12 Potato field moisture content [ULT]

This tutorial demonstrates the applicability of PLAXIS 2D to agricultural problems. The potato field tutorial involves a loam layer on top of a sandy base. The water level in the ditches remains unchanged. The precipitation and evaporation may vary on a daily basis due to weather conditions. The calculation aims to predict the variation of the water content in the loam layer in time as a result of time-dependent boundary conditions.

Objectives

- Defining precipitation

Geometry

Due to the symmetry of the problem, it is sufficient to simulate a strip with a width of 15.0 m, as indicated in Figure 162 (on page 186). The thickness of the loam layer is 2.0 m and the sand layer is 3.0 m deep.

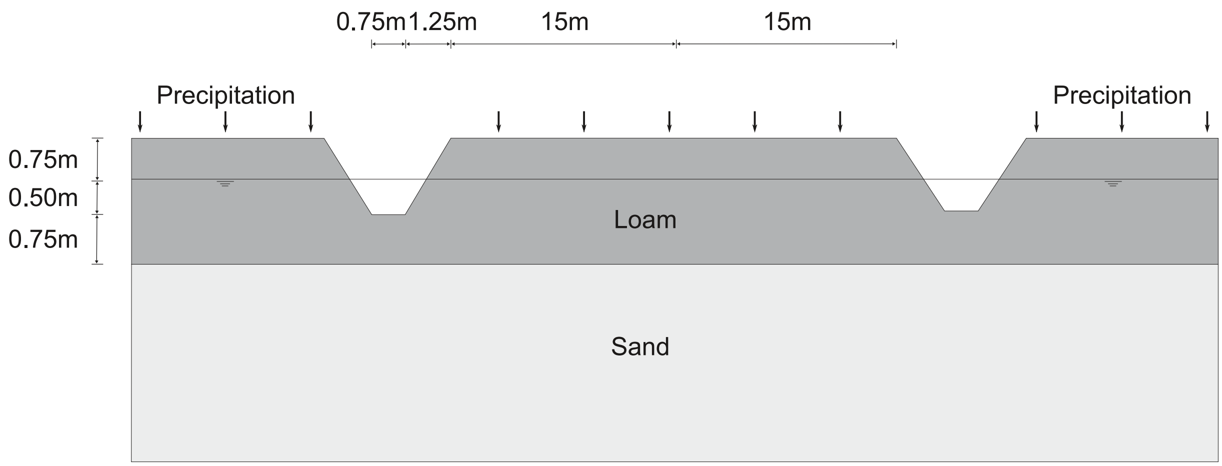

Figure 162: Potato field geometry

12.1 Create new project

To create a new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.

2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet keep the default options for **Model** (**Plane strain**), and **Elements** (**15-Node**).
4. Set the model dimensions to: x_{min} = 0 m, x_{max} = 15 m, y_{min} = 0 m and y_{max} = 5 m.
5. Keep the default values for units, constants and the general parameters and press **OK**.
   The **Project properties** window closes.

12.2 Define the soil stratigraphy

Due to the geometry of the model, the options for snapping should be changed.

1. Click the **Snapping options** button in the bottom toolbar and snapping window appears as shown in Figure 163 (on page 187).

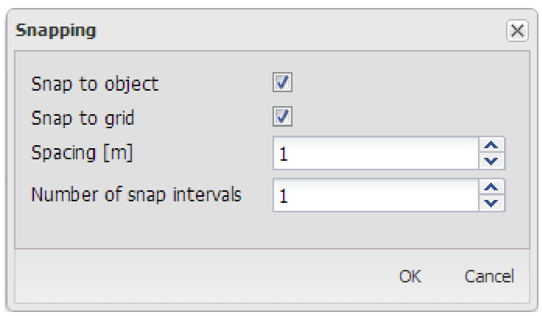

*Figure 163: Modification of the **Number of snap intervals***

2. In the appearing window set the **Number of snap intervals** to 100.
3. Click **OK** to close the **Snapping** window.

   To define the soil stratigraphy:

4. Click the **Create borehole** button and create two boreholes located at x = 0.75 and x = 2 respectively.
5. In the **Modify soil layers** window add two soil layers.
6. In the first borehole set **Top** = 3.75 and **Bottom** = 3 for the uppermost soil layer. Set **Bottom** = 0 for the lowest soil layer.
7. In the second borehole set **Top** = 5 and **Bottom** = 3 for the uppermost soil layer. Set **Bottom** = 0 for the lowest soil layer.
8. For both boreholes the **Head** is located at y = 4.25.

   Figure 164 (on page 188) shows the soil stratigraphy defined in the **Modify soil layers** window.

