point 1**, set **X** to 5.
4. In row **Point 2**, set **X** to 5.
5. Click to expand the **Advanced** section. Select the **Remove elements on the symmetry axis** check box.

Mirror 2D 2

1. Right-click **Mirror 2D 1** and choose **Duplicate**.
2. In the **Settings** window for **Mirror 2D**, locate the **Data** section.
3. From the **Dataset** list, choose **Study: BBMx/Solution 2 (sol2)**.

Degree of Saturation at GWL = 3[m]

1. In the **Model Builder** window, under **Results** click **Flownet (dl)**.
2. In the **Settings** window for **2D Plot Group**, type `Degree of Saturation at GWL = 3[m]` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.
4. From the **Parameter value (para)** list, choose **1**.
5. In the **Model Builder** window, expand the **Degree of Saturation at GWL = 3[m]** node.

Contour 1, Streamline 1

1. In the **Model Builder** window, under **Results>Degree of Saturation at GWL = 3[m]**, Ctrl-click to select **Contour 1** and **Streamline 1**.
2. Right-click and choose **Delete**.

Surface 1

1. In the **Model Builder** window, right-click **Degree of Saturation at GWL = 3[m]** and choose **Surface**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Richards' Equation>Retention model>dl.Se - Effective saturation**.

Degree of Saturation at GWL = 3[m]

1. In the **Model Builder** window, click **Degree of Saturation at GWL = 3[m]**.
2. In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **New view**.

4 In the **Degree of Saturation at GWL = 3[m]** toolbar, click **Plot**.

Streamline 1

1 In the **Model Builder** window, expand the **Pressure (dl)** node.

2 Right-click **Results>Pressure (dl)>Streamline 1** and choose **Delete**.

Surface

1 In the **Model Builder** window, under **Results>Pressure (dl)** click **Surface**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Parameter value (para)** list, choose **1**.

5 Locate the **Expression** section. From the **Unit** list, choose **kPa**.

Surface 2

1 Right-click **Surface** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **From parent**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface**.

Translation 1

1 Right-click **Surface 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type 12.

4 In the **Pressure (dl)** toolbar, click **Plot**.

Pressure (dl)

In the **Model Builder** window, under **Results** click **Pressure (dl)**.

Table Annotation 1

1 In the **Pressure (dl)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
2.5	0	GWL = 3[m]
14.5	0	GWL = 5[m]

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Pressure (dl)

1 In the **Model Builder** window, click **Pressure (dl)**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 Click to expand the **Title** section. Find the **Solution** subsection. Clear the **Solution** check box.

5 From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Surface: Pressure (kPa)`.

7 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

8 Locate the **Plot Settings** section. From the **View** list, choose **View 2D 3**.

9 In the **Pressure (dl)** toolbar, click **Plot**.

Stress, MCC

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 2D 4**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress, MCC** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Smoothing threshold** list, choose **None**.

4 In the **Stress, MCC** toolbar, click **Plot**.

Stress, BBMx

1 In the **Model Builder** window, right-click **Stress, MCC** and choose **Duplicate**.

2 Drag and drop **Stress, MCC 1** below **Stress, MCC**.

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

4 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 2**.

5 In the **Stress, BBMx** toolbar, click **Plot**.

Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]

1 In the **Model Builder** window, under **Results** click **Volumetric Plastic Strain (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Parameter value (para)** list, choose **1**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 2

1 In the **Model Builder** window, expand the **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]** node.

2 Right-click **Results>Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]>Surface 1** and choose **Duplicate**.

3 In the **Settings** window for **Surface**, locate the **Data** section.

4 From the **Dataset** list, choose **Mirror 2D 2**.

5 From the **Parameter value (para)** list, choose **1**.

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

7 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Translation 1

1 Right-click **Surface 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `12`.

Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]

In the **Model Builder** window, under **Results** click **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]**.

Table Annotation 1

1 In the **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
2.5	-0.2	MCC Model
14.5	-0.2	BBMx Model

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Arrow Line 1

1 Right-click **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Load>solid.F_Ax,solid.F_Ay - Load (spatial frame)**.

