# PLAXIS

PLAXIS 2D 2024.1

Tutorial Manual 2D

Bentley®
Advancing Infrastructure

Last Updated: December 29, 2023

# Table of Contents

# Introduction

PLAXIS 2D is a finite element package that has been developed specifically for the analysis of deformation, stability and flow in geotechnical engineering projects. The simple graphical input procedures enable a quick generation of complex finite element models, and the enhanced output facilities provide a detailed presentation of computational results. The calculation itself is fully automated and based on robust numerical procedures. This concept enables new users to work with the package after only a few hours of training.

Though the various tutorials deal with a wide range of interesting practical applications, this Tutorial Manual is intended to help new users become familiar with PLAXIS 2D. The tutorials and the respective material data sets should therefore not be used as a basis for practical projects.

Users are expected to have a basic understanding of soil mechanics and should be able to work in a Windows environment. It is strongly recommended that the tutorials are followed in the order that they appear in the manual. Please note that minor differences in results maybe found, depending on hardware and software configuration.

The Tutorial Manual does not provide theoretical background information on the finite element method, nor does it explain the details of the various soil models available in the program. The latter can be found in the Material Models Manual, as included in the full manual, and theoretical background is given in the Scientific Manual. For detailed information on the available program features, the user is referred to the Reference Manual . In addition to the full set of manuals, short courses are organised on a regular basis at several places in the world to provide hands-on experience and background information on the use of the program.

**Tutorials available for different licencing levels:**

Given PLAXIS 2D features and soil models are provided for separated licencing services, the present tutorial manuals are available with previous installation of a specific licence level.

For more information about licencing levels please visit: General Information Manual, Reference Manual and Material Models Manual).

As a summary, the tutorials available for each licence level can be identified with the following conventions:

- Tutorials with no identification - generally available for PLAXIS 2D licence.
- [ADV] - tutorials for users with PLAXIS 2D Advanced licence.
- [ULT] - tutorials for users with PLAXIS 2D Ultimate licence.
- [GSE] - tutorials for users with Geotechnical SELECT subscription (previous Basic, Advanced or Ultimate licence level required).

# 1 Settlement of a circular footing on sand

In this chapter a first application is considered, namely the settlement of a circular foundation footing on sand. This is the first step in becoming familiar with the practical use of PLAXIS 2D. The general procedures for the creation of a geometry model, the generation of a finite element mesh, the execution of a finite element calculation and the evaluation of the output results are described here in detail. The information provided in this chapter will be utilised in the later tutorials. Therefore, it is important to complete this first tutorial before attempting any further tutorial examples.

**Objectives:**

- Starting a new project
- Creating an axisymmetric model
- Creating soil stratigraphy using the *Borehole* feature
- Creating and assigning of material data sets for soil (*Mohr-Coulomb model*)
- Defining prescribed displacements
- Creation of footing using the *Plate* feature
- Creating and assigning material data sets for plates
- Creating loads
- Generating the mesh
- Generating initial stresses using the K0 procedure
- Defining a *Plastic* calculation
- Activating and modifying the values of loads in calculation phases
- Viewing the calculation results
- Selecting points for curves
- Creating a 'Load - displacement' curve

## 1.1 Geometry

A circular footing with a radius of 1.0 m is placed on a sand layer of 4.0m thickness as shown in Figure 1 (on page 10). Under the sand layer there is a stiff rock layer that extends to a large depth. The purpose of the exercise is to find the displacements and stresses in the soil caused by the load applied to the footing. Calculations are performed for both rigid and flexible footings. The geometry of the finite element model for these two situations is similar. The rock layer is not included in the model; instead, an appropriate boundary condition is applied at the bottom of the sand layer. To enable any possible mechanism in the sand and to avoid any influence of the outer boundary, the model as shown in figure is extended in horizontal direction to a total radius of 5.0 m.

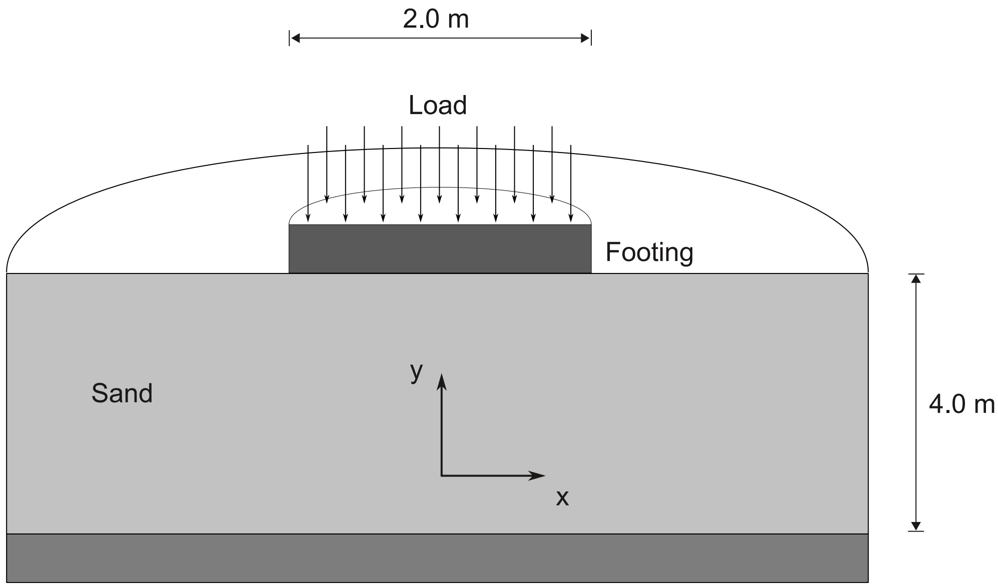



*Figure 1: Geometry of a circular footing on a sand layer*

## 1.2 Case A: Rigid footing

In the first calculation, the footing is considered to be very stiff and rough. In this calculation the settlement of the footing is simulated by means of a uniform indentation at the top of the sand layer instead of modelling the footing itself. This approach leads to a very simple model and is therefore used as a first exercise, but it also has some disadvantages. For example, it does not give any information about the structural forces in the footing.

The second part of this tutorial deals with an external load on a flexible footing, which is a more advanced modelling approach.

### 1.2.1 Create a new project

**1.** Start PLAXIS 2D by double clicking the icon of the Input program .

The **Quick start** dialog box appears in which you can create a new project or select an existing one.

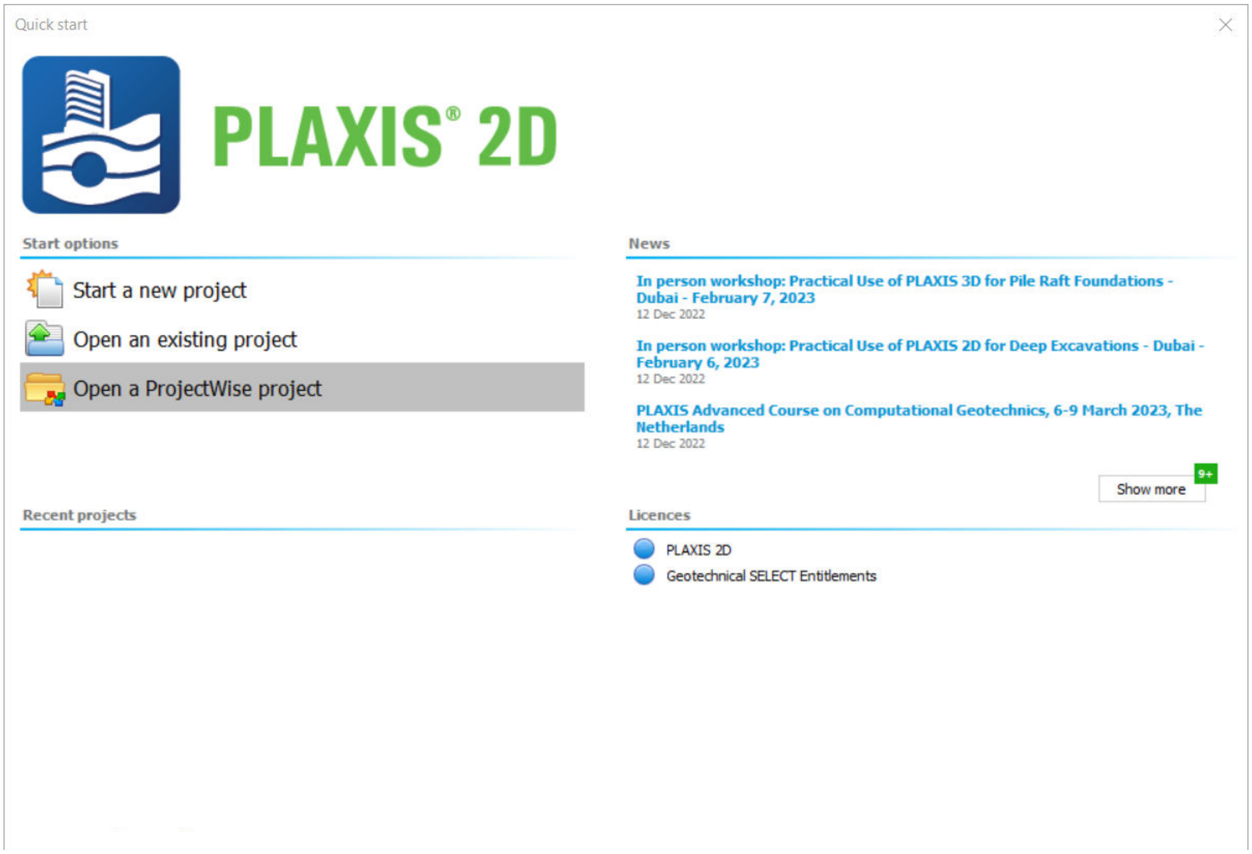



*Figure 2: Quick start - PLAXIS 2D*

2. Click **Start a new project**.

   The **Project properties** window appears with three tabsheets: **Project**, **Model** and **Cloud services** .

**Note:** For the different licencing tiers the **Project properties** window will vary from three tabsheets to four tabsheets with the addition of **Constants** alongwith **Project**, **Model** and **Cloud services**.

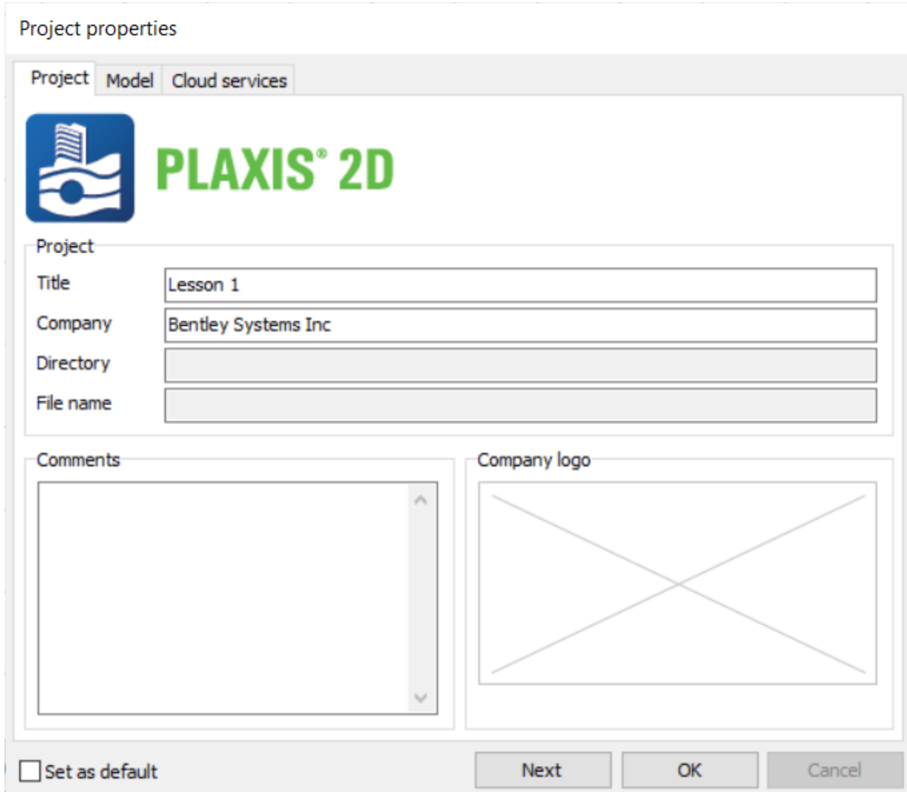



*Figure 3: Project properties window - PLAXIS 2D*

**Note:**

The first step in every analysis is to set the basic parameters of the finite element model. This is done in the **Project properties** window. These settings include the description of the problem, the type of model, the basic type of elements, the basic units and the size of the drawing area.

To enter the appropriate settings for the footing calculation follow the steps below.

3. In the **Project** tabsheet, enter `Lesson 1` in the **Title** box and type `Settlement of a circular footing` in the **Comments** box.
4. Click the **Next** button at the bottom or click the **Model** tab.
   The **Model** properties are shown in Figure 4 (on page 13):

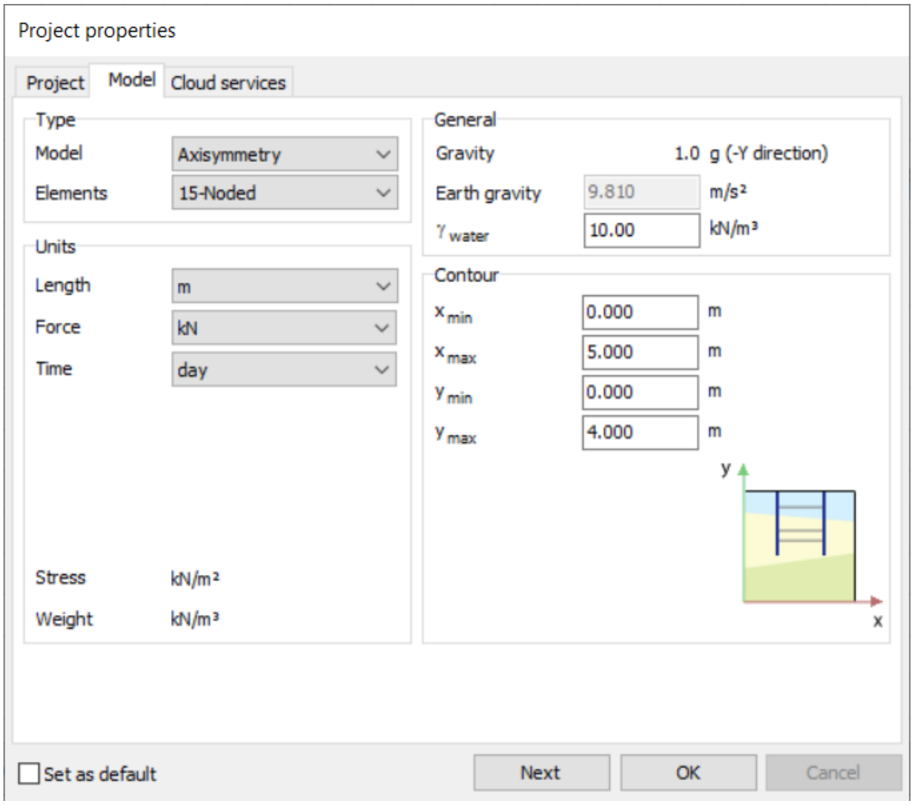



*Figure 4: Model properties tabsheet*

5. In the **Type** group the type of the model (Model) and the basic element type (Elements) are specified. Since this tutorial concerns a circular footing, select the `Axisymmetry` and the `15-Noded` options from the **Model** and the **Elements** drop-down menus respectively.
6. In the **Contour** group set the model dimensions to $x_{min}$ = 0, $x_{max}$ = 5, $y_{min}$ = 0 and $y_{max}$ = 4.
7. Click the **OK** button to confirm the settings.

The project is created with the given properties. The **Project properties** window closes and the **Soil mode** view will be shown, where the soil stratigraphy can be defined.

**Note:** The project properties can be changed later. You can access the **Project properties** window by selecting the corresponding option from the **File** menu.

## 1.2.2 Define the soil stratigraphy

In the **Soil mode** of PLAXIS 2D the soil stratigraphy can be defined.

Information on the soil layers is entered in boreholes. Boreholes are locations in the drawing area at which the information on the position of soil layers and the water table is given. If multiple boreholes are defined, PLAXIS 2D will automatically interpolate between the boreholes. The layer distribution beyond the boreholes is kept horizontal.

> **Note:** The modelling process is completed in five modes (Soil, Structures, Mesh, Flow conditions and Stage construction). More information on modes is available in the *Input Program* **Structure Mode** of the Reference Manual .