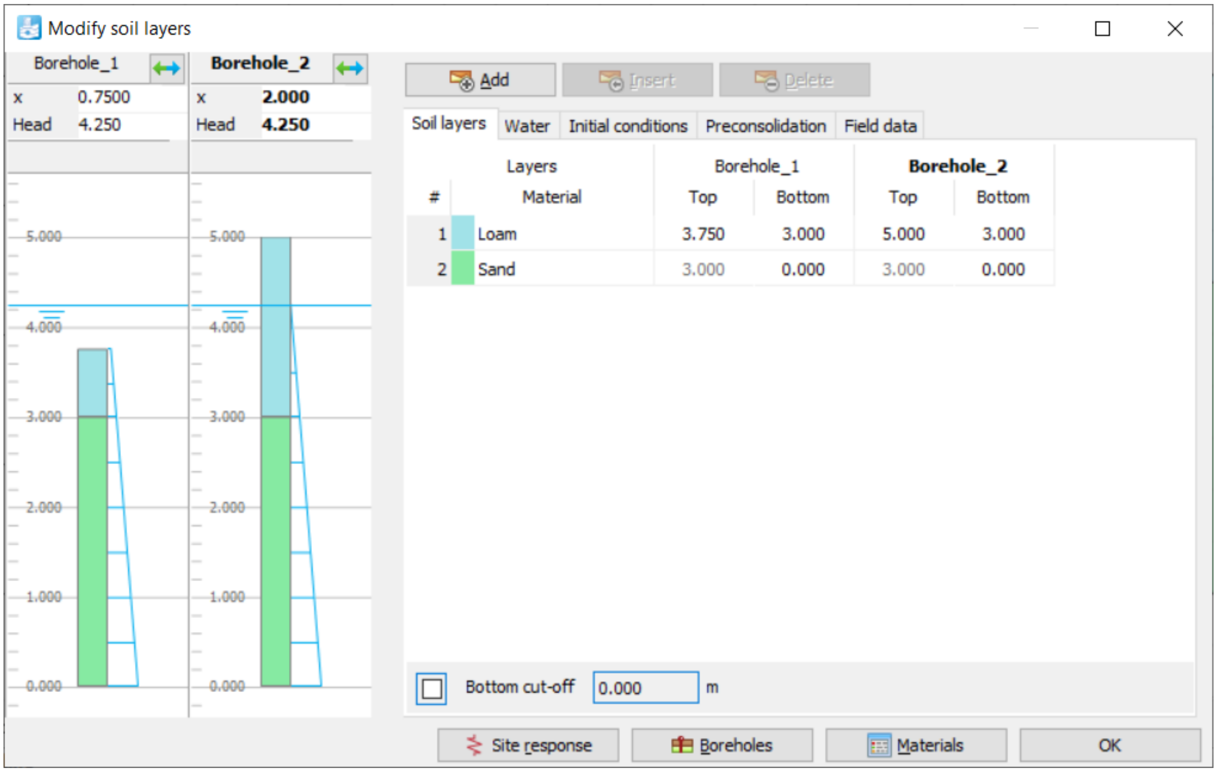

*Figure 164: Soil stratigraphy in the **Modify soil layers window***

12.3 Create and assign material data sets

Two material data sets need to be created for the soil layers.

The layers have the following properties:

Table 27: Material properties of the material

Parameter	Name	Loam	Sand	Unit
General				
Soil model	Model	Linear elastic	Linear elastic	-
Drainage type	Type	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	19	20	kN/m^3
Saturated unit weight	γ_{sat}	19	20	kN/m^3

Mechanical				
Stiffness	E'_{ref}	$1 \cdot 10^3$	$10 \cdot 10^3$	kN/m^2
Poisson's ratio	ν	0.3	0.3	-

Groundwater				
Classification type	Type	Staring	Staring	-
SWCC fitting method	-	Van Genuchten	Van Genuchten	-
Subsoil/Topsoil	-	Topsoil	Subsoil	-
Soil class	-	Clayey loam (B9)	Loamy sand (O2)	-
Flow parameters - Use defaults	-	From data set	From data set	-
Permeability in horizontal direction	k_x	0.01538	0.1270	m/day
Permeability in vertical direction	k_y	0.01538	0.1270	m/day

To create the material sets, follow these steps:

1. Create the material data sets according to Table 27 (on page 188).
2. Assign the material data set to the corresponding clusters in the model.

12.4 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Multi-select the line segments composing the upper boundary of the model as shown in Figure 165 (on page 189).