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

4 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

Arrow Line 2

Right-click **Arrow Line 1** and choose **Duplicate**.

Translation 1

1 In the **Model Builder** window, right-click **Arrow Line 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type 12.

Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]

1 In the **Model Builder** window, under **Results** click **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]**.

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

3 From the **View** list, choose **View 2D 3**.

4 In the **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]** toolbar, click **Plot**.

Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]

1 Right-click **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]** and choose **Duplicate**.

2 Drag and drop **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa] 1** below **Volumetric Plastic Strain at GWL = 3[m] and F_P = 130[kPa]**.

3 In the **Settings** window for **2D Plot Group**, type `Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]` in the **Label** text field.

4 Locate the **Data** section. From the **Parameter value (para)** list, choose **1.1**.

Surface 2

1 In the **Model Builder** window, expand the **Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]** node, then click **Surface 2**.

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

3 From the **Parameter value (para)** list, choose **1.1**.

Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]

1 In the **Model Builder** window, click **Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]**.

2 In the **Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa]** toolbar, click **Plot**.

Void Ratio at GWL = 3[m] and F_P = 130[kPa]

1 In the **Model Builder** window, under **Results** click **Current Void Volume Fraction (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Void Ratio at GWL = 3[m] and F_P = 130[kPa]` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Parameter value (para)** list, choose **1**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Void Ratio at GWL = 3[m] and F_P = 130[kPa]** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

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

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Traffic>Traffic** in the tree.

6 Click **OK**.

Surface 2

1 Right-click **Results>Void Ratio at GWL = 3[m] and F_P = 130[kPa]>Surface 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Mirror 2D 2**.

4 From the **Parameter value (para)** list, choose **1**.

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

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Translation 1

1 Right-click **Surface 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `12`.

Table Annotation 1

In the **Model Builder** window, under **Results>**
Volumetric Plastic Strain at GWL = 5[m] and F_P = 130[kPa] right-click **Table Annotation 1** and choose **Copy**.

Table Annotation 1

In the **Model Builder** window, right-click **Void Ratio at GWL = 3[m] and F_P = 130[kPa]** and choose **Paste Table Annotation**.

Void Ratio at GWL = 3[m] and F_P = 130[kPa]

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

2 From the **View** list, choose **View 2D 3**.

3 In the **Void Ratio at GWL = 3[m] and F_P = 130[kPa]** toolbar, click **Plot**.

Void Ratio at GWL = 5[m] and F_P = 130[kPa]

1 Right-click **Void Ratio at GWL = 3[m] and F_P = 130[kPa]** and choose **Duplicate**.

2 Drag and drop **Void Ratio at GWL = 3[m] and F_P = 130[kPa] 1** below **Void Ratio at GWL = 3[m] and F_P = 130[kPa]**.

3 In the **Settings** window for **2D Plot Group**, type `Void Ratio at GWL = 5[m] and F_P = 130[kPa]` in the **Label** text field.

4 Click **Plot Last**.

Surface 2

1 In the **Model Builder** window, expand the **Void Ratio at GWL = 5[m] and F_P = 130[kPa]** node, then click **Surface 2**.

2 In the **Settings** window for **Surface**, click **Plot Last**.

In order to plot the characteristic curve of the footing pressure versus settlement, offset the initial deformation due to pore pressure and gravity.

Footing Pressure vs. Settlement

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 Drag and drop below **Void Ratio at GWL = 5[m] and F_P = 130[kPa]**.

3 In the **Settings** window for **1D Plot Group**, type `Footing Pressure vs. Settlement` in the **Label** text field.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

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

7 Select the **y-axis label** check box. In the associated text field, type `Footing Pressure (kPa)`.

8 Locate the **Axis** section. Select the **Manual axis limits** check box.

9 In the **x maximum** text field, type `65`.

10 In the **y minimum** text field, type `-2`.

11 In the **y maximum** text field, type `150`.

12 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Footing Pressure vs. Settlement** and choose **Point Graph**.