In order to construct the soil stratigraphy follow these steps:

1. Click the **Create borehole** button in the side (vertical) toolbar to start defining the soil stratigraphy.
2. Click at x = 0 in the drawing area to locate the borehole.

   The **Modify soil layers** window will appear as shown in Figure 5 (on page 14) .
3. Add a soil layer by clicking the **Add** button in the **Modify soil layers** window.
4. Set the top boundary of the soil layer at y = 4 and keep the bottom boundary at y = 0 m.
5. Set the **Head** to 2.0 m.

   By default the **Head** value (groundwater head) in the borehole column is set to 0 m.

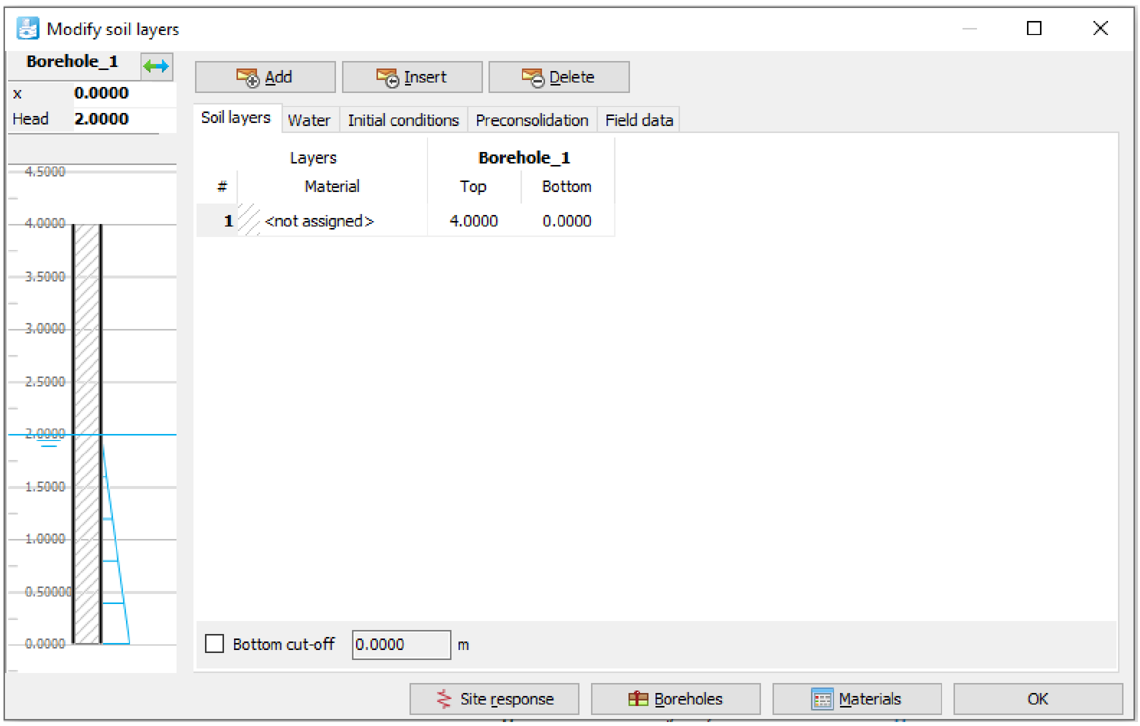



*Figure 5: **Modify soil layers** window*

Next the material data sets are defined and assigned to the soil layers, see Create and assign material data sets (on page 14).

## 1.2.3 Create and assign material data sets

In order to simulate the behaviour of the soil, a suitable soil model and appropriate material parameters must be assigned to the geometry. In PLAXIS 2D, soil properties are collected in material data sets and the various data sets are stored in a material database. From the database, a data set can be assigned to one or more soil layers. For structures (like walls, plates, anchors, geogrids, etc.) the system is similar, but different types of structures have different parameters and therefore different types of material data sets. PLAXIS 2D distinguishes between material data sets for **Soil and interfaces**, **Discontinuities** ,**Plates**, **Geogrids**, **Embedded beams** , **Cables** and **Anchors**.

The sand layer that is used in this tutorial has the following properties as shown in Table 1 (on page 15) :

**Table 1: Material properties of the sand layer**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | Model | Mohr-Coulomb | - |
| Drainage type | Type | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 17 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | $kN/m^3$ |
| **Mechanical** | | | |
| Young's modulus | $E'_{ref}$ | $13 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 0.3 | - |
| Cohesion | $c'_{ref}$ | 1 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 30 | ° |
| Dilatancy angle | $\psi$ | 0 | ° |

To create a material set for the sand layer, follow these steps:

1. Open the **Material sets** window by clicking the **Materials** button  in the **Modify soil layers** window or in the side toolbar.
The **Material sets** window pops up as shown in Figure 6 (on page 16).

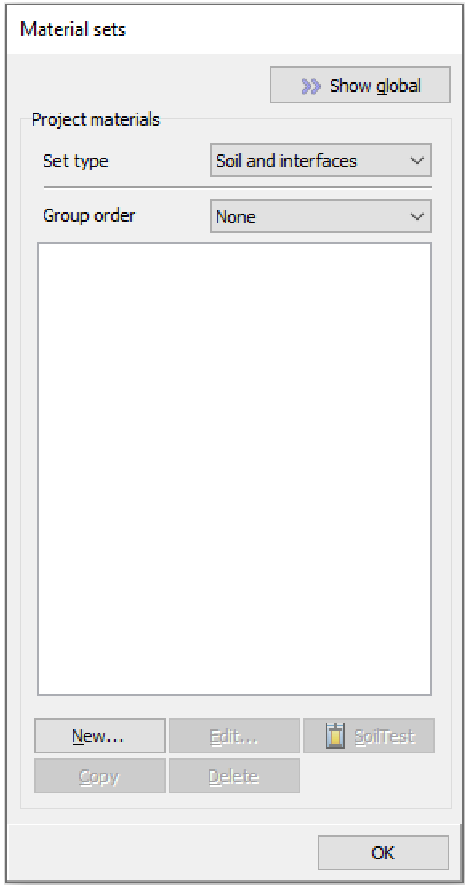



*Figure 6: Material sets window*

2. Click the **New** button at the lower side of the **Material sets** window.
   A new window will appear with these tabsheets: **General**, **Mechanical**, **Groundwater**, **Thermal**, **Interfaces** and **Initial**.
3. In the **Material set** box of the **General** tabsheet, write `Sand` in the **Identification** box.
   The default material model (Mohr-Coulomb) and drainage type (Drained) are valid for this example.
4. Enter the proper values in the **General properties** box (Figure 7 (on page 17)) according to the material properties listed in Table 1 (on page 15). Keep parameters that are not mentioned in the table at their default values.

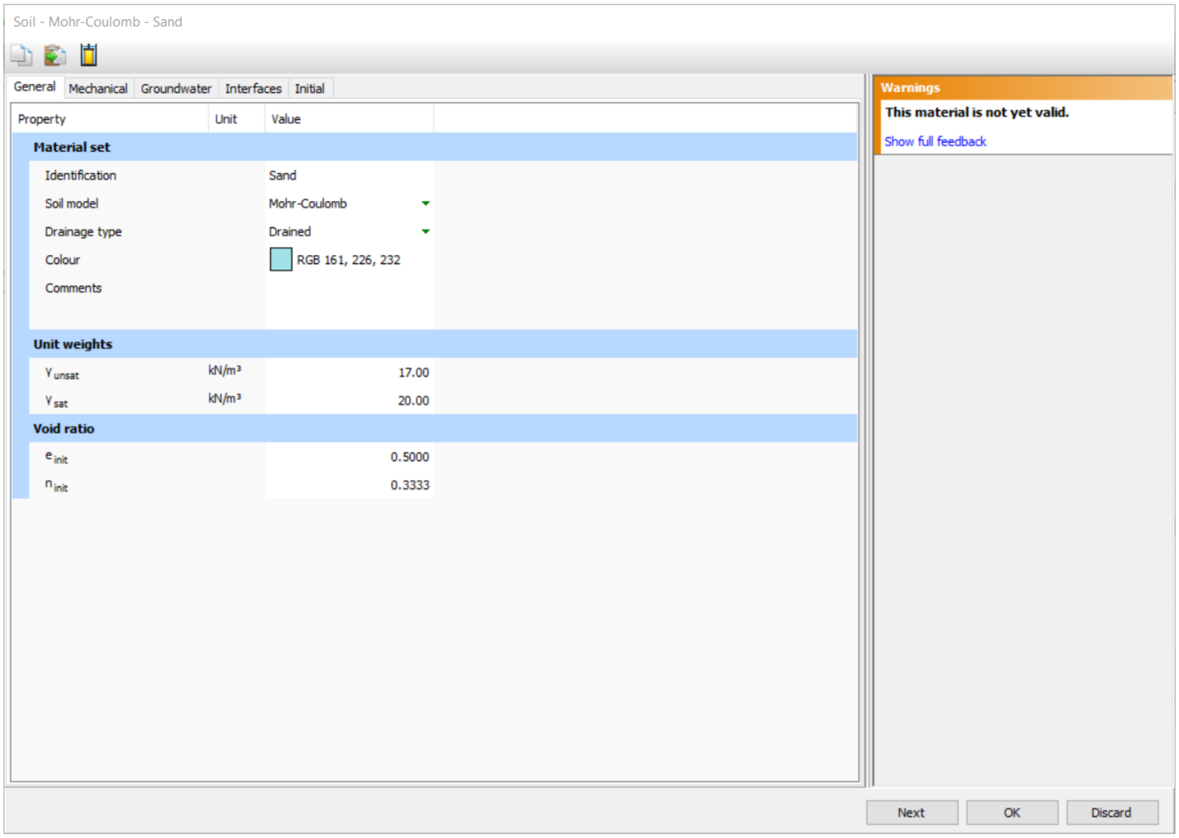



*Figure 7: The **General** tabsheet of the **Soil** window*

**Note:**

1. As displayed in Figure 7 (on page 17) a **Feedback side panel** is included in the **Material** window. This panel prevents the definition of an invalid material data set. To display the list of detailed messages please select *Show full feedback*. Three types of messages are possible:
   - **Errors:** the parameter value or combination of parameter values must be changed, otherwise the material set could be invalid and calculation of the project will be blocked.
   - **Warnings:** the parameter value seems to deviate from a recommended parameter value or parameter range. Generally the material set will not be considered invalid and calculating the project will not be blocked. The chosen parameter could however cause unexpected results.
   - **Hints:** the entered parameter can be defined under certain circumstances or options.
2. The **Feedback side panel** is displayed at the moment of defining materials and structures. For the sake of simplicity, this panel will only be shown in some tutorial examples..

5. Click the **Next** button or click the **Mechanical** tab to proceed with the input of model parameters.
   The parameters appearing on the **Mechanical** tabsheet depend on the selected material model (in this case the Mohr-Coulomb model).

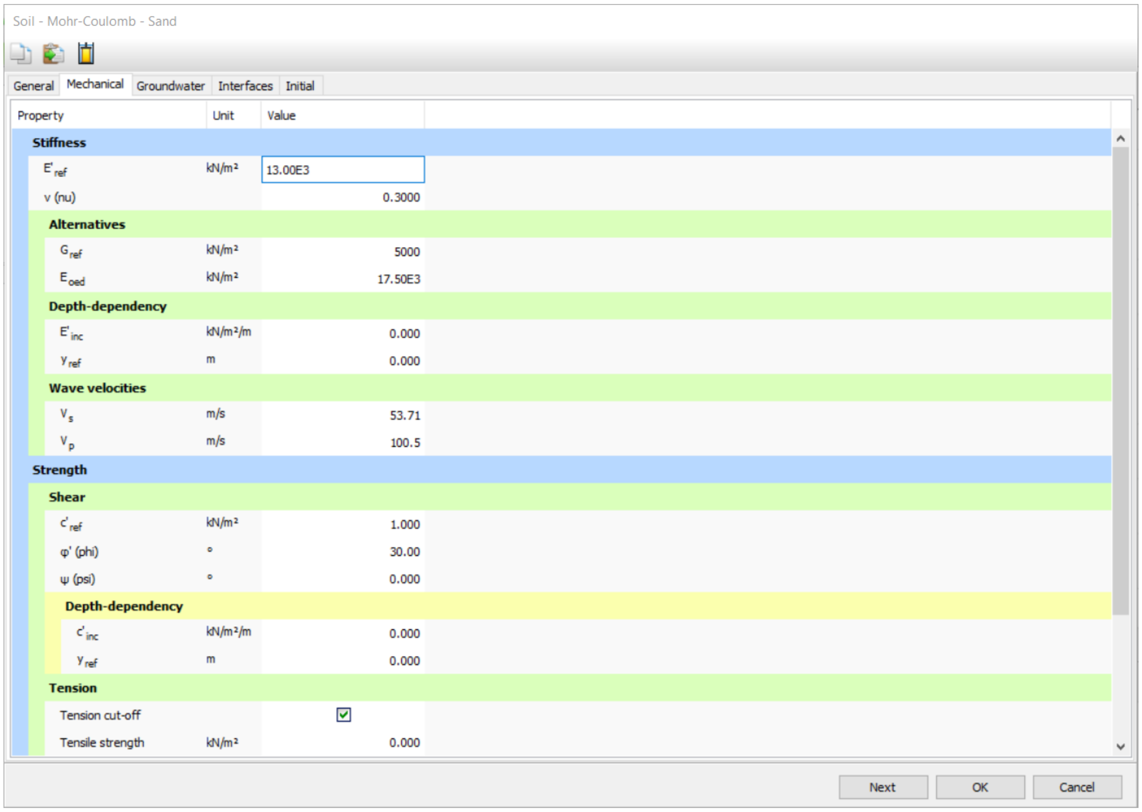



*Figure 8: **Mechanical** tabsheet of the **Soil** window - Soil and interfaces set type*

6. Enter the model parameters of Table 1 (on page 15) in the corresponding edit boxes of the **Mechanical** tabsheet (Figure 8 (on page 18)) and keep the other parameters as their default values. A detailed description of different soil models and their corresponding parameters can be found in the Material Models Manual.

> **Note:** To understand why a particular soil model has been chosen, see Appendix B of the Material Models Manual.

7. The soil material is drained, the geometry model does not include interfaces and the default thermal and initial conditions are valid for this case, therefore the remaining tabsheets can be skipped. Click **OK** to confirm the input of the current material data set.
   Now the created data set will appear in the tree view of the **Material sets** window.
8. Drag the set **Sand** from the **Material sets** window (select it and hold down the left mouse button while moving) to the graph of the soil column on the left hand side of the **Modify soil layers** window and drop it there (release the left mouse button).
9. Click **OK** in the **Material sets** window to close the database.
10. Click **OK** to close the **Modify soil layers** window.

> **Note:**
>
> - Existing data sets may be changed by opening the **Material sets** window, selecting the data set to be changed from the tree view and clicking the **Edit** button. As an alternative, the **Material sets** window can be opened by clicking the corresponding button in the side toolbar.
> - PLAXIS 2D distinguishes between a project database and a global database of material sets. Data sets may be exchanged from one project to another using the global database. The global database can be shown in

the **Material sets** window by clicking the *Show global* button. The data sets of all tutorials in the Tutorial Manual are stored in the global database during the installation of the program.

- The material assigned to a selected entity in the model can be changed in the **Material** drop-down menu in the **Selection explorer**. Note that all the material datasets assignable to the entity are listed in the drop-down menu. However, only the materials listed under **Project materials** are listed, and not the ones listed under **Global materials**.
- The program performs a consistency check on the material parameters and will give a warning message in the case of a detected inconsistency in the data.

## 1.2.4 Define the footing

Structural elements and loads are created in the **Structures mode** of the program. In this exercise a uniform indentation will be created to model a very stiff and rough footing.

**Note:**

Visibility of a grid in the drawing area can simplify the definition of geometry. The grid provides a matrix on the screen that can be used as reference. It may also be used for snapping to regular points during the creation of the geometry. The grid can be activated by clicking the corresponding button under the drawing area. To define the size of the grid cell and the snapping options:

Click the **Snapping options** button in the bottom toolbar. The **Snapping** window pops up where the size of the grid cells and the snapping interval can be specified. The spacing of snapping points can be further divided into smaller intervals by the **Number of snap intervals** value. Use the default values in this tutorial.