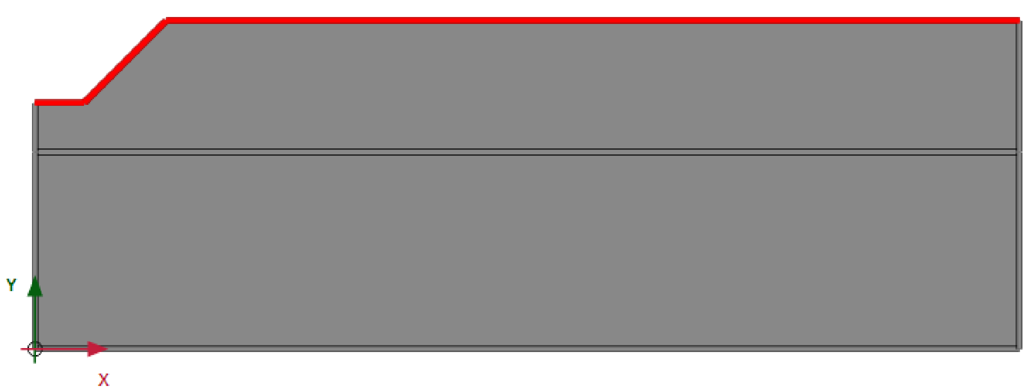

Figure 165: The upper boundary of the model

3. In the **Selection explorer** set the **Coarseness factor** parameter to 0.5.
4. Click the **Generate mesh** button to generate the mesh. Use the default option for the **Element distribution** parameter (**Medium**).
5. Click the **View mesh** button to view the mesh which is shown in Figure 166 (on page 190).

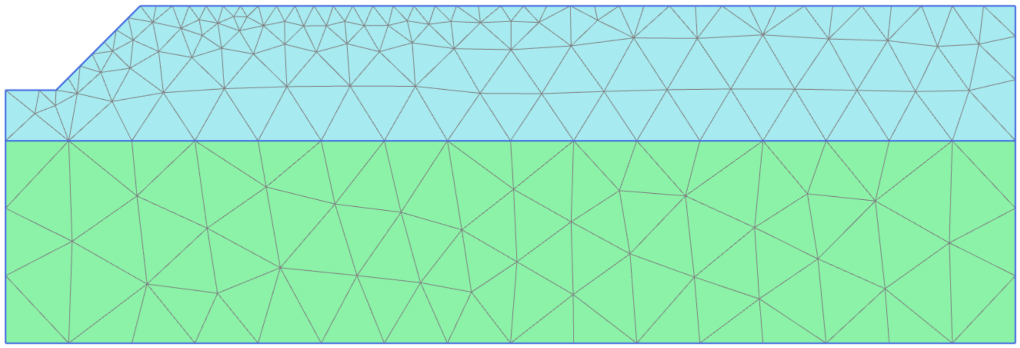

Figure 166: The generated mesh

6. Click the **Close** tab to close the Output program.

12.5 Define and perform the calculation

The calculation process consists of two phases. In the initial phase, the groundwater flow in steady state is calculated. In Phase 1, the transient groundwater flow is calculated.

12.5.1 Initial phase

1. Proceed to the **Staged construction mode** . In this project only groundwater flow analysis will be performed.
2. In the **Phases** window, select the **General** subtree, set the **Calculation type** as **Flow only** option.
3. The default values of the remaining parameters are valid for this phase. Click **OK** to close the **Phases** window.
4. **Right-click** the bottom boundary of the model and select the **Activate** option in the appearing menu.
5. In the **Selection explorer** in the **Behaviour** drop-down menu select the **Head** option and set h_{ref} to 3.0 as shown in Figure 167 (on page 191).

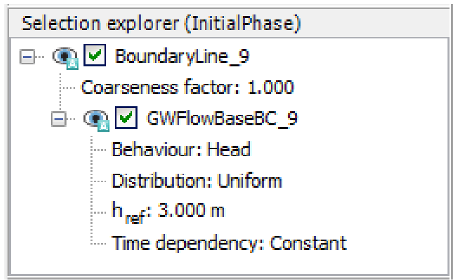

Figure 167: Initial phase with ground water flow base

6. In the **Model explorer** expand the **Model conditions** subtree.
7. Expand the **GroundwaterFlow** subtree. Set **BoundaryXMin** and **BoundaryXMax** to **Closed**.
8. Expand the **Water** subtree. The borehole water level is assigned to **GlobalWaterLevel**.

Note: Note that the conditions explicitly assigned to groundwater flow boundaries are taken into account. In this tutorial the specified **Head** will be considered for the bottom boundary of the model, NOT the **Closed** condition specified in the **GroundwaterFlow** subtree under the **Model conditions**.