2 Select Point 6 only.

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

4 In the **Expression** text field, type `F_P(para)`.

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

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

7 In the **Expression** text field, type `abs(v-withsol('sol1',v,setval(para,0)))`.

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

9 Click to expand the **Coloring and Style** section. From the **Width** list, choose **1**.

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

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

12 In the table, enter the following settings:

Legends
`MCC Model`

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 **Dataset** list, choose **Study: BBMx/Solution 2 (sol2)**.

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(v-withsol('sol2',v,setval(para,0)))`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

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

Legends
`BBMx Model`

Footing Pressure vs. Settlement

In the **Model Builder** window, click **Footing Pressure vs. Settlement**.

Table Annotation 1

1 In the **Footing Pressure vs. Settlement** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	**y-coordinate**	**Annotation**
4	50	`Elastic`
22	100	`Elastoplastic, MCC`
9	110	`Elastoplastic, BBMx`
30	137	`Wetting, BBMx`
50	137	`Wetting, MCC`

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Footing Pressure vs. Settlement

1 In the **Model Builder** window, click **Footing Pressure vs. Settlement**.

2 In the **Footing Pressure vs. Settlement** toolbar, click **Plot**.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to 10.

5 In row **Point 1**, set **y** to 5.

6 In row **Point 2**, set **y** to 5.

Cut Line 2D 2

1 Right-click **Cut Line 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 2D 2**.

Vertical Displacement of Soil due to Wetting

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

2 Drag and drop below **Footing Pressure vs. Settlement**.

3 In the **Settings** window for **1D Plot Group**, type `Vertical Displacement of Soil due to Wetting` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

5 From the **Parameter selection (para)** list, choose **Last**.

6 Locate the **Title** section. From the **Title type** list, choose **Label**.

7 Locate the **Plot Settings** section.

8 Select the **x-axis label** check box. In the associated text field, type `X(m)`.

9 Select the **y-axis label** check box. In the associated text field, type `Vertical Displacement (m)`.

10 Locate the **Axis** section. Select the **Manual axis limits** check box.

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

12 In the **x maximum** text field, type `13`.

13 In the **y minimum** text field, type `-40`.

14 In the **y maximum** text field, type `5`.

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

Line Graph 1

1 Right-click **Vertical Displacement of Soil due to Wetting** and choose **Line Graph**.

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

3 In the **Expression** text field, type `v-withsol('sol1',v,setval(para,1))`.

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

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

6 In the **Expression** text field, type `cln1x`.

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

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

9 In the table, enter the following settings:

Legends
MMC Model

Line Graph 2

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

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

3 From the **Dataset** list, choose **Cut Line 2D 2**.

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

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `v-withsol('sol2',v, setval(para,1))`.

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

Legends
BBMx Model

Vertical Displacement of Soil due to Wetting

In the **Model Builder** window, click **Vertical Displacement of Soil due to Wetting**.

Table Annotation 1

1 In the **Vertical Displacement of Soil due to Wetting** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
`9.7`	`2.8`	`Heave due to wetting`
`5.3`	`-20`	`Settlement due to wetting`

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Vertical Displacement of Soil due to Wetting

1 In the **Model Builder** window, click **Vertical Displacement of Soil due to Wetting**.

2 In the **Vertical Displacement of Soil due to Wetting** toolbar, click **Plot**.

Volumetric Suction Strain due to Wetting

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 Drag and drop below **Vertical Displacement of Soil due to Wetting**.

3 In the **Settings** window for **1D Plot Group**, type `Volumetric Suction Strain due to Wetting` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study: BBMx/Solution 2 (sol2)**.

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

6 In the **Parameter indices (1-111)** text field, type `range(100,1,111)`.

7 Locate the **Title** section. From the **Title type** list, choose **Label**.

8 Locate the **Plot Settings** section.

9 Select the **x-axis label** check box. In the associated text field, type `Suction (kPa)`.

10 Select the **y-axis label** check box. In the associated text field, type `Volumetric Suction Strain (1)`.

Point Graph 1

1 Right-click **Volumetric Suction Strain due to Wetting** and choose **Point Graph**.

2 Select Point 5 only.

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

4 In the **Expression** text field, type `solid.epsm2.evols`.

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

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

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

8 In the **Volumetric Suction Strain due to Wetting** toolbar, click **Plot**.