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .
2. Click the **Create prescribed displacement** button in the side toolbar.
3. Select the **Create line displacement** option in the expanded menu.
4. In the drawing area move the cursor to point (0 4) and click the left mouse button.
5. Move along the upper boundary of the soil to point (1 4) and click the left mouse button again.
6. Click the right mouse button to stop drawing.
7. In the **Selection explorer** set the x-component of the prescribed displacement (Displacement$_x$) to `Fixed`.
8. Specify a uniform prescribed displacement in the vertical direction by assigning a value of -0.05 to $u_{y,start,ref}$, signifying a downward displacement of 0.05 m as shown in Figure 9 (on page 20).

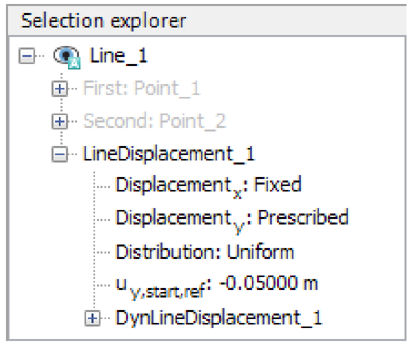



*Figure 9: Prescribed displacement in the **Selection explorer***

The geometry of the model is complete.

When the geometry model is complete, the finite element mesh can be generated. Proceed to Generate the mesh (on page 20)

## 1.2.5 Generate the mesh

PLAXIS 2D allows for a fully automatic mesh generation procedure, in which the geometry is divided into elements of the basic element type and compatible structural elements, if applicable.

The mesh generation takes full account of the position of points and lines in the model, so that the exact position of layers, loads and structures is accounted for in the finite element mesh. The generation process is based on a robust triangulation principle that searches for optimised triangles. In addition to the mesh generation itself, a transformation of input data (properties, boundary conditions, material sets, etc.) from the geometry model (points, lines and clusters) to the finite element mesh (elements, nodes and stress points) is made.

In order to generate the mesh, follow these steps:

1. Proceed to the **Mesh mode** by clicking the corresponding tab.
2. Click the **Generate mesh** button in the side toolbar.
   The **Mesh options** window pops up as shown in Figure 10 (on page 20). The **Medium** option is by default selected as element distribution.

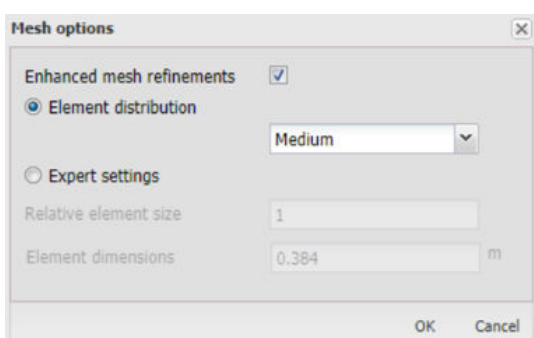



*Figure 10: The **Mesh options** window*

3. Click **OK** to start the mesh generation.
4. As the mesh is generated, click the **View mesh** button.

   A new window is opened displaying the generated mesh as shown in Figure 11 (on page 21). Note that the mesh is automatically refined under the footing.

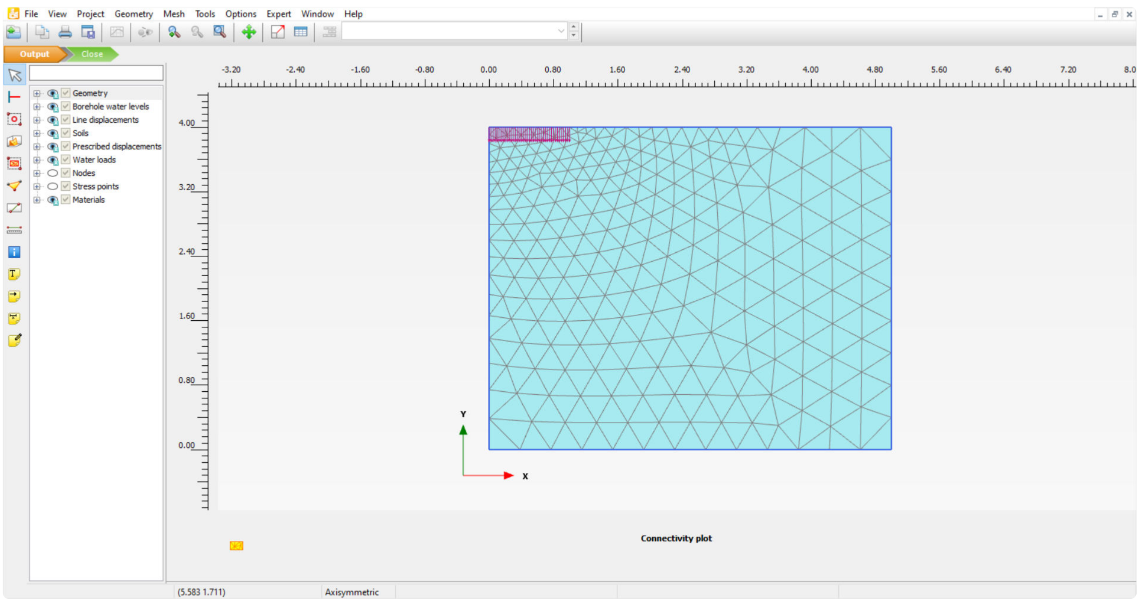



*Figure 11: The generated mesh in the **Output** window*

5. Click on the **Close** tab to close the Output program and go back to the **Mesh mode** of the **Input** program.

> **Note:**
>
> - By default, the **Element distribution** is set to `Medium`. The **Element distribution** setting can be changed in the **Mesh options** window. In addition, options are available to refine the mesh globally or locally (for more information see the Reference Manual ).
> - The finite element mesh has to be regenerated if the geometry is modified.
> - The automatically generated mesh may not be perfectly suitable for the intended calculation. Therefore it is recommended that the user inspects the mesh and makes refinements if necessary.

Once the mesh has been generated, the finite element model is complete.

After the mesh was generated, the calculation phases are defined and the calculation is done, see Initial phase (on page 22) for instructions.

## 1.2.6 Define and perform the calculation

The calculation has to be defined in phases before the actual calculation can be performed. This example needs two phases: the initial phase and one to simulate the settlement of the footing.

### Initial phase

The 'Initial phase' always involves the generation of initial conditions. In general, the initial conditions comprise the initial geometry configuration and the initial stress state, i.e. effective stresses, pore pressures and state parameters, if applicable.

1. Click the **Staged construction** tab to proceed with the definition of calculation phases. The **Flow conditions mode** may be skipped.

   When a new project has been defined, a first calculation phase named ' Initial phase', is automatically created and selected in the **Phases explorer** as shown in Figure 12 (on page 22):

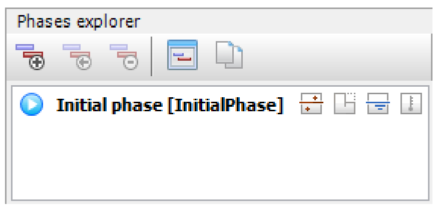
   
   

   *Figure 12: Phases explorer - Initial Phase*

   All structural elements and loads that are present in the geometry are initially automatically switched off, only the soil volumes are initially active.

2. Click the **Edit phase** button or double click the phase in the **Phases explorer**.

   In this tutorial lesson the properties of the Initial phase will be described. Below an overview is given of the options to be defined even though the default values of the parameters are used.

| | |
|---|---|
| | By default the `K0 procedure` is selected as **Calculation type** in the **General** subtree of the **Phases window**. This option will be used in this project to generate the initial stresses. |
| | The **Staged construction** option is available as **Loading type**. |
| | The `Phreatic` option is selected by default as the **Pore pressure calculation type**. |

**Note:** The **K0 procedure** should be primarily used for horizontally layered geometries with a horizontal ground surface and, if applicable, a horizontal phreatic level. See the Reference Manual for more information on the **K0 procedure**.

The other default options in the **Phases** window will be used as well in this tutorial.

The **Phases** window is displayed in Figure 13 (on page 23).

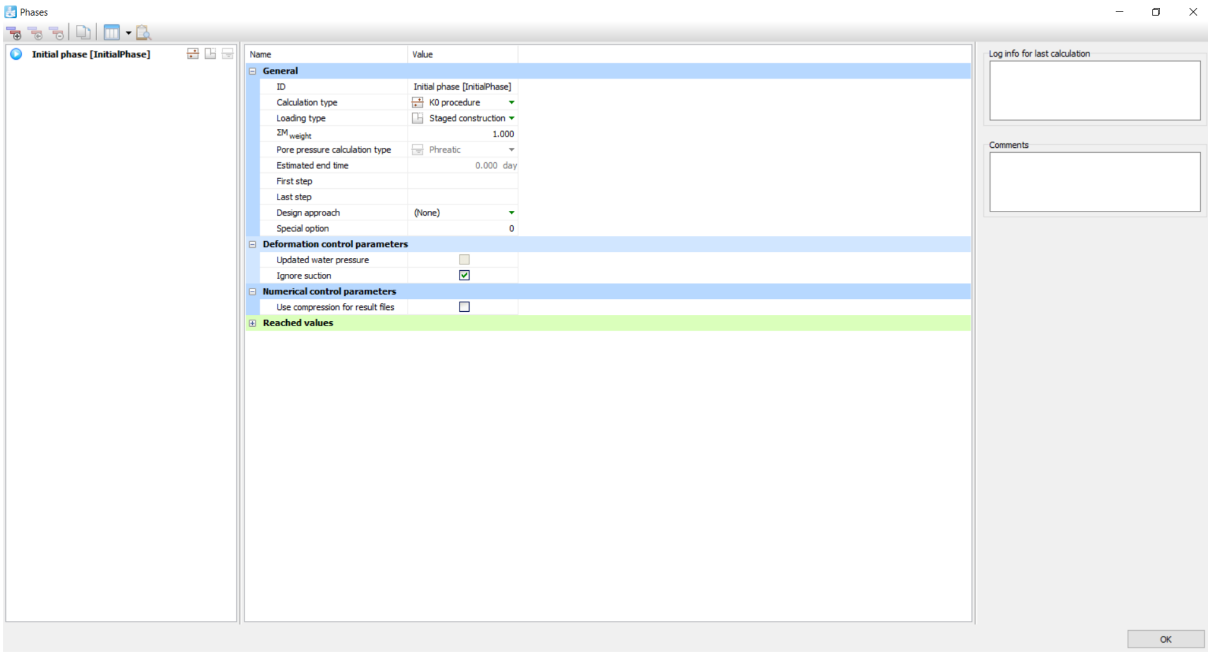



*Figure 13: Phases window - Initial phase*

3. Click **OK** to close the **Phases** window.
4. In the **Model explorer** expand the **Model conditions** subtree as shown in Figure 14 (on page 24).

   For deformation problems two types of boundary conditions exist: Prescribed displacement and prescribed forces (loads). In principle, all boundaries must have one boundary condition in each direction. That is to say, when no explicit boundary condition is given to a certain boundary (a free boundary), the natural condition applies, which is a prescribed force equal to zero and a free displacement.

   To avoid the situation where the displacements of the geometry are undetermined, some points of the geometry must have prescribed displacements. The simplest form of a prescribed displacement is a fixity (zero displacement), but non-zero prescribed displacements may also be given.
5. Expand the **Deformations** subtree.
   Note that the box is checked by default. By default, a full fixity is generated at the base of the geometry, whereas roller supports are assigned to the vertical boundaries (**BoundaryXMin** and **BoundaryXMax** are normally fixed, **BoundaryYMin** is fully fixed and **BoundaryYMax** is free).
6. Expand the **Water** subtree.

   The initial water level has been entered already in the Modify soil layers window. The water level generated according to the **Head** value assigned to boreholes in the **Modify soil layers** window (BoreholeWaterLevel_1) is automatically assigned to **GlobalWaterLevel** .

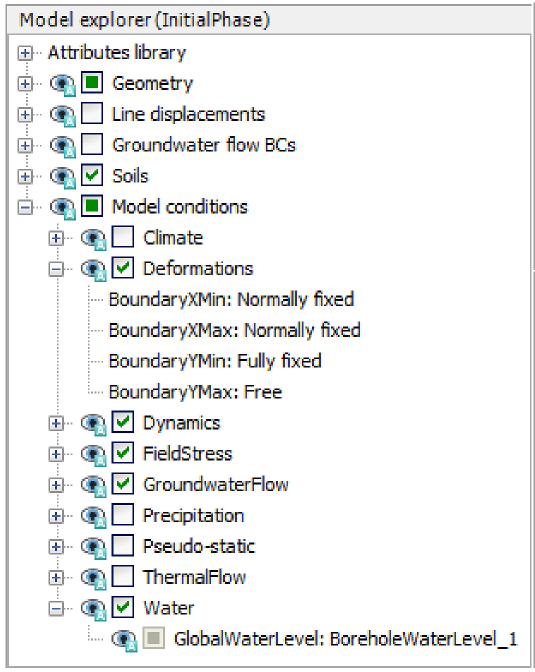



*Figure 14: Model explorer showing model conditions and Deformations*

The water level defined according to the **Head** specified for boreholes is displayed in the model explorer window. Note that only the global water level is displayed in both **Phase definition** modes. All the water levels are displayed in the model only in the **Flow conditions mode** .

The model of the project in the initial phase is shown in Figure 15 (on page 25).

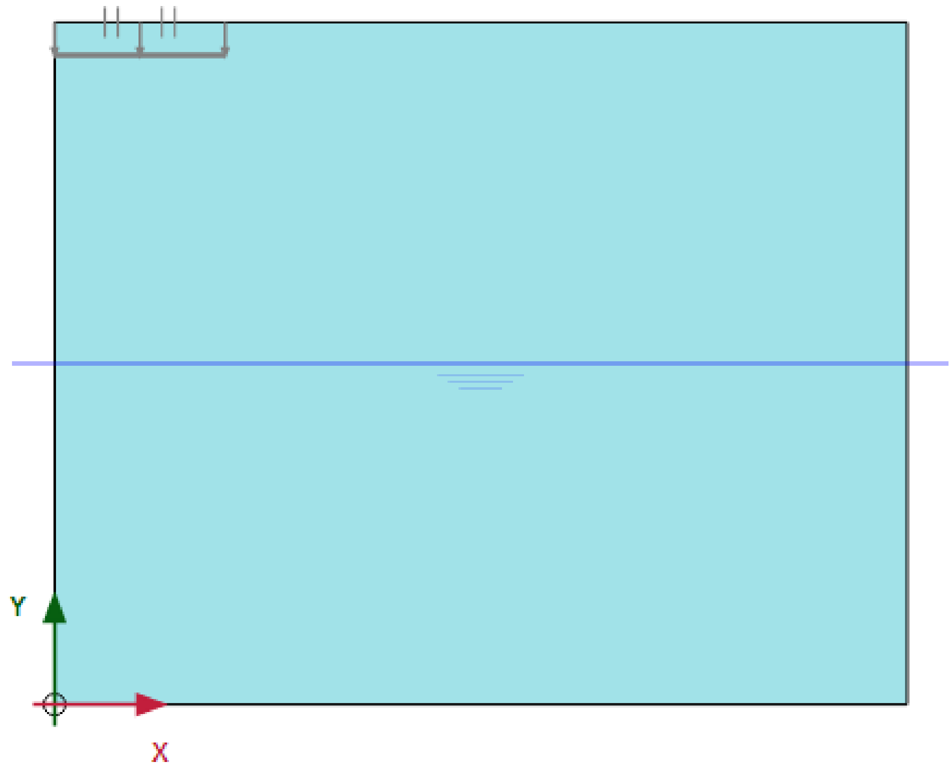



*Figure 15: Initial phase in the **Staged construction** mode*

Next, the calculation phase for the footing settlement is defined.

## Phase 1: Footing

In order to simulate the settlement of the footing in this analysis, a plastic calculation is required. PLAXIS 2D has a convenient procedure for automatic load stepping, which is called 'Load advancement'. This procedure can be used for most practical applications. Within the plastic calculation, the prescribed displacements are activated to simulate the indentation of the footing. In order to define the calculation phase follow these steps:

1. Click the **Add phase** button  in the **Phases explorer**.
   A new phase, named `Phase_1` will be added in the **Phases explorer**.
2. Double click **Phase_1** to open the **Phases** window. In the **ID** box of the **General** section, write (optionally) an appropriate name for the new phase (for example `Indentation`).

   The current phase starts from the **Initial phase**, which contains the initial stress state. The default options and values assigned are valid for this phase as shown in Figure 16 (on page 26).

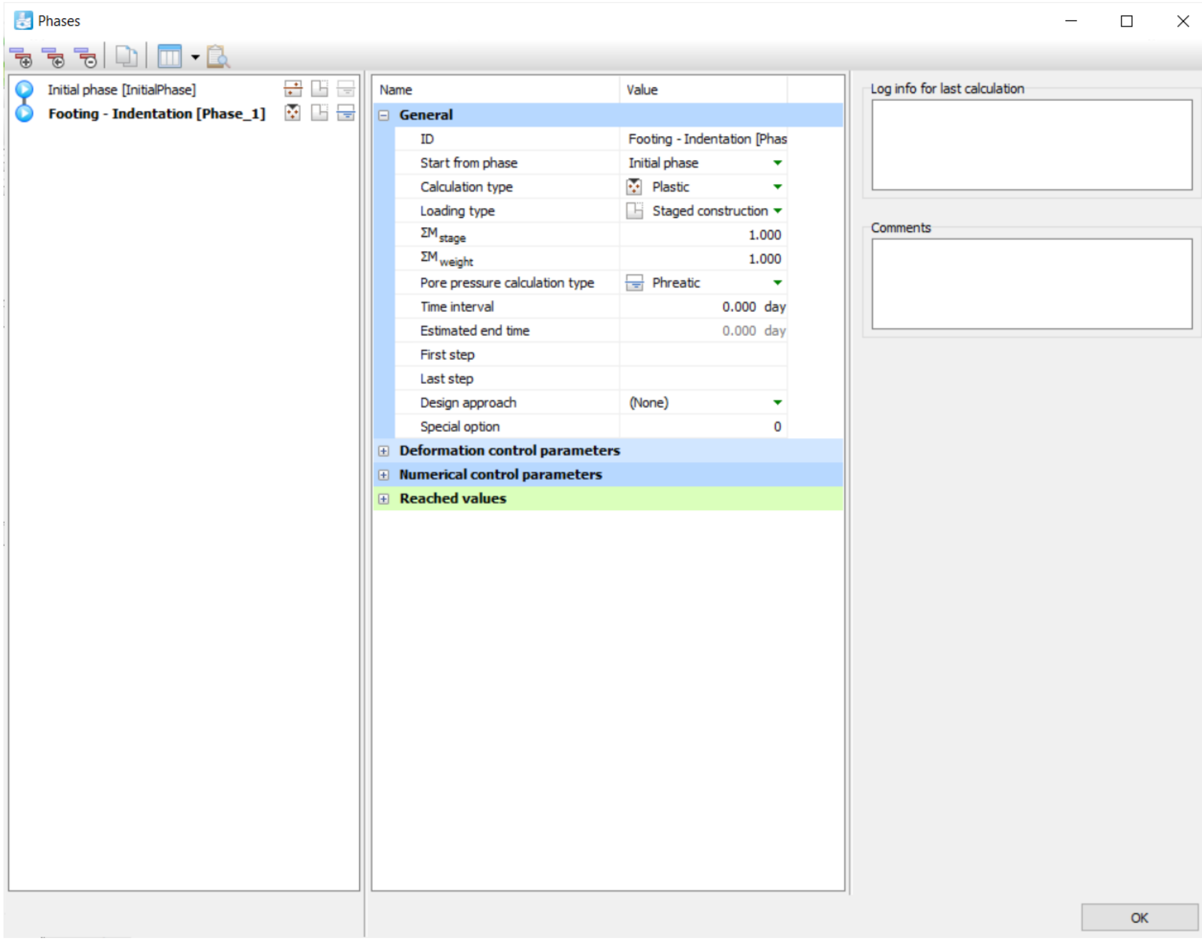



*Figure 16: The **Phases** window for the Indentation phase*

3. Click **OK** to close the **Phases** window.
4. Click the **Staged construction** tab to enter the corresponding mode.
5. Right-click the prescribed displacement in the drawing area and select the **Activate** option in the appearing menu shown in Figure 17 (on page 27).

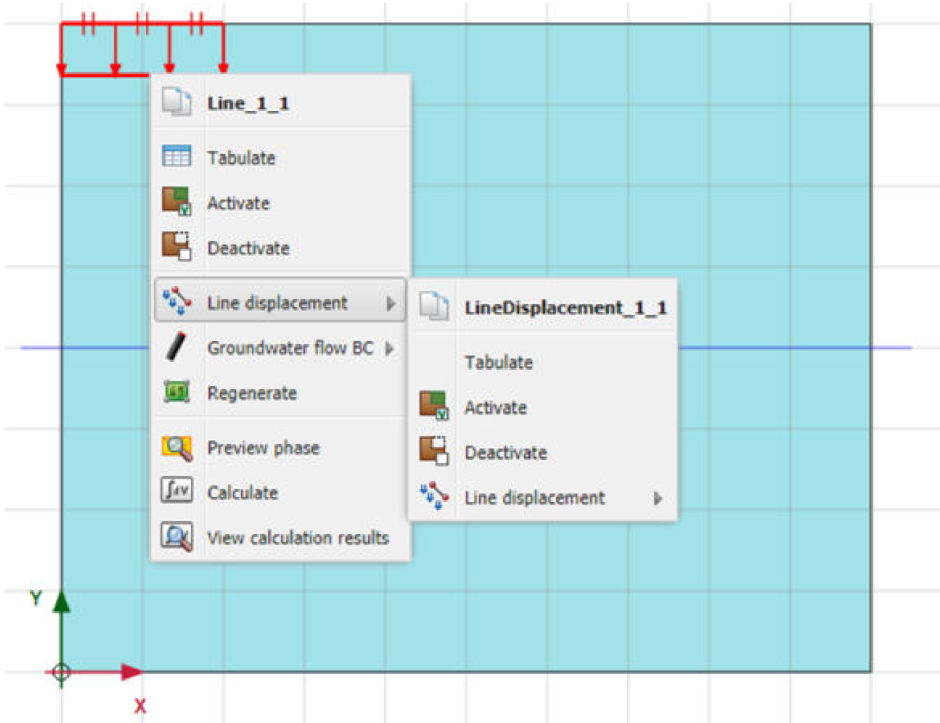



*Figure 17: Activation of the prescribed displacement in the* ***Staged construction*** *mode*

**Note:** Calculation phases may be added, inserted or deleted using the **Add**, **Insert** and **Delete** buttons in the **Phases explorer** or in the **Phases** window.

## Execute the calculation

Both calculation phases are marked for calculation, as indicated by the blue arrows. The execution order is controlled by the **Start from phase** parameter.

1. Click the **Calculate** button  to start the calculation process. Ignore the warning that no nodes and stress points have been selected for curves.

   During the execution of a calculation, a window appears which gives information about the progress of the actual calculation phase as shown in Figure 18 (on page 28) .

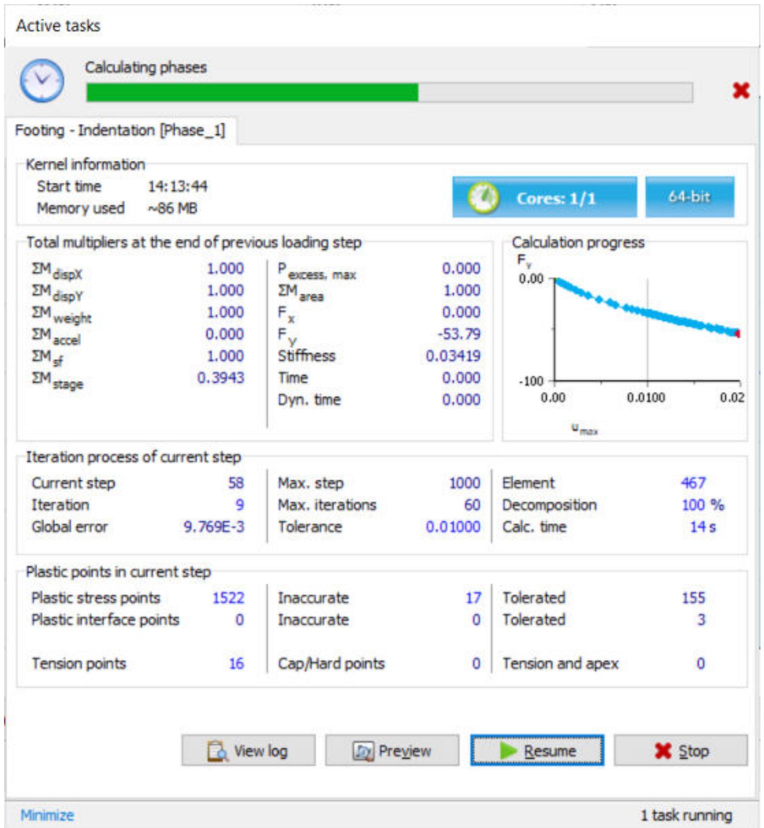



*Figure 18: Calculation progress*

The information, which is continuously updated, shows the calculation progress, the current step number, the global error in the current iteration and the number of plastic points in the current calculation step. It will take a few seconds to perform the calculation. When a calculation ends, the window is closed and focus is returned to the main window.

The phase list in the **Phases explorer** is updated. A successfully calculated phase is indicated by a check mark inside a green circle.

2. Save the project by clicking the **Save** button before viewing results.

Once the calculation has been completed, the results can be displayed in the **Output** program.

## View the calculation results

In the **Output** program, the displacement and stresses in the full two-dimensional model as well as in cross sections or structural elements can be viewed. The computational results are also available in tabular form. To check the applied load that results from the prescribed displacement of 0.05 m:

1. Open the **Phases** window.
2. From the **Reached values** subtree look for the **Force-Y** which is an important value of the current application. This value represents the total reaction force corresponding to the applied prescribed vertical displacement, which corresponds to the total force under 1.0 radian of the footing (note that the analysis is

axisymmetric). In order to obtain the total footing force, the value of **Force-Y** should be multiplied by 2π (this gives a value of about 588 kN).

The results can be evaluated in the Output program. In the **Output** window you can view the displacements and stresses in the full geometry as well as in cross sections and in structural elements, if applicable.

The computational results are also available in tabulated form. To view the results of the footing analysis, follow these steps:

3. Select the last calculation phase in the **Phases** explorer.
4. Click the **View calculation results** button in the side toolbar.

As a result, the **Output** program is started, showing the deformed mesh at the end of the selected calculation phase as shown in Figure 19 (on page 29):

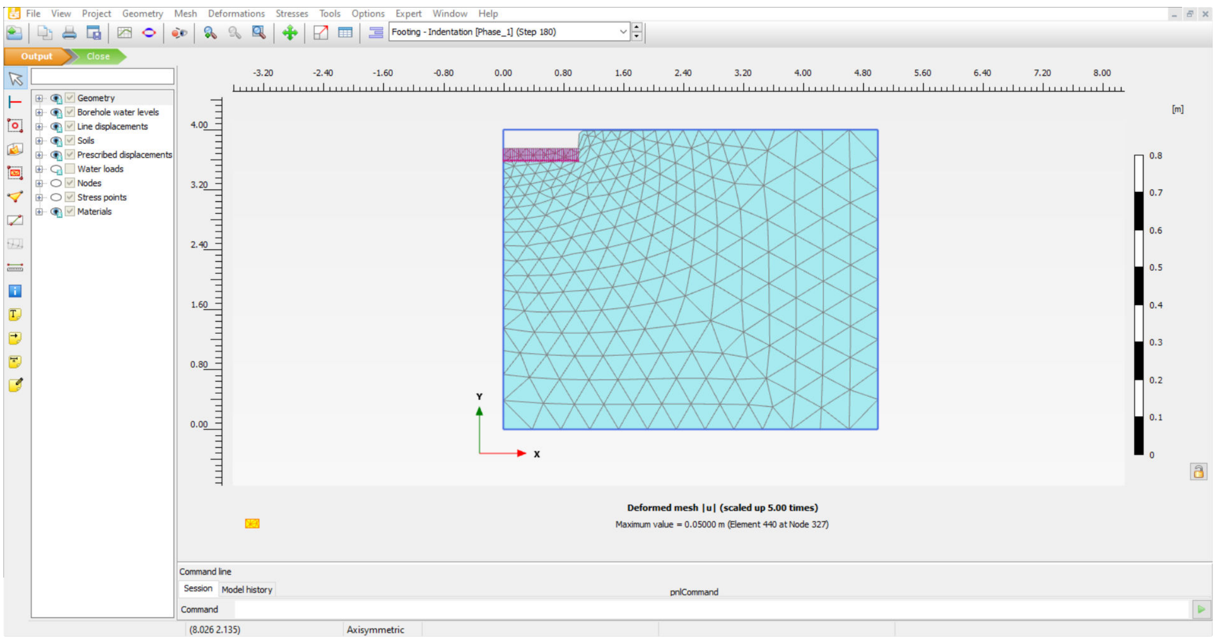



*Figure 19: Deformed mesh*

The deformed mesh is scaled to ensure that the deformations are visible.

5. Select the menu **Deformations** > **Total displacements** > **|u|**.

The plot shows colour shadings of the total displacements. The colour distribution is displayed in the legend at the right hand side of the plot.

**Note:** The legend can be toggled on and off by clicking the corresponding option in the **View** menu.

6. The total displacement distribution can be displayed in contours by clicking the corresponding button in the toolbar.
The plot shows contour lines of the total displacements, which are labelled. An index is presented with the displacement values corresponding to the labels.
7. Click the **Arrows** button .
The plot shows the total displacements of all nodes as arrows, with an indication of their relative magnitude.
8. Click the menu **Stresses** > **Principal effective stresses** > **Effective principal stresses**.

The plot shows the effective principal stresses at the stress points of each soil element with an indication of their direction and their relative magnitude as shown in Figure 20 (on page 30):

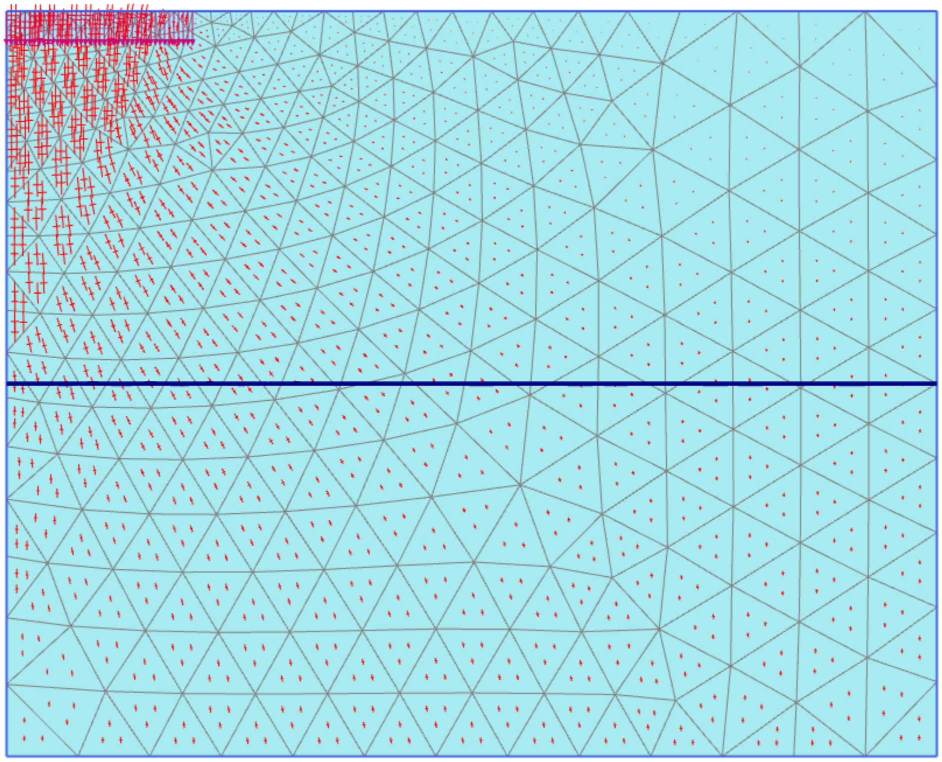


*Figure 20: Effective principal stresses*

9. Click the **Table** button on the toolbar.
A new window is opened in which a table is presented, showing the values of the principal stresses and other stress measures in each stress point of all elements.

**Note:**

- In addition to the total displacements, the **Deformations** menu allows for the presentation of **Incremental displacements**. The incremental displacements are the displacements that occurred within one calculation step (in this case the final step). Incremental displacements may be helpful in visualising an eventual failure mechanism.
- The plots of stresses and displacements may be combined with geometrical features, as available in the **Geometry** menu.

## 1.3 Case B: Flexible footing

The project is now modified so that the footing is modelled as a flexible plate. This enables the calculation of structural forces in the footing. The geometry used in this exercise is the same as the previous one, except that additional elements are used to model the footing. The calculation itself is based on the application of load rather than prescribed displacement. It is not necessary to create a new model; you can start from the previous model, modify it and store it under a different name. To perform this, follow these steps:

## 1.3.1 Modify the geometry

1. In the Input program select the **File** > **Save project as** menu. Enter a non-existing name for the current project file and click the **Save** button.
2. Go back to the **Structures mode** . Make sure you are in Select mode by clicking the **Select** button.
3. Right-click the prescribed displacement and select **Line displacement** > **Delete** as shown in Figure 21 (on page 31).

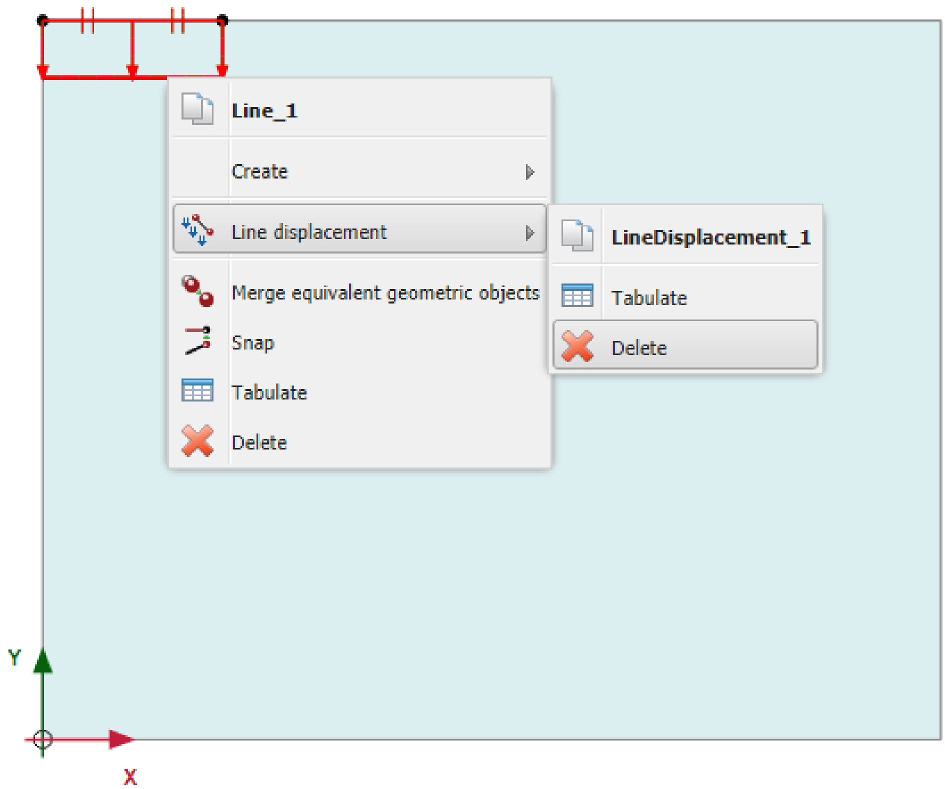



*Figure 21: Delete the line displacement option*

4. In the model right-click the line at the location of the footing. Select **Create** > **Create Plate** as shown in Figure 22 (on page 32).

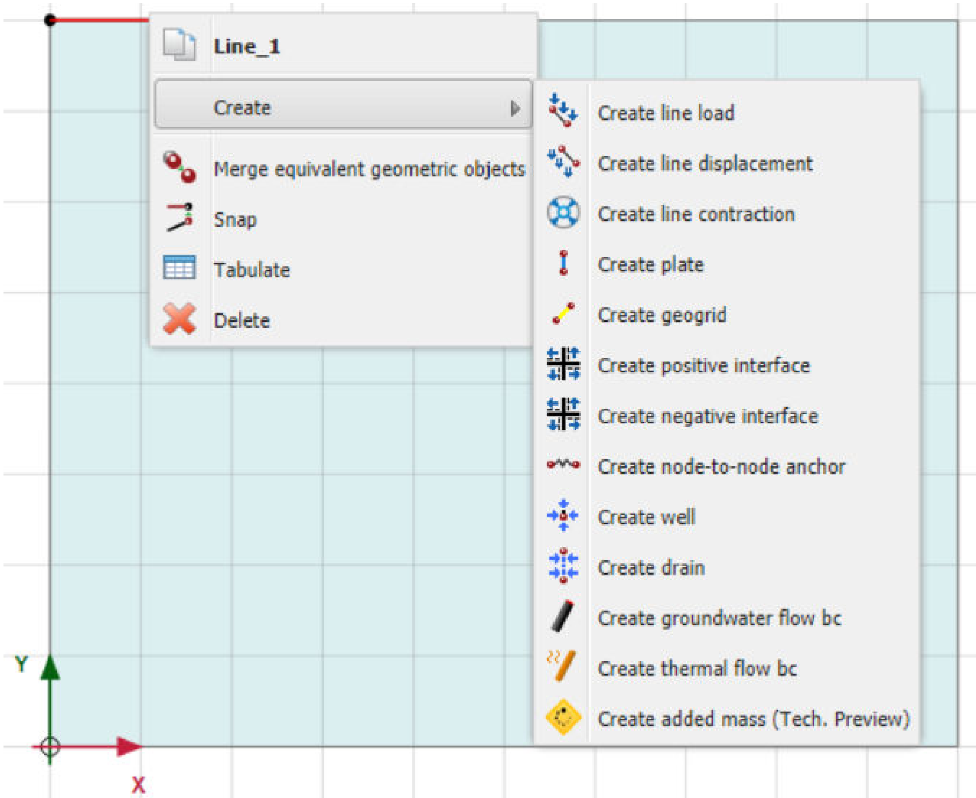



*Figure 22: Create Plate option*

A plate is created, which simulates the flexible footing.

5. In the model right-click again the line at the location of the footing and select **Create** > **Create Line load** as shown in Figure 23 (on page 33).

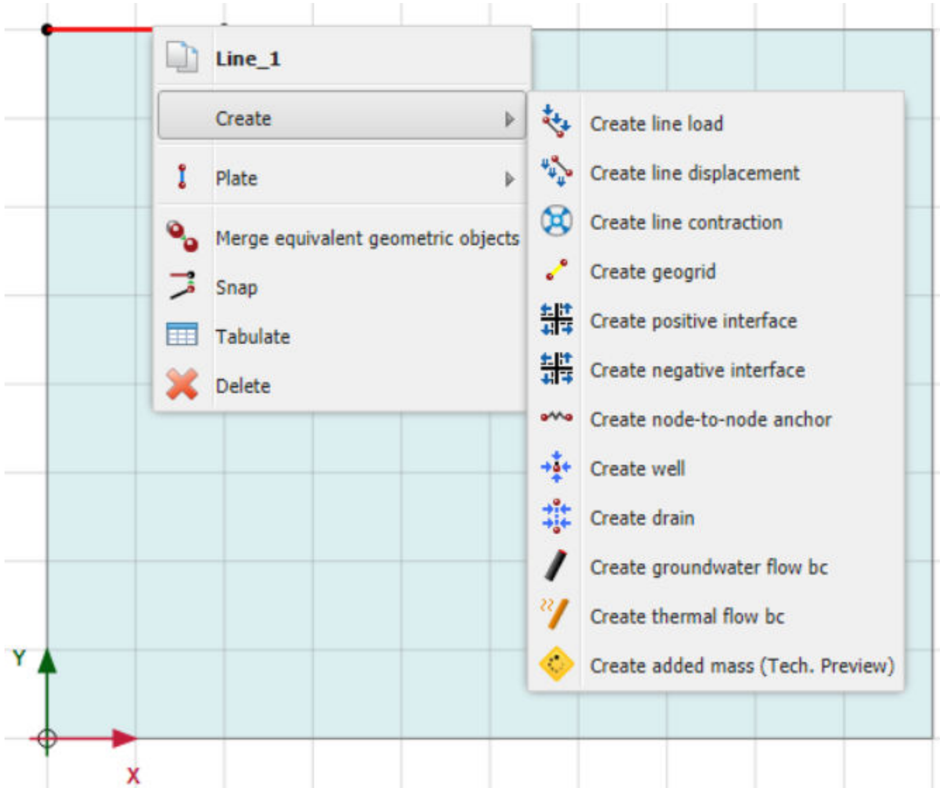



*Figure 23: Create line load option*

6. In the **Selection explorer** the default input value of the distributed load is -1.0 kN/m$^2$ in the y-direction. The input value will later be changed to the real value when the load is activated.

## 1.3.2 Add material properties for the footing

The material properties for the flexible footing are as follows:

**Table 2: Material properties of the footing**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| Unit weight | $w$ | 0.0 | kN/m/m |
| Prevent punching | - | No | |

| Mechanical | | | |
|---|---|---|---|
| Isotropic | - | Yes | - |
| Axial stiffness | $EA_1$ | $5 \cdot 10^6$ | kN/m |
| Bending stiffness | $EI$ | $8.5 \cdot 10^3$ | $kNm^2/m$ |
| Poisson's ratio | $\nu$ | 0.0 | - |

1. Click the **Materials** button in the side toolbar.
2. In the **Material sets** window, from the **Set type** drop-down menu, select **Plates**.
3. Click the **New** button.
   A new window appears where the properties of the footing can be entered.
4. Type `Footing` in the **Identification** box. The **Elastic** option is selected by default for the material type. Keep this option for this example.
5. Enter the properties as listed in Table 2 (on page 33). Keep parameters that are not mentioned in the table at their default values.
6. **Note:** The equivalent thickness is automatically calculated by PLAXIS 2D from the values of EA and EI. It cannot be defined manually.

   Click **OK**.

   The new data set now appears in the tree view of the **Material sets** window.
7. Drag the set called Footing to the drawing area and drop it on the footing. Note that the shape of the cursor changes to indicate that it is valid to drop the material set.

   **Note:** If the **Material sets** window is displayed over the footing and hides it, click on its header and drag it to another position.
8. Click **OK** to close the materials database.

## 1.3.3 Generate the mesh

In order to generate the mesh, follow these steps:

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh.
4. Click the **Close** tab to close the Output program.

**Note:** Regeneration of the mesh results in a redistribution of nodes and stress points.

## 1.3.4 Calculations

1. Proceed to the **Staged construction mode** .
2. Leave the initial phase as it is. The initial phase is the same as in the previous case.
3. Double-click the following phase (Phase_1) and enter an appropriate name for the phase ID. Keep the **Calculation type** as **Plastic** and keep the **Loading type** as **Staged construction**.
4. Close the **Phases** window.
5. In the **Staged construction mode** activate the load and plate.

   The model is shown- in Figure 24 (on page 35):

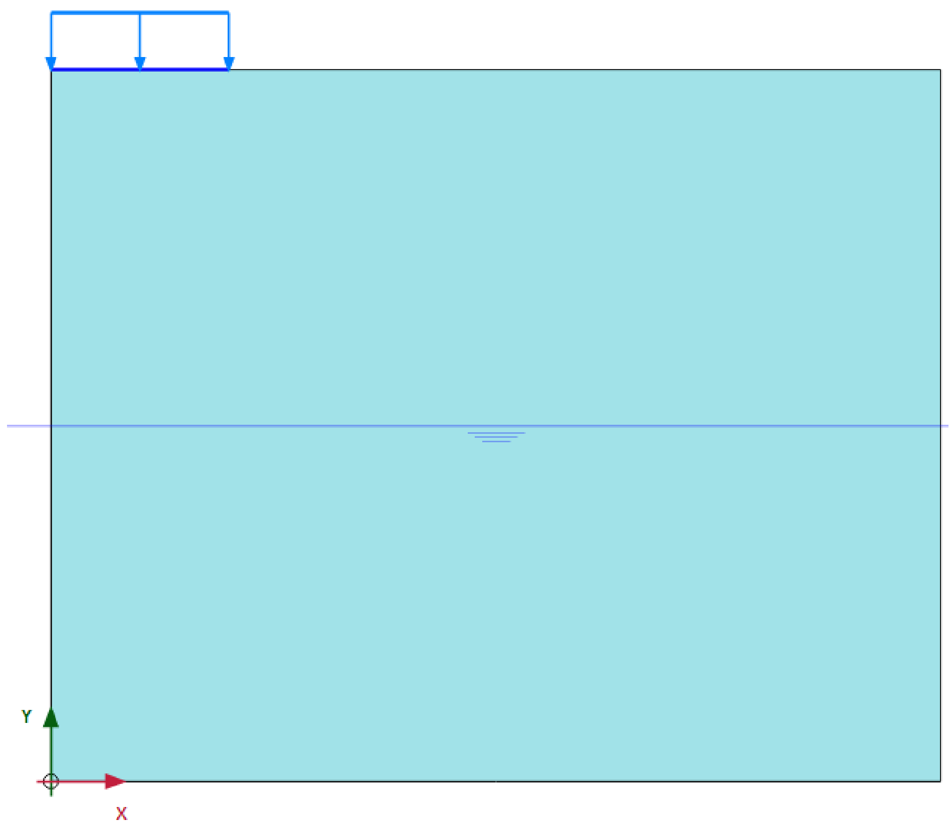



*Figure 24: Active plate and load in the model*

6. In the **Selection explorer** shown in Figure 25 (on page 36) assign -188 kN/m$^2$ to the vertical component of the line load. Note that, this gives a total load that is approximately equal to the footing force that was obtained from the first part of this tutorial. (188 kN/m$^2$ · π ·(1.0 m)$^2$ ≈ 590 kN).

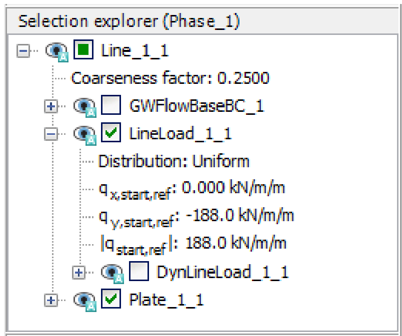



*Figure 25: Definition of the load components in the **Selection explorer***

7. No changes are required in the **Flow conditions** tabsheet.
   The calculation definition is now complete. Before starting the calculation it is advisable to select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow these steps:
8. Click the **Select points for curves** button in the side toolbar.
   As a result, all the nodes and stress points are displayed in the model in the Output program. The points can be selected either by directly clicking on them or by using the options available in the **Select points** window.
9. In the **Select points** window enter (0.0 4.0) for the coordinates of the point of interest and click **Search closest**.
   The nodes and stress points located near that specific location are listed.
10. Select the node at exactly (0.0 4.0) by checking the box in front of it. The selected node is indicated by **Node 4*** in the model when the **Selection labels** option is selected in the **Mesh** menu.

**Note:** Instead of selecting nodes or stress points for curves before starting the calculation, points can also be selected after the calculation when viewing the output results. However, the curves will be less accurate since only the results of the saved calculation steps will be considered. To select the desired nodes by clicking on them, it may be convenient to use the **Zoom in** option on the toolbar to zoom into the area of interest.

11. Click the **Update** button on the top left to return to the Input program.
12. Check if both calculation phases are marked for calculation by a blue arrow. If this is not the case click the symbol of the calculation phase or right-click and select **Mark for calculation** from the pop-up menu.
13. Click the **Calculate** button to start the calculation.
14. Click the **Save** button to save the project after the calculation has finished.

## 1.3.5 View the calculation results

1. After the calculation the results of the final calculation step can be viewed by clicking the **View calculation results** button. Select the plots that are of interest. The displacements and stresses should be similar to those obtained from the first part of the exercise.
2. Click the **Select structures** button in the side toolbar and double click the footing.
   A new window opens in which either the displacements or the bending moments of the footing may be plotted (depending on the type of plot in the first window).

3. Note that the menu has changed. Select the various options from the **Forces** menu to view the forces in the footing.

> **Note:** Multiple (sub-)windows may be opened at the same time in the Output program. All windows appear in the list of the **Window** menu. PLAXIS 2D follows the Windows standard for the presentation of sub-windows (**Cascade**, **Tile**, **Minimize**, **Maximize**, etc).

## 1.3.6 Generate a load-displacement curve

In addition to the results of the final calculation step it is often useful to view a load-displacement curve. In order to generate the load-displacement curve, follow these steps:

1. Click the **Curves manager** button in the toolbar.
   The **Curves manager** window pops up.
2. In the **Charts** tabsheet, click **New**.

   The **Curve generation** window pops up as shown in Figure 26 (on page 37).

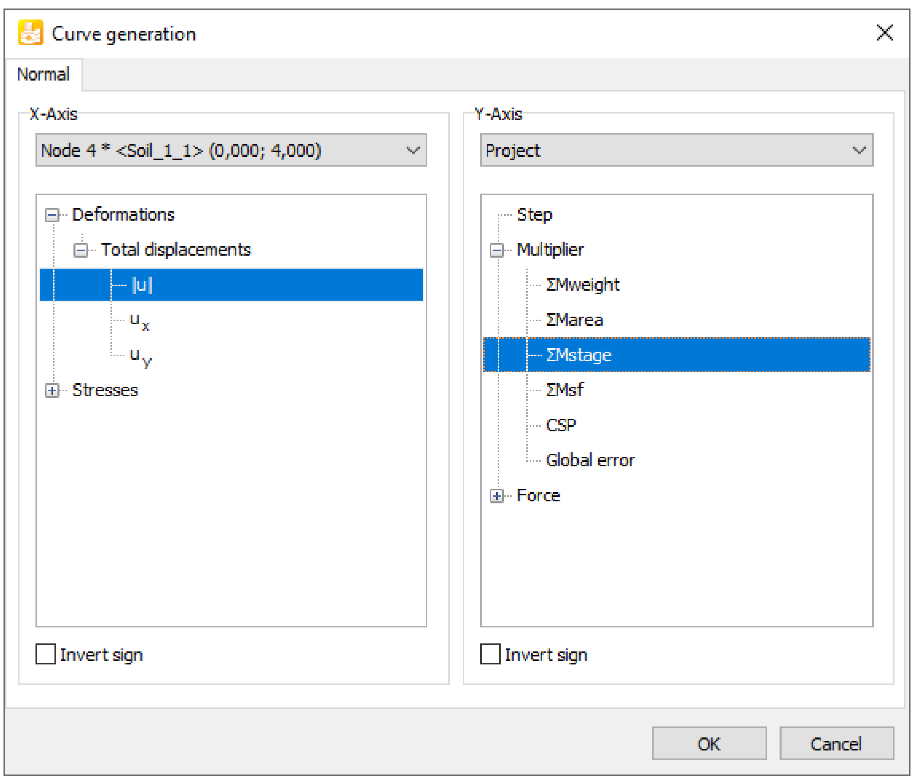



*Figure 26: **Curve generation** window*

3. For the x-axis, select **Node 4* (0.00 / 4.00)** from the drop-down menu. Select the **Deformations** > **Total displacements** > **|u|**.
4. For the y-axis, select the Project option from the drop-down menu. Select the **Multipliers** > **ΣMstage** option. ΣMstage is the proportion of the specified changes that has been applied. Hence the value will range from 0 to

1, which means that 100% of the prescribed load has been applied and the prescribed ultimate state has been fully reached.

5. Click **OK** to accept the input and generate the load-displacement curve.
As a result the curve of is plotted as shown in Figure 27 (on page 38):

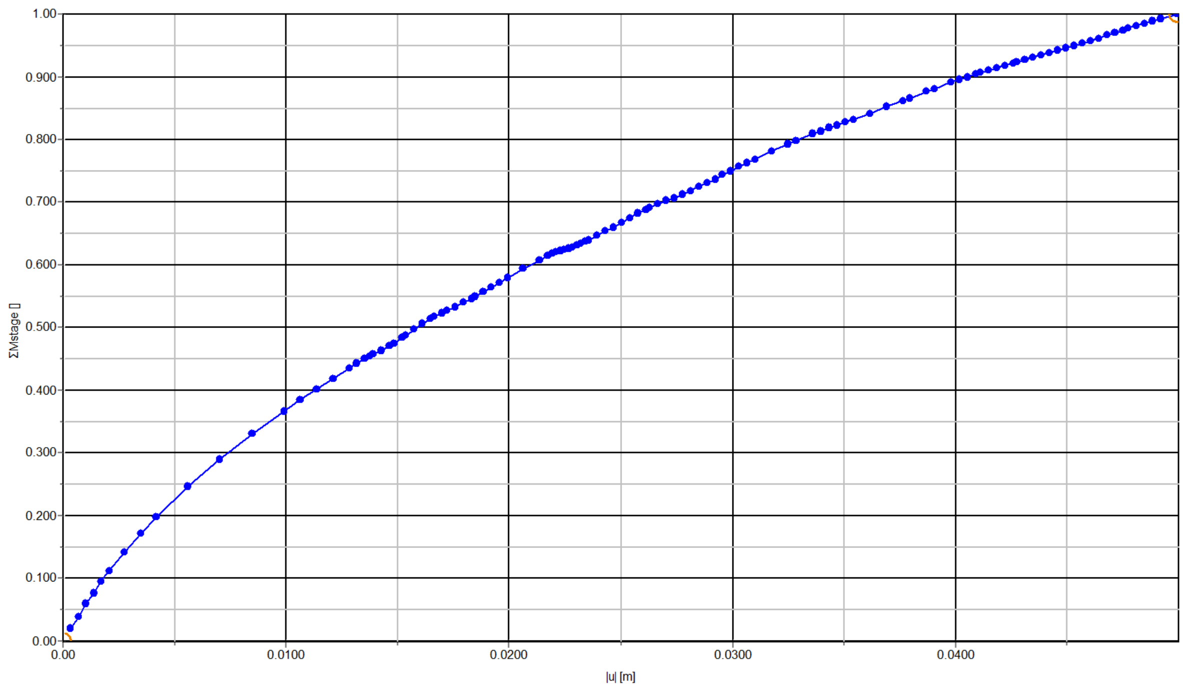



*Figure 27: Load-displacement curve for the footing*

**Note:**

You can re-enter the **Settings** window (in the case of a mistake, a desired regeneration or modification) by:

- Double click the curve in the legend of the chart OR
- Select the menu **Format** > **Settings**.

The properties of the chart can be modified in the **Chart** tab sheet whereas the properties curve can be modified in the corresponding tab sheet.

2

# Drained and undrained stability of an embankment

In this chapter the construction of an embankment on clay is simulated. The clay will first be considered a drained material and then an undrained material. For both cases the factor of safety will be determined. This would for instance give an indication of both the long term and short term stability of the embankment.

**Objectives**

- Modelling **Drained** and **Undrained** soil behaviour.
- Changing material sets during the calculation.
- Calculating a factor of safety.

**Geometry**

Figure 28 (on page 39) shows the layout of an embankment . The embankment is 4 m high and the crest of the embankment has a width of 2.0 m. The groundwater level is just below surface, but to simplify the problem it will be defined at ground level.

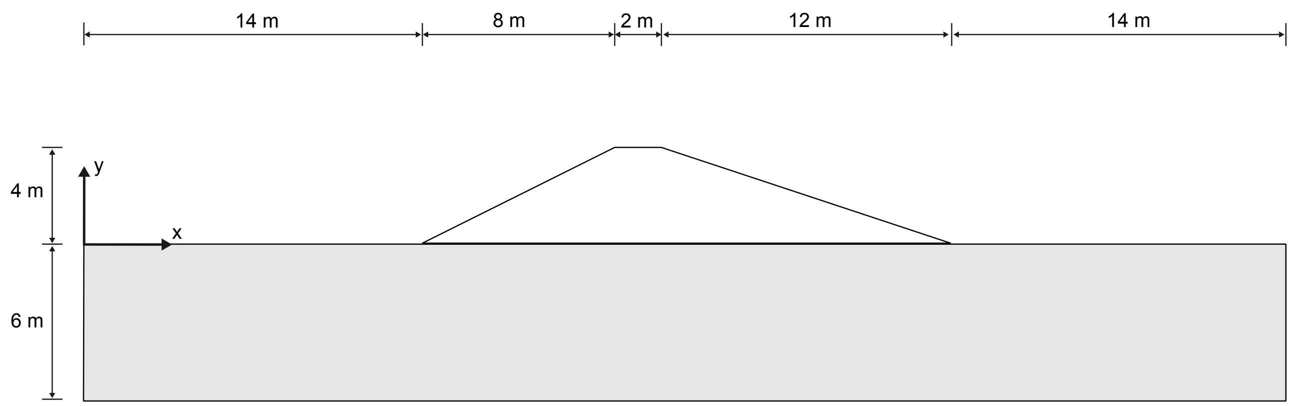



*Figure 28: Geometry of the project*

## 2.1 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 start** 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-Noded**).

4. Set the model dimensions to: $x_{min}$ = 0 m, $x_{max}$ = 50 m, $y_{min}$ = -6 m and $y_{max}$ = 4 m.
5. Keep the default values for units, constants and the general parameters and click **OK** to close the **Project properties** window.

## 2.2 Define the soil stratigraphy

The subsoil profile consists of a single clay layer extending until large depth. Since we are only interested in the stability of the embankment it is not necessary to model the clay layer until very large depth; the model has to be deep enough to allow the failure mechanism to form. Please note that for a deformation analysis a deeper model may be required as deformations due to the construction of the embankment will still occur at considerable depth.

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up as shown in Figure 29 (on page 40).
2. **Add** a single soil layer from **Top** = 0 to **Bottom** = -6.
3. Keep the **Head** for this borehole to 0. Hence, the groundwater level coincides with ground level.

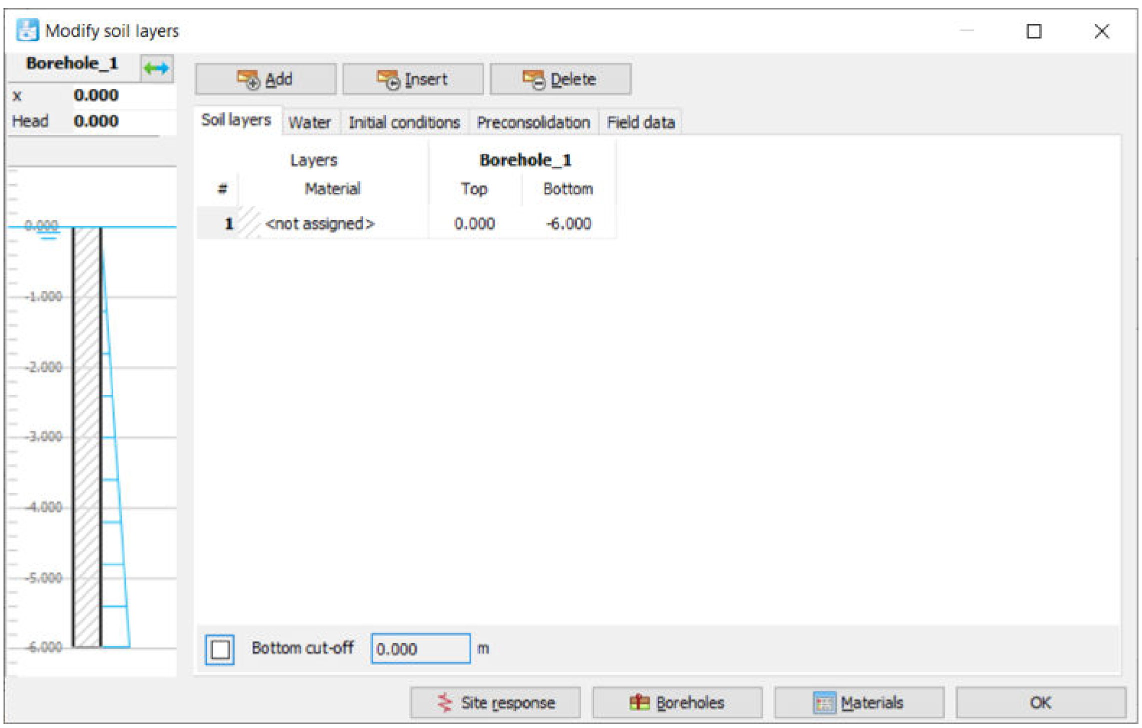



*Figure 29: The soil stratigraphy in the **Modify soil layers** window*

## 2.3 Create and assign material data sets

For this project three material sets have to be defined. One material to represent the sand of which the embankment will be constructed and two materials representing the subsoil: a drained and an undrained material set representing the clay in long term and short term conditions.

**Table 3: Material properties of the soil layer**

| Parameter | Name | Embankment | Clay layer | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | Hardening soil | Hardening soil | - |
| Type of material behaviour | - | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 13 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 16 | 13 | kN/m$^3$ |
| **Mechanical** | | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $15 \cdot 10^3$ | 5,600 | kN/m$^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $15 \cdot 10^3$ | 5,000 | kN/m$^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $45 \cdot 10^3$ | $20 \cdot 10^3$ | kN/m$^2$ |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | 1.0 | - |
| Cohesion | $c'_{ref}$ | 3 | 10 | kN/m$^2$ |
| Friction angle | $\varphi'$ | 30 | 25 | ° |
| **Initial** | | | | |
| $K_0$-determination | - | Automatic | Automatic | - |
| Overconsolidation ratio | *OCR* | 1.0 | 1.2 | - |

1. Select the **Show materials** button so that the **Materials sets** window shows up.
2. Using the **New** button define the two material sets as defined in the Table 3 (on page 41) .

3. In order to create the undrained material set for the clay layer select the drained material in the **Material sets** window and click the **Copy** button to duplicate the material set. In the copied material set, change the name and set the **Drainage type** to **Undrained (A)**.
4. Assign the material set representing the drained clay to the subsoil.

## 2.4 Create the embankment

We will now create the embankment. This is done by defining a soil polygon after which the polygon gets assigned the material set representing the embankment material.

1. Go to **Structures mode** .
2. From the tools side bar select the **Create soil polygon** option and then from the small popup button menu that appears select the **Create soil polygon** option again.
3. Now draw a polygon starting from (x y) = (14 0) continuing to (22 4), (24 4) and finally (36 0).
4. Now assign the embankment material to the polygon. This can either be done by opening the **Material sets** window and then drag and drop the material set onto the polygon, or by selecting the polygon and then in the **Selection explorer** set the **Material** option of the soil polygon to the embankment material.

## 2.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button to generate the mesh.
   The **Mesh options** window appears.
3. Select the **Fine** option in the **Element distribution** list and generate the mesh.
4. Click the **View mesh** button to view the mesh and the generated mesh is shown in Figure 30 (on page 42).

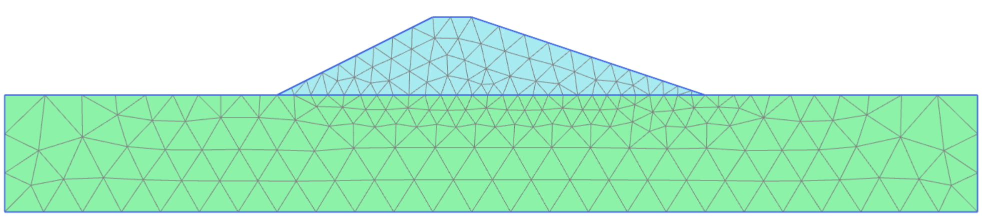


*Figure 30: The generated mesh*

5. Select the **Close** button on the top left of the Output program to close the mesh view.

## 2.6 Define and perform the calculation

The embankment construction is done in a single stage here, as we're only interested in the stability. Note that for settlement predictions it is better to divide the construction of the embankment in multiple phases, if necessary with periods of consolidation in between. The embankment construction has to be done both on drained and undrained subsoil, so a total of three calculation phases have to be defined: the initial phase and two construction phases.

### 2.6.1 Initial phase: Initial conditions

In the initial situation the embankment is not present yet. Since the subsoil consists of only 1 layer with a horizontal ground level the **K0 procedure** can be used to generate the initial stresses. As this is the default option, no changes have to be made to the initial phase. The model in the initial phase is shown in Figure 31 (on page 43).

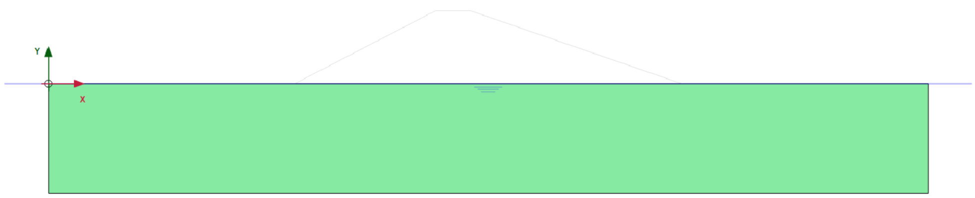



*Figure 31: Configuration of the initial phase*

### 2.6.2 Phase 1: Embankment construction on drained subsoil

1. Click the **Add phase** button to create a new phase .
   In the new phase the **Calculation Type** > **Plastic** analysis and the **Loading type** > **Staged construction** are used as a default.
2. Right-click on the embankment and from the popup menu select the **Activate** option to activate the soil representing the embankment. The model of phase 1 is shown in Figure 32 (on page 43).

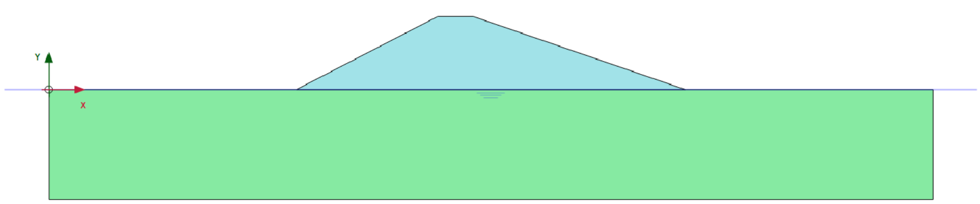



*Figure 32: Configuration of phase 1*

### 2.6.3 Phase 2: Embankment construction on undrained subsoil

In this second calculation phase the same embankment will be constructed, by now under the assumption of undrained behaviour of the subsoil. This implies that the material set of the subsoil must be changed, but also that this construction phase must start from the initial phase as it is an alternative calculation for phase 1 rather than a continuation of phase 1.

1. In the **Phase explorer** select the **Initial phase** and then create a new phase using the **Add phase** button. Because the initial phase was the selected phase the newly created phase 2 will start from the initial phase. In case by mistake phase 2 does start from phase 1, this can be changed double-clicking on **Phase 2** in the **Selection explorer** so that the **Phases** window opens. In the **General** section now set the option **Start from phase** to the initial phase and close the **Phases** window again.
2. Now the soil behaviour of the subsoil must be changed to undrained by assigning the undrained material set to the subsoil. There are several ways to do this:

   From the navigation side bar select the **Show materials** button. From the **Material sets** window that opens drag and drop the material set for undrained clay on the subsoil.

   or

   Right-click on the subsoil and from the popup menu(s) that open consecutively select the options **Soil** ( ), **Soil** ( ), **Set material** and finally the material representing the undrained subsoil to assign it.

   or

   Select the subsoil and in the **Selection explorer** change the **Material** under the **Soil** object for the material representing the undrained subsoil.
3. Finally, activate the embankment, and the resulting model is shown in Figure 33 (on page 44).

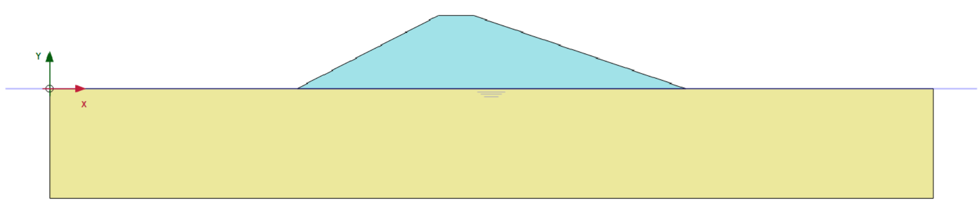



*Figure 33: Configuration of phase 2*

### 2.6.4 Calculate

Before starting the calculation it's recommended to choose some nodes or stress points to later evaluate the results in for instance load-displacement or stress-strain curves. In this project we will select a point halfway the left hand side slope to later evaluate the factor of safety.

1. Click the **Select points for curves** button on the navigation side bar.
2. Select a **Node** halfway the left hand side slope, hence around (x y) = (18 2).

3. Click the **Update** button on the top left to close the Output program and store the selected point.
4. Click the **Calculate** button to start the calculation.

## 2.7 Results

1. After the calculation ended, select the first phase and click the **View calculations results** button. The Output program now opens showing the deformed mesh after the construction of the embankment on the drained subsoil.
2. From the dropdown list on the button bar at the top now choose to see the results for Phase 2. Now the deformed mesh after construction of the embankment on undrained subsoil is shown.

   Figure 34 (on page 45) shows the deformed mesh for both phases. In case of the drained subsoil the embankment settles everywhere, but in case of the undrained subsoil the embankment settles in the middle but heaves near the toes. This makes perfect sense: the subsoil is now undrained and because of that there can be no volume change. Hence, if the subsoil settles in the middle due to the weight of the embankment it must heave somewhere else, typically just next to the embankment.

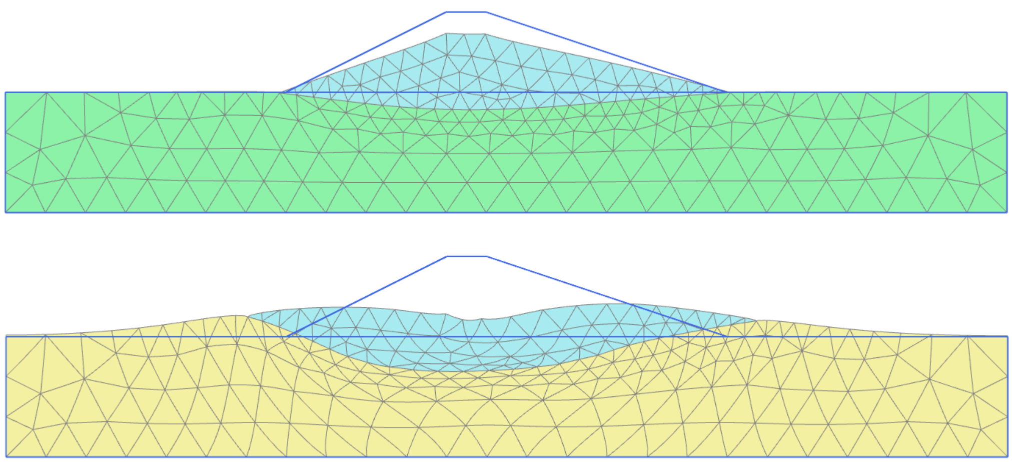


*Figure 34: Deformed mesh after phase 1 (top) and phase 2 (bottom)*

   It can also be seen that in case of the undrained subsoil the embankment seems to widen leading to a larger settlement at the top of the embankment. This is not immediately obvious and it will be discussed in more detail after performing a factor of safety analysis.
3. From the **Stresses** menu select the option **Pore pressures** and then $\mathbf{p_{excess}}$. This will show the excess pore pressures due to the undrained loading of the subsoil. By default the pore pressures are shown as iso-shadings but with the buttons and from the horizontal button bar at the top it is possible to see the principal directions of the pore pressures, either for all or a reduced number of stress points.

   Figure 35 (on page 46) shows that excess pore pressures have developed due to the construction of the embankment. The highest excess pore pressures are of course directly underneath the embankment, but also on either side of the toes of the embankment some excess pore pressures occur.

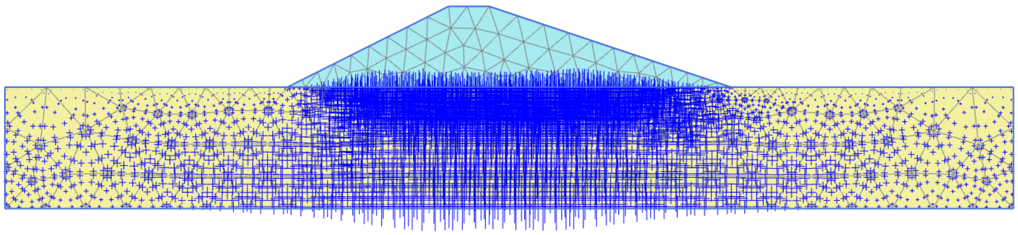


*Figure 35: Excess pore pressures after construction on undrained subsoil*

**4.** After viewing the results, close the Output program and return to the Input program.

## 2.8 Safety analysis

In the design of an embankment it is important to consider not only the settlements, but also the stability in terms of a factor of safety. It could be seen that the settlements are different for the construction of an embankment on drained or undrained subsoil and so it would be interesting to evaluate a global safety factor for both cases.

In structural engineering, the safety factor is usually defined as the ratio of the collapse load to the working load. For soil structures, however, this definition is not always useful. For embankments, for example, most of the loading is caused by soil weight and an increase in soil weight would not necessarily lead to collapse. Indeed, a slope of purely frictional soil will not fail in a test in which the self weight of the soil is increased (like in a centrifuge test). A more appropriate definition of the factor of safety is therefore:

$$Safety\ factor = \frac{S_{maximum\ \ available}}{S_{needed\ \ for\ equilibrium}}$$

Where S represents the shear strength. The ratio of the true strength to the computed minimum strength required for equilibrium (hence, the mobilized strength) is the safety factor that is conventionally used in soil mechanics. For soil models using the standard Coulomb failure condition, the safety factor is obtained as:

$$Safety\ factor = \frac{c - \sigma_n \tan(\varphi)}{c_r - \sigma_n \tan(\varphi_n)}$$

Where c and $\varphi$ are the input strength parameters and $\sigma_n$ is the actual normal stress component.

The parameters $c_r$ and $\varphi_r$ are reduced strength parameters that are just large enough to maintain equilibrium. The principle described above is the basis of the method of Safety that can be used in PLAXIS 2D to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:

$$\frac{c}{c_r} = \frac{\tan(\varphi)}{\tan(\varphi_r)} = \Sigma Msf$$

The reduction of strength parameters is controlled by a multiplier called ΣMsf that is increased step-by-step until failure occurs. The safety factor is then defined as the value of ΣMsf at failure, provided that at failure a more or less constant value is obtained for a number of successive load steps with ongoing deformations.

Note that for soil models that do no use the Coulomb failure criterion the concept of strength reduction remains the same, but with reduction of strength parameters specific to that soil model.

The **Safety** calculation option is available in the **Calculation type** drop-down menu in the **General** section of the **Phases** window. If the **Safety** option is selected the **Loading input** on the **Parameters** tabsheet is automatically set to **Incremental multipliers**, this means that the multiplier ΣMsf will be incremented until failure occurs. The other option for the **Loading input** is **Target-ΣMsf**, which means the multiplier ΣMsf will only increase until the specified target value and not until failure. However, this latter option is not used here.

To calculate the global safety factor for the road embankment for both cases, follow these steps:

1. Select Phase 1 in the **Phases** explorer.
2. Add a new calculation phase.
3. Double-click on the new phase to open the **Phases** window.
4. In the **Phases** window the selected phase is automatically selected in the **Start from phase_1** drop-down menu.
5. In the **General** subtree, set the **Calculation type** as **Safety** from the drop down menu.
6. The **Incremental multipliers** option is already selected in the **Loading input** box. The first increment of the multiplier that controls the strength reduction process, **Msf**, is set to 0.1.
7. In order to exclude existing deformations from the resulting failure mechanism, select the **Reset displacements to zero** option in the **Deformation control parameters** subtree.
8. The first safety calculation has now been defined.
9. Follow the same steps to create a new calculation phase that analyses the stability at the end of construction of the embankment on undrained subsoil.
10. Click the **Calculate** button to start the calculation .

The phases explorer window displaying the safety calculation is shown in Figure 36 (on page 47).

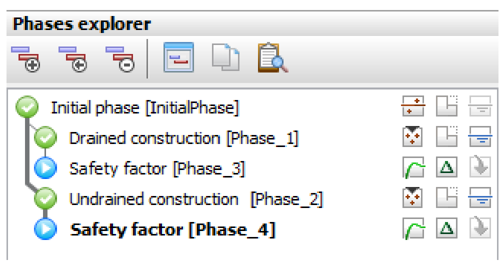



*Figure 36: Phases explorer displaying the Safety calculation phases*

**Note:**

Note that for a **Safety** phase the option **Use pressures from the previous phase** in the **Pore pressure calculation type** drop-down menu is automatically selected and grayed out indicating that this option cannot be changed. A **Safety** calculation always uses the same pore pressures as the phase for which the safety factor has to be calculated.

**Note:**

The default value of **Max steps** in a **Safety** calculation is 100. In contrast to a **Staged construction** calculation, the specified number of steps is always fully executed. In most **Safety** calculations, 100 steps are sufficient to arrive at a state of failure. If not, the number of steps can be increased to a maximum of 1000.

For most **Safety** analyses Msf = 0.1 is an adequate first step to start up the process. During the calculation process, the development of the total multiplier for the strength reduction, ΣMsf, is automatically controlled by the load advancement procedure.

## 2.8.1 Evaluation of safety analysis results

Due to the ongoing reduction of the soil strength, additional displacements are generated during a **Safety** calculation. The resulting total displacements do not have a physical meaning as they depend on the amount of load steps that was applied: more load steps means the calculation would push further into failure thus generating larger displacement while in reality a failing embankment would re-establish a new equilibrium with limited deformations. However, the incremental displacements in the final step (at failure) are very useful as they give an indication of the likely failure mechanism. The incremental displacements are the change of displacement per load increment. Typically this is a very small value as the load increments are small, but in case of failure the failure zone only needs a very small change of load to generate large changes of displacements, hence the failure zone then has large incremental displacement whereas anywhere else in the model the incremental displacements should be small.

In order to view the mechanism for the embankment on drained subsoil:

1. Select Phase 3, that is the **Safety** phase following on Phase 1, and click the **View calculation results** button .
2. In the Output program select the menu **Deformations** > **Incremental displacements** > **|Δu|**.

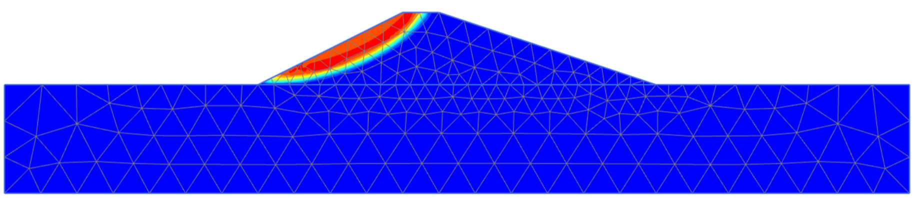


*Figure 37: Shadings of the displacement increments indicating the most applicable failure mechanism of the embankment after drained construction*

The resulting plot shown in Figure 37 (on page 48) gives a good impression of the failure mechanisms. The magnitude of the displacement increments is not relevant. From the results it can be seen that the slope on the left side of the embankment fails with a classical slip surface .

By choosing phase 4 from the drop down list at the toolbar the failure mechanism for the embankment constructed on undrained subsoil can be evaluated. The failure mechanism is no longer limited to just the embankment, but is in fact mostly a failure of the subsoil underneath the embankment. This also explains the widening of the embankment at the base as was observed earlier: the soil underneath the embankment fails and moves horizontally away from the center of the embankment shown in Figure 38 (on page 49).

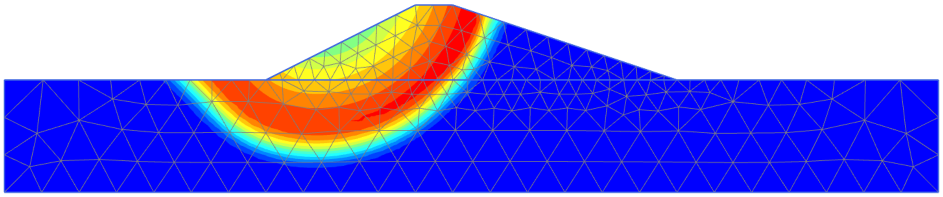


*Figure 38: Shadings of the displacement increments indicating the most applicable failure mechanism of the embankment after undrained construction*

The safety factor can be obtained from the **Calculation info** option of the **Project** menu. The **Multipliers** tabsheet of the **Calculation information** window represents the actual values of the load multipliers. The value of ΣMsf represents the safety factor, provided that this value is indeed more or less constant during the previous few steps.

The best way to evaluate the safety factor, however, is to plot a curve in which the parameter ΣMsf is plotted against the displacements of a certain node. Although the displacements are not relevant, they indicate whether or not a failure mechanism has developed.

In order to evaluate the safety factors for the three situations in this way, follow these steps:

1. Click the **Curves manager** button in the toolbar.
2. Click **New** in the **Charts** tabsheet.
3. In the **Curve generation** window, use from the dropdown list the previously selected **Node** for the x-axis. Select **Deformations** > **Total displacements** > **|u|**.
4. For the y-axis, select **Project** > **Multipliers** > **ΣMsf**. The **Safety** phases are considered in the chart.
5. Press **OK** to close the window and generate the chart.
6. Right-click on the chart and select the **Settings** option in the appearing menu. The **Settings** window pops up.
7. In the tabsheet corresponding to the curve click the **Phases** button.
8. In the **Select phases** window select only Phase 3 as shown in Figure 39 (on page 49):

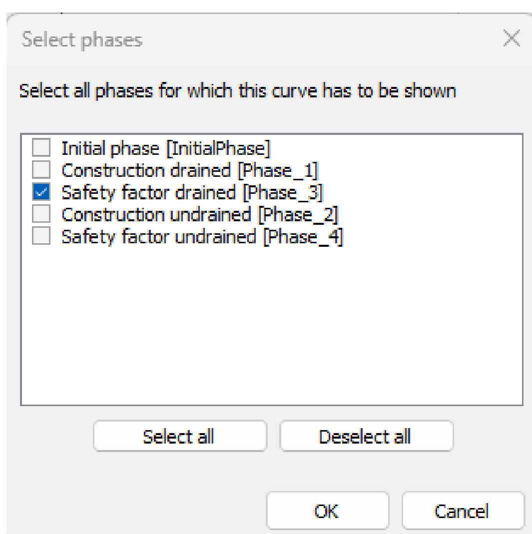



*Figure 39: The Select phases window*

9. Click **OK** to close the **Select phases** window.
10. In the **Settings** window change the titles of the curve in the corresponding tabsheet.
11. Click the **Add curve** button and select the **From current project** option in the appearing menu. Define the curve for the phase 4 by following the same steps as described for phase 3.
12. In the **Chart** tabsheet shown in Figure 40 (on page 50), set the scaling of the x-axis to **Manual** and set the value of **Maximum** to 0.5:

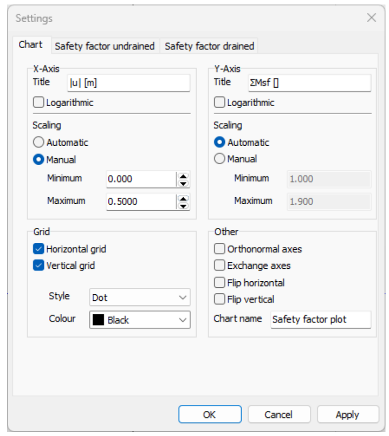



*Figure 40: The Chart tabsheet in the Settings window*

13. Click **Apply** to update the chart according to the changes made and click **OK** to close the **Settings** window.
14. C
15. The plot is shown as follows in Figure 41 (on page 50):

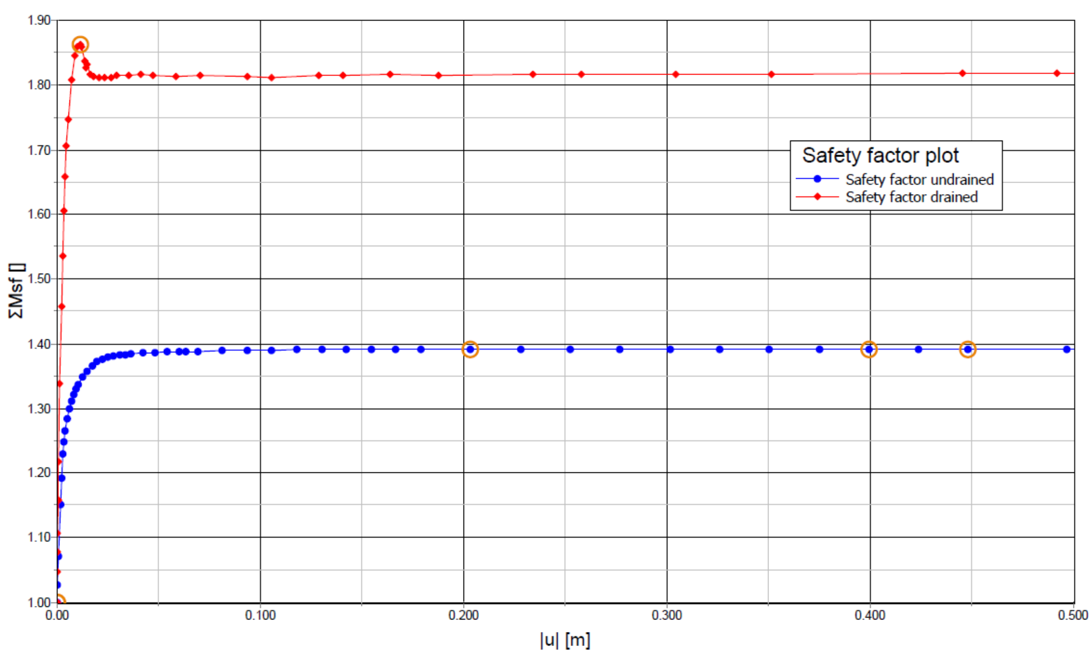



*Figure 41: Evaluation of safety factor*

The maximum displacements plotted are not relevant. It can be seen that for both curves a more or less constant value of ΣMsf is obtained. Hovering the mouse cursor over a point on the curves, a box shows up with the exact value of ΣMsf as well as the calculation phase. With the latter it can be determined that the upper curve with a factor of safety of 1.8 is phase 3, hence the embankment on drained soil. Similarly the lower curve with a factor of safety of 1.4 is phase 4, the embankment on undrained soil.

# 3 Submerged construction of an excavation

This tutorial illustrates the use of PLAXIS 2D for the analysis of submerged construction of an excavation. Most of the program features that were used in Tutorial 1 will be utilised here again. In addition, some new features will be used, such as the use of interfaces and anchor elements, the generation of water pressures and the use of multiple calculation phases. The new features will be described in full detail, whereas the features that were treated in Tutorial 1 will be described in less detail. Therefore it is suggested that Tutorial 1 should be completed before attempting this exercise.

**Objectives**

- Modelling soil-structure interaction using the Interface feature.
- Advanced soil models (Soft Soil model and Hardening Soil model).
- **Undrained (A)** drainage type.
- Defining **Fixed-end-anchor**.
- Creating and assigning material data sets for anchors.
- Simulation of excavation (cluster de-activation).

**Geometry**

This tutorial concerns the construction of an excavation close to a river. The submerged excavation is carried out in order to construct a tunnel by the installation of prefabricated tunnel segments which are 'floated' into the excavation and 'sunk' onto the excavation bottom. The excavation is 30 m wide and the final depth is 20 m. It extends in longitudinal direction for a large distance, so that a plane strain model is applicable. The sides of the excavation are supported by 30 m long diaphragm walls, which are braced by horizontal struts at an interval of 5 m. Along the excavation a surface load is taken into account. The load is applied from 2 m from the diaphragm wall up to 7 m from the wall and has a magnitude of 5 kN/m$^2$/m.

The upper 20 m of the subsoil consists of soft soil layers, which are modelled as a single homogeneous clay layer. Underneath this clay layer there is a stiffer sand layer, which extends to a large depth and 30 m of the sand layer are considered in the model which is shown in Figure 42 (on page 53).

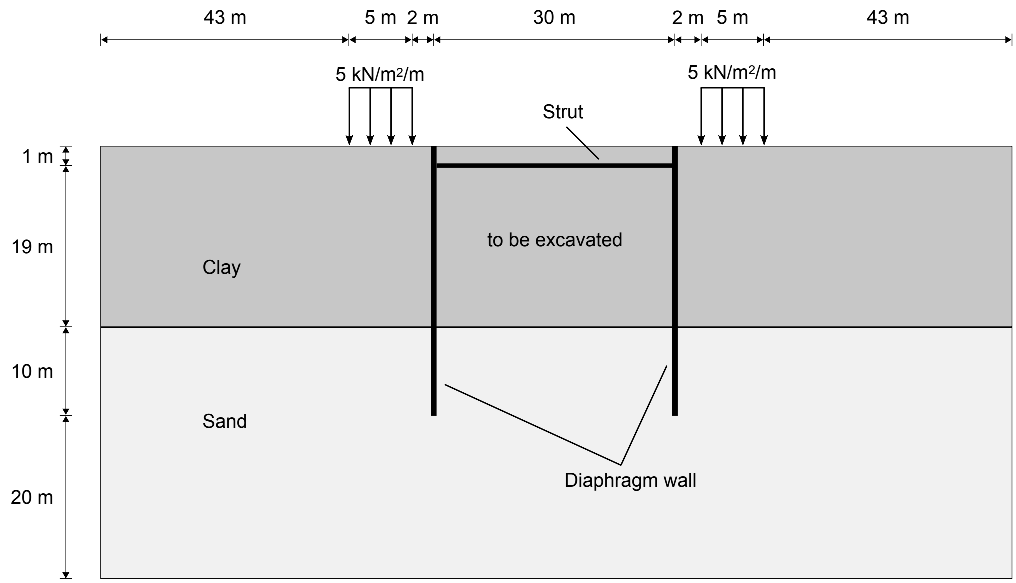



*Figure 42: Geometry model of the situation of a submerged excavation*

Since the geometry is symmetric, only one half (the left side) is considered in the analysis. The excavation process is simulated in three separate excavation stages. The diaphragm wall is modelled by means of a plate, such as used for the footing in the previous tutorial. The interaction between the wall and the soil is modelled at both sides by means of interfaces. The interfaces allow for the specification of a reduced wall friction compared to the friction in the soil. The strut is modelled as a spring element for which the normal stiffness is a required input parameter.

## 3.1 Create new project

1. Start PLAXIS 2D by double clicking the icon of the Input program .
2. Click **Start a new project**.
3. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
4. In the **Model** tabsheet keep the default options for **Model** (Plane strain}, and **Elements** (15-Node).
5. Set the model **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 65 m, $y_{min}$ = -30 m and $y_{max}$ = 20 m.
6. Keep the default values for units and constants and click **OK**.

The project is created with the given properties. The **Project properties** window closes and the **Soil mode** view will be shown, where the soil stratigraphy can be defined.

## 3.2 Define the soil stratigraphy

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Add the top soil layer and specify its height by setting the top level to 20 m and the bottom level to 0 m.
3. Add the bottom soil layer and specify its height by keeping the top level at 0 m and by setting the bottom level to -30 m.
4. Set the **Head** in the borehole to 18 m.

Next the material data sets are defined and assigned to the soil layers, see Create and assign material data sets (on page 54).

## 3.3 Create and assign material data sets

Two data sets need to be created; one for the clay layer and one for the sand layer.

The layers have the following properties as shown in Table 4 (on page 54):

**Table 4: Material properties of the sand and clay layer and the interfaces**

| Parameter | Name | Clay | Sand | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | Model | Hardening Soil | Hardening soil | - |
| Drainage type | Type | Undrained (A) | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 17 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 18 | 20 | $kN/m^3$ |
| **Mechanical** | | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $4 \cdot 10^3$ | $40 \cdot 10^3$ | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $3.3 \cdot 10^3$ | $40 \cdot 10^3$ | $kN/m^2$ |

| **Mechanical** | | | | |
|---|---|---|---|---|
| Unloading / reloading stiffness | $E_{ur}^{ref}$ | $12 \cdot 10^3$ | $120 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu_{ur}$ | 0.15 | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 1.0 | 0.5 | - |
| Cohesion (constant) | $c'_{ref}$ | 1 | 0 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 25 | 32 | ° |
| Dilatancy angle | $\psi$ | 0 | 2 | ° |
| $K_0$-value for normal consolidation | $K_0^{nc}$ | 0.5774 | 0.4701 | - |
| **Groundwater** | | | | |
| Data set | - | Standard | Standard | - |
| Soil type | - | Coarse | Coarse | - |
| Use defaults | - | None | None | - |
| Permeability in horizontal direction | $k_x$ | $1 \cdot 10^{-3}$ | 1 | m/day |
| Permeability in vertical direction | $k_y$ | $1 \cdot 10^{-3}$ | 1 | m/day |
| **Interfaces** | | | | |
| Srength determination | - | Manual | Manual | - |
| Interface reduction factor | $R_{inter}$ | 0.5 | 0.67 | - |
| **Initial** | | | | |
| $K_0$ determination | - | Automatic | Automatic | - |
| Pre-overburden pressure | $POP$ | 5 | 0 | $kN/m^2$ |
| Over-consolidation ratio | $OCR$ | 1 | 1 | - |

To create the material sets, follow these steps:

1. Click the **Materials** button in the **Modify soil layers** window.
The **Material sets** window pops up, where the **Soil and interfaces** option is selected by default as the **Set type**.
2. Click the **New** button in the **Material sets** window to create a new data set.
3. For the clay layer, enter `Clay` for the **Identification** and select Hardening Soil as the **Soil model**. Set the **Drainage type** to Undrained (A).
4. Enter the properties of the clay layer, as listed in Table 4 (on page 54), in the **General**, **Mechanical** and **Flow parameters** tabsheets.
5. Click the **Interfaces** tab. Select the **Manual** option in the **Strength** drop-down menu. Enter a value of 0.5 for the parameter $R_{inter}$.

   This parameter relates the strength of the soil to the strength in the interfaces, according to the equations: $tan(\varphi_{interface}) = R_{inter} tan\ (\varphi_{soil})$ and $c_{inter} = R_{inter} c_{soil}$ where: $c_{soil} = c_{ref}$ , see Table 4 (on page 54)

   Hence, using the entered $R_{inter}$-value gives a reduced interface friction (wall frictions) and interface cohesion (adhesion) compared to the friction angle and the cohesion in the adjacent soil.
6. In the **Initial** tabsheet keep the default option for the $K_0$ determination and the default value for the overconsolidation ratio (OCR). Set the pre-overburden pressure (POP) value to 5.0.
7. For the sand layer, enter `Sand` for the **Identification** and again select Hardening soil as the **Soil model**. The drainage type should be set to Drained.
8. Enter the properties of the sand layer, as listed in Table 4 (on page 54), in the corresponding edit boxes of the **General** and **Mechanical** tabsheet.
9. Click the **Interfaces** tab. In the **Strength** box, select the **Manual** option. Enter a value of 0.67 for the parameter $R_{inter}$. Close the data set.
10. Assign the material datasets to the corresponding soil layers.

**Note:**

- When the **Rigid** option is selected in the **Strength** drop-down, the interface has the same strength properties as the soil ($R_{inter}$ = 1.0).
- Note that a value of $R_{inter} < 1.0$ , reduces the strength as well as the stiffness of the interface (for more info see the corresponding section in the Reference Manual .
- Instead of accepting the default data sets of interfaces, data sets can directly be assigned to interfaces in the Model explorer by selecting **Interfaces** > **Interface_#_#** > **Material mode**.

## 3.4 Define the structural elements

The creation of diaphragm walls, strut, surface load and excavation levels is described below.

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode**.

## 3.4.1 To define the diaphragm wall:

A diaphragm wall with the following material properties mentioned in Table 5 (on page 57) has to be defined:

**Table 5: Material properties of the diaphragm wall (plate)**

| Property | Name | Value | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| Weight | *w* | 10 | kN/m/m |
| Prevent punching | - | No | - |
| **Mechanical** | | | |
| Isotropic | - | Yes | - |
| Axial stiffness | $EA_1$ | $7.5 \cdot 10^6$ | kN/m |
| Bending stiffness | $EI$ | $1.0 \cdot 10^6$ | kNm²/m |
| Poisson's ratio | $\nu$ | 0.0 | - |

1. Click the **Create structure** button in the side toolbar.
2. In the expanded menu select the **Create plate** as shown in Figure 43 (on page 57).

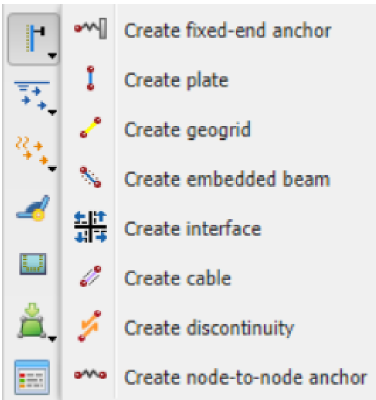



*Figure 43: Create plate option*

3. In the drawing area move the cursor to position (50 20) at the upper horizontal line and click. Move 30 m down (50 -10) and click. Click the right mouse button to finish the drawing.

4. Click the **Show materials** button in the side toolbar. Set the **Set type** parameter in the **Material sets** window to **Plates** and click the **New** button. Enter `Diaphragm wall` as an **Identification** of the data set and enter the properties as given in Table 5 (on page 57).
5. Click **OK** to close the data set.
6. Drag the **Diaphragm wall** data set to the wall in the geometry and drop it as soon as the cursor indicates that dropping is possible.
7. Click **OK** to close the **Material sets** window.

> **Note:** In general, only one point can exist at a certain coordinate and only one line can exist between two points. Coinciding points or lines will automatically be reduced to single points or lines. More