



Model created in COMSOL Multiphysics 6.4

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.

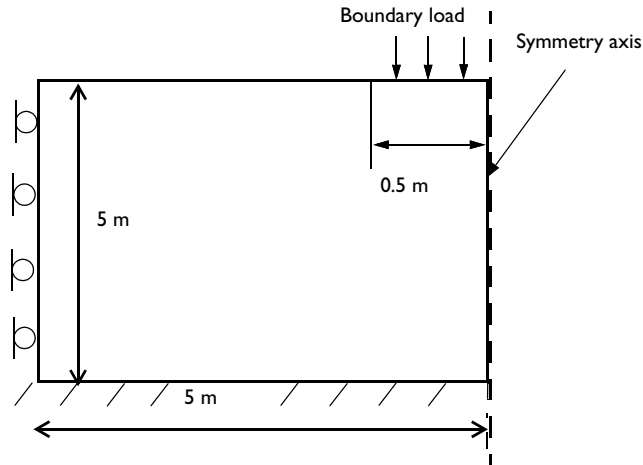


Figure 1: Dimensions, boundary conditions, and pressure load for the unsaturated soil.

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_b	10
Tension to suction ratio	k_b	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 afterward 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.

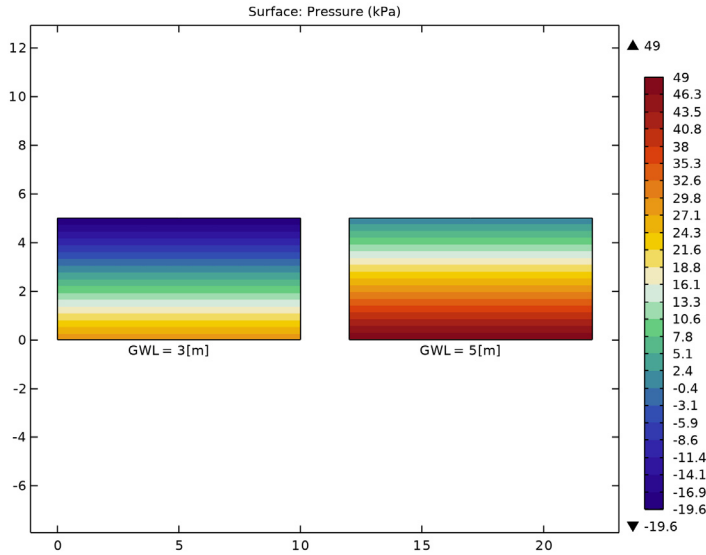


Figure 2: Pore pressure at different ground water levels.

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.

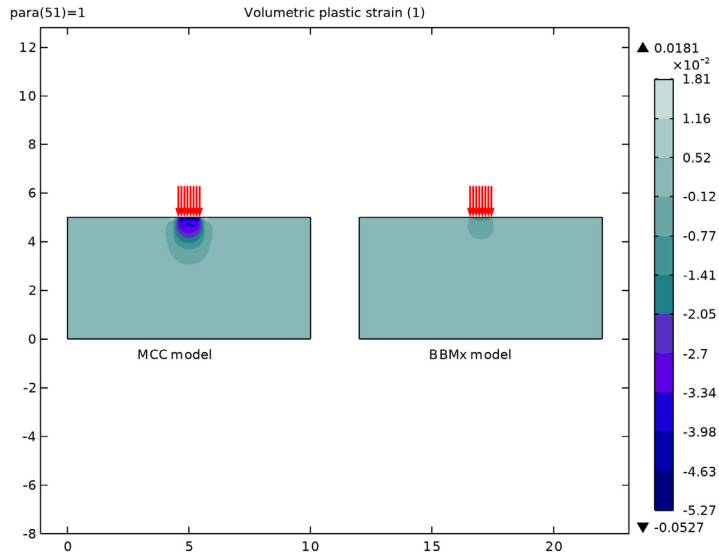


Figure 3: Volumetric plastic strain for a ground water level equal to 3 m.