12.5.2 Transient phase

In the transient phase the time-dependent variation of precipitation is defined.

A discharge function with the following precipitation data will be defined as shown in Table 28 (on page 191).

Table 28: Precipitation data

ID	Time [days]	Δ Discharge [m^3/day/m]
1	0	0
2	1	0.01
3	2	0.03
4	3	0
5	4	-0.02
6	5	0
7	6	0.01
8	7	0.01
9	8	0
10	9	-0.02

ID	Time [days]	∆ Discharge [m^3/day/m]
11	10	-0.02
12	11	-0.02
13	12	-0.01
14	13	-0.01
15	14	0
16	15	0

1. Click the **Add phase** button to create a new phase.
2. In **General** subtree of the **Phases** window select the **Transient groundwater flow** as **Pore pressure calculation type**.
3. Set the **Time interval** to 15 days.
4. In the **Numerical control parameters** subtree set the **Max number of steps stored** to 250. The default values of the remaining parameters will be used.
5. Click **OK** to close the **Phases** window.
6. To define the precipitation data a discharge function should be defined. In the **Model explorer** expand the **Attributes library** subtree.
7. Right-click on **Flow functions** and select the **Edit** option in the appearing menu. The **Flow functions** window pops up.
8. In the **Discharge functions** tabsheet add a new function.
9. Specify a name for the function and select the **Table** option in the **Signal** drop-down menu.
10. Click the **Add row** button to introduce a new row in the table. Complete the data using the values given in Table 28 (on page 191).

    Figure 168 (on page 193) shows the defined function for precipitation.

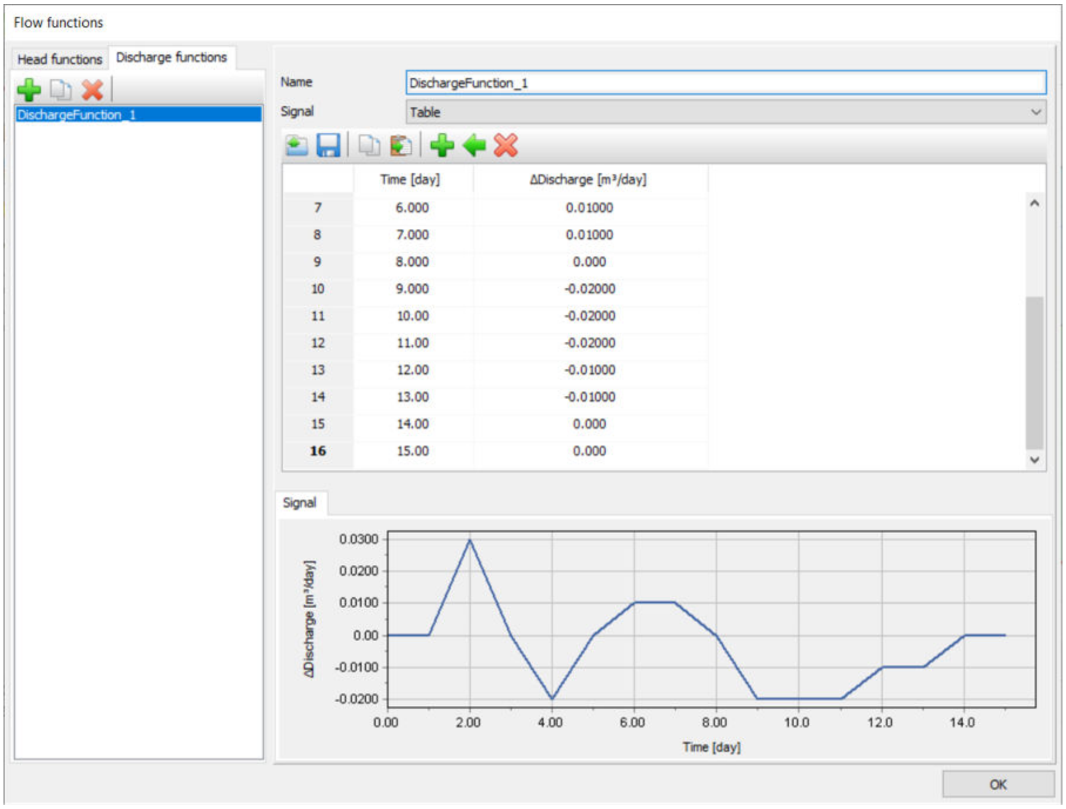

*Figure 168: The **Flow function** window displaying the precipitation data and plot*

11. Close the windows by clicking **OK**.
12. In the **Model explorer** under **Model conditions** expand the **Precipitation** subtree and activate it. The default values for discharge (q) and condition parameters (ψ_{min} = -1.0 m and ψ_{max} = 0.1m) are valid.
13. For the precipitation select the **Time dependent** option in the corresponding drop-down menu and assign the defined function.
14. In the **Model explorer** set DischargeFunction_1 under **Discharge function** as shown in Figure 169 (on page 194).

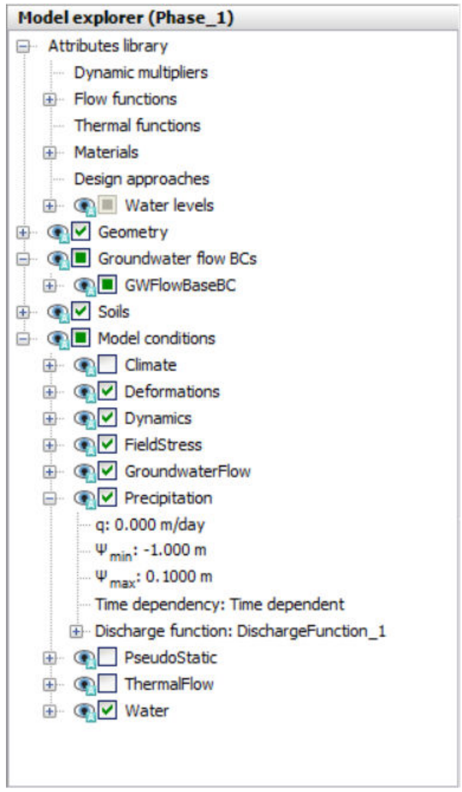

Figure 169: Precipitation in the Model explorer

Note: Negative values of precipitation indicate evaporation.

12.5.3 Execute the calculation

1. Click the **Calculate** button, ignore the feedback and continue to calculate the project. .
2. Save the project after the calculation has finished.

12.6 Results

The calculation was focused on the time-dependent saturation of the potato field.

To view the results:

1. Click the menu **Stresses** > **Groundwater flow** > **Saturation**.
2. Double click the legend.

The **Legend settings** window pops up. Define the settings as shown in Figure 170 (on page 195).

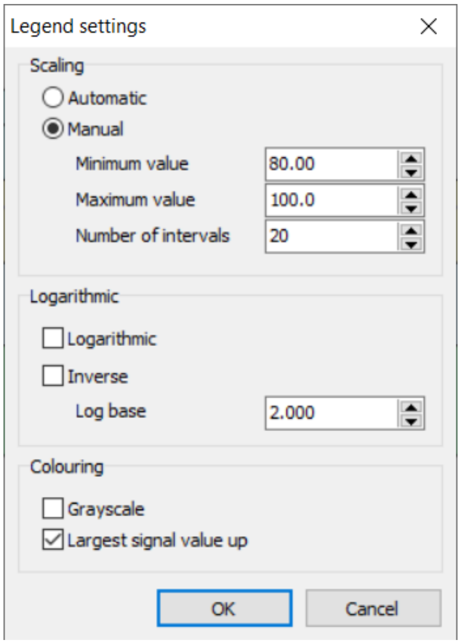

Figure 170: Value for settings

3. Figure 171 (on page 195) shows the spatial distribution of the saturation for the last time step.

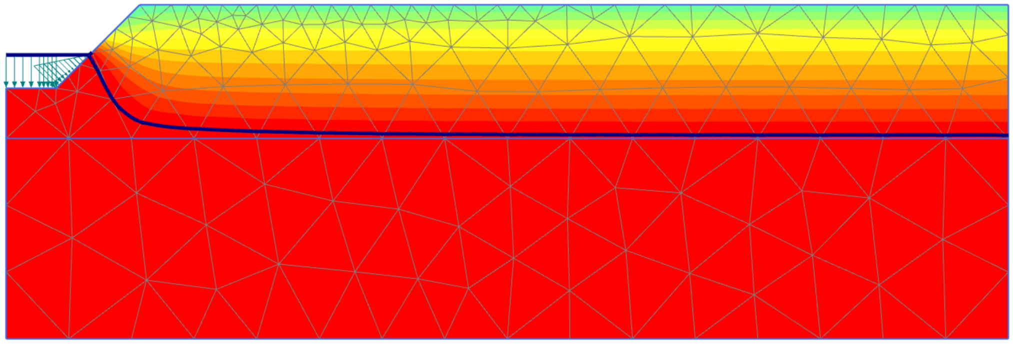

Figure 171: Saturation field at day 15

4. Create an animation of the transient phase for a better visualisation of the results.
5. It is also interesting to create a vertical cross section at x = 4 m and draw cross section curves for pore pressure and saturation.