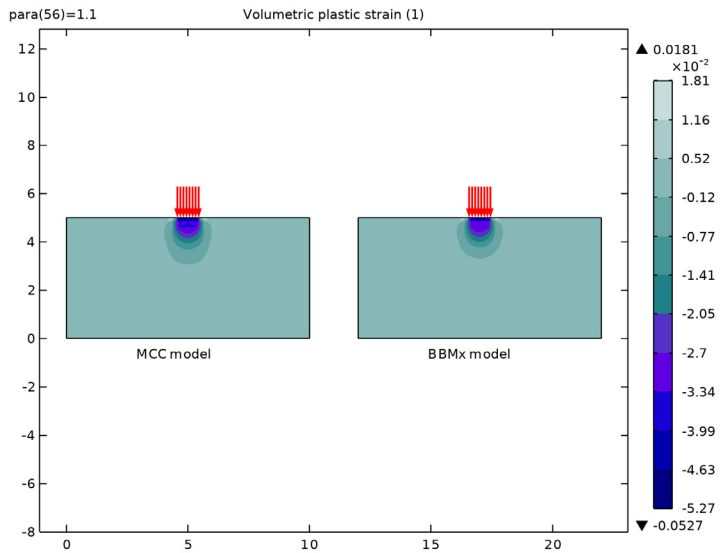


Figure 4: Volumetric plastic strain for a ground water level equal to 5 m.

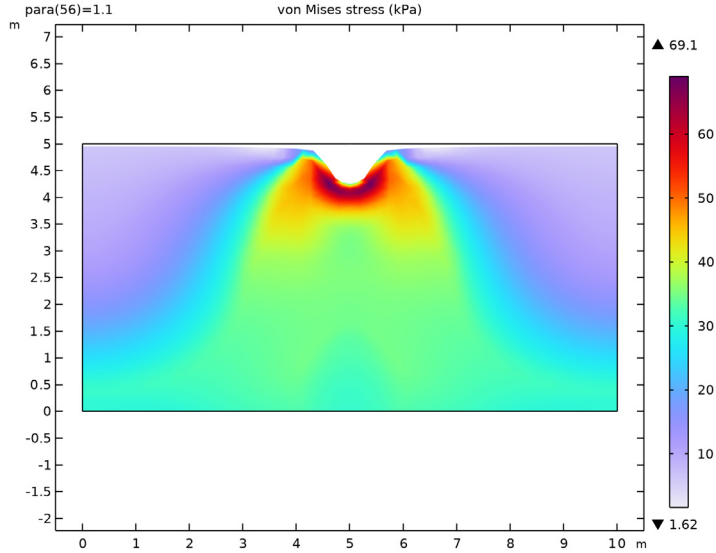


Figure 5: Distribution of von Mises stress with the MCC model for a ground water level equal to 5 m.

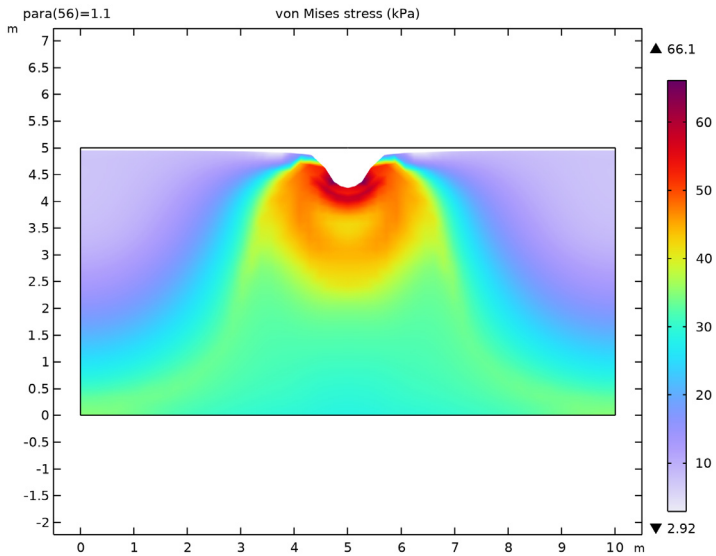


Figure 6: Distribution of von Mises stress with the BBMx model for a ground water level equal to 5 m.

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*.

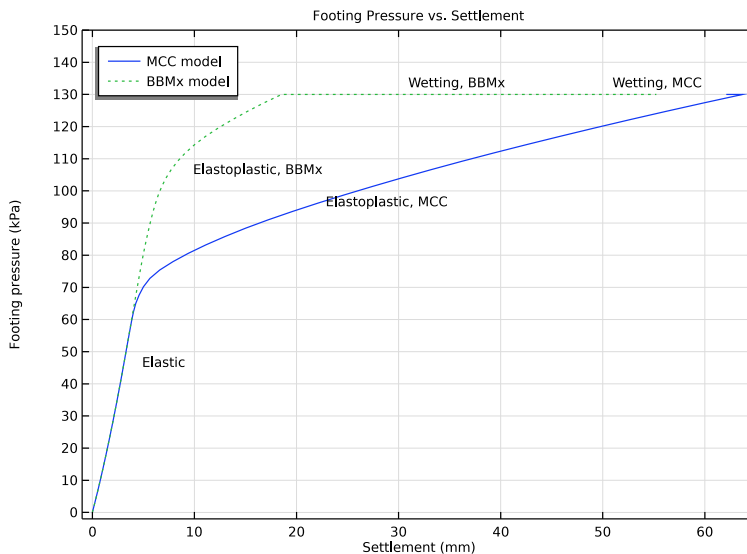


Figure 7: Footing pressure versus settlement.

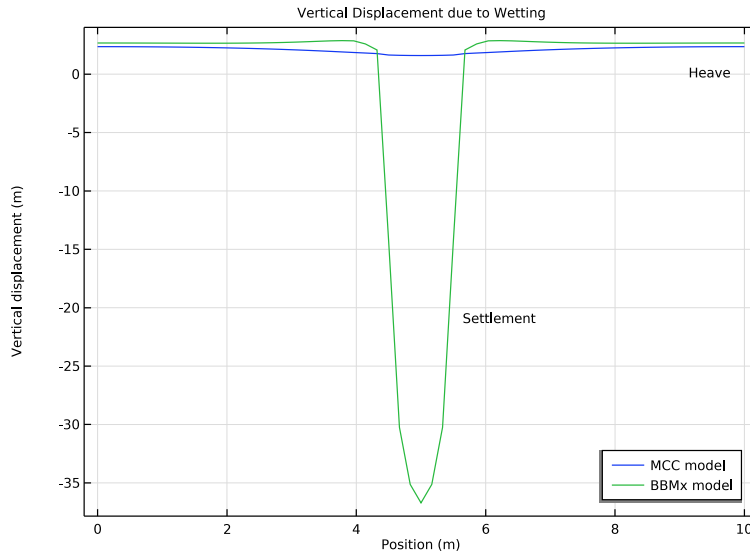


Figure 8: Vertical displacement of the stratum surface due to wetting.

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 I


Model parameters are available in the appended text file.

GLOBAL DEFINITIONS

Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **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  **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  **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 $\text{rho}w * g_const * (Y - 3) * (Y >= 3)$.
- 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:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	Y	0	5	0	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 S_e .
- 4 In the θ_l text field, type $S_{res}*\phi_0+Se*(\phi_0-S_{res}*\phi_0)$.
- 5 In the C_m text field, type C_m .
- 6 In the κ_r text field, type k_{rel} .
- 7 In the θ_r text field, type $S_{res}*\phi_0$.

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 $GWL(\text{para}) - Y$.

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 $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.
- 5 Find the **Parameters** subsection. From the M list, choose **From material**.

- 6 In the p_{ref} text field, type pref.
- 7 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/m ³	Basic
Poisson's ratio	nu	Nu		Critical state model
Swelling index	kappaSwelling	kappa		Modified Cam-clay
Initial void ratio	evoid0	e0		Critical state model
Permeability	kappa_iso ; kappa _{ii} = kappa_iso, kappa _{ij} = 0	k	m ²	Basic
Slope of critical state line	M	MB		Critical state model
Compression index	lambdaComp	lambda		Modified 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 Select Domain 1 only.
- 5 From the $\Gamma(\theta)$ list, choose **Matsuoka–Nakai**.
- 6 Find the **Parameters** subsection. From the M list, choose **From material**.
- 7 In the s_0 text field, type InitSuction(Y).
- 8 In the s text field, type Suction.
- 9 In the p_{ref} text field, type pref.
- 10 In the p_{c0} text field, type pc0.

External Stress I

- 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 I (pmatI)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Materials** click **Porous Material I (pmatI)**.
- 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
Plastic potential smoothing parameter	bb	bB	l	Barcelona Basic
Swelling index for changes in suction	kappaSwellingsu	kappa_s	l	Barcelona Basic
Weight parameter	wb	wB	l	Barcelona Basic
Initial yield value for suction	sy0	syB0	Pa	Barcelona Basic
Soil stiffness parameter	mb	mB	Pa	Barcelona Basic
Initial void ratio	evoid0b	e0	l	Barcelona Basic
Tension to suction ratio	kb	kB	l	Barcelona Basic
Compression index for changes in suction	lambdaCompsu	lambda_s	l	Barcelona 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/m ³	Basic
Dynamic viscosity	mu	muW	Pa·s	Basic
Porosity	epsilon	l	l	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

Mapped 1

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

Size

- 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**.

Mapped 1


In the **Model Builder** window, right-click **Mapped 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 **Study Settings** section.
- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type 0.0001.
- 5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.
- 6 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid) > Extended Barcelona Basic Model (BBMx)**.
- 7 Right-click and choose **Disable**.
- 8 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 9 Click  **Add**.
- 10 In the table, enter the following settings:



Parameter name	Parameter value list	Parameter unit
para (Parameter)	range(0,0.02,1.1)	

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


RESULT TEMPLATES

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study: MCC/Solution 1 (sol1) > Solid Mechanics > Volumetric Plastic Strain (solid)**, **Study: MCC/Solution 1 (sol1) > Solid Mechanics > Void Ratio (solid)**, and **Study: MCC/Solution 1 (sol1) > Solid Mechanics > Applied Loads (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

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** checkbox.
- 1 In the **Model Builder** window, under **Study: BBMx** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.
- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type 0.0001.
- 5 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 6 Click  **Add**.
- 7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	range(0,0.02,1.1)	


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** checkbox.
- 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. Find the **Space variables** subsection. Select the **Remove elements on the symmetry axis** checkbox.

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)**.

Set default units for result presentation.

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/m²)** in the tree.
- 5 Click **OK**.
- 6 In the **Settings** window for **Preferred Units**, locate the **Units** section.
- 7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m ²	kPa

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

Surface


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

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**.

Transformation 1


- 1 Right-click **Surface 2** and choose **Transformation**.
- 2 In the **Settings** window for **Transformation**, locate the **Transformation** 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** checkbox.

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** checkbox.
- 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** checkbox.
- 8 In the **Pressure (dl)** toolbar, click  **Plot**.

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 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 - 1**.


Degree of Saturation at GWL = 3[m]

- 1 In the **Model Builder** window, click **Degree of Saturation at GWL = 3[m]**.
- 2 In the **Degree of Saturation at GWL = 3[m]** 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 **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

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]

- 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] 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** checkbox.

Surface 1

- 1 In the **Model Builder** window, expand the **Volumetric Plastic Strain at GWL = 3[m]** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 3 From the **Color table transformation** list, choose **Reverse**.

Surface 2

- 1 Right-click **Results > Volumetric Plastic Strain at GWL = 3[m] > 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 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.


Transformation 1

- 1 Right-click **Surface 2** and choose **Transformation**.
- 2 In the **Settings** window for **Transformation**, locate the **Transformation** section.
- 3 In the **x** text field, type 12.

Volumetric Plastic Strain at GWL = 3[m]

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

Table Annotation 1

- 1 In the **Volumetric Plastic Strain at GWL = 3[m]** 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** checkbox.

Arrow Line 1

- 1 Right-click **Volumetric Plastic Strain at GWL = 3[m]** 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.fax,solid.fay - Force per deformed area (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**.

Transformation 1

- 1 In the **Model Builder** window, right-click **Arrow Line 2** and choose **Transformation**.
- 2 In the **Settings** window for **Transformation**, locate the **Transformation** section.
- 3 In the **x** text field, type 12.

Volumetric Plastic Strain at GWL = 3[m]

- 1 In the **Model Builder** window, under **Results** click **Volumetric Plastic Strain at GWL = 3[m]**.
- 2 In the **Volumetric Plastic Strain at GWL = 3[m]** toolbar, click  **Plot**.

Volumetric Plastic Strain at GWL = 5[m]

- 1 Right-click **Volumetric Plastic Strain at GWL = 3[m]** and choose **Duplicate**.
- 2 Drag and drop **Volumetric Plastic Strain at GWL = 3[m] 1** below **Volumetric Plastic Strain at GWL = 3[m]**.
- 3 In the **Settings** window for **2D Plot Group**, type **Volumetric Plastic Strain at GWL = 5[m]** 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]** 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]

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

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

Void Ratio at GWL = 3[m]

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

2 In the **Settings** window for **2D Plot Group**, type **Void Ratio 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 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Surface 2

1 In the **Model Builder** window, expand the **Void Ratio at GWL = 3[m]** node.

2 Right-click **Results > Void Ratio at GWL = 3[m] > 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 Locate the **Title** section. From the **Title type** list, choose **None**.

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

Transformation 1

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

2 In the **Settings** window for **Transformation**, locate the **Transformation** 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]** 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 choose **Paste Table Annotation**.


Void Ratio at GWL = 3[m]

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

Void Ratio at GWL = 5[m]


- 1 Right-click **Void Ratio at GWL = 3[m]** and choose **Duplicate**.
- 2 Drag and drop **Void Ratio at GWL = 3[m]** 1 below **Void Ratio at GWL = 3[m]**.
- 3 In the **Settings** window for **2D Plot Group**, type Void Ratio at GWL = 5[m] 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]** 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 **Results** toolbar, click  **ID Plot Group**.
- 2 Drag and drop below **Void Ratio at GWL = 5[m]**.
- 3 In the **Settings** window for **ID 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** checkbox. In the associated text field, type Settlement (mm).
- 7 Select the **y-axis label** checkbox. In the associated text field, type Footing pressure (kPa).
- 8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
- 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 $F_P(\text{para})$.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** text field, type $\text{abs}(v-\text{withsol}('sol1',v,\text{setval}(\text{para},0)))$.
- 7 From the **Unit** list, choose **mm**.
- 8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **1**.
- 9 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 10 From the **Legends** list, choose **Manual**.
- 11 In the table, enter the following settings:

Legends
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 $\text{abs}(v-\text{withsol}('sol2',v,\text{setval}(\text{para},0)))$.
- 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** checkbox.

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

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

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

3 In the **Settings** window for **ID Plot Group**, type Vertical Displacement 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 **Manual**.

7 In the **Title** text area, type Vertical Displacement due to Wetting.

- 8 Locate the **Plot Settings** section.
- 9 Select the **x-axis label** checkbox. In the associated text field, type **Position (m)**.
- 10 Select the **y-axis label** checkbox. In the associated text field, type **Vertical displacement (m)**.
- 11 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

- 1 Right-click **Vertical Displacement** 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('so11',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 `c1n1x`.
- 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
MCC 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('so12',v,setval(para,1))`.
- 6 Locate the **Legends** section. In the table, enter the following settings:

Legends
BBMx model

Vertical Displacement

In the **Model Builder** window, click **Vertical Displacement**.

Table Annotation 1


- 1 In the **Vertical Displacement** 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	1	Heave
5.5	-20	Settlement

- 5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Vertical Displacement

- 1 In the **Model Builder** window, click **Vertical Displacement**.
- 2 In the **Vertical Displacement** toolbar, click  **Plot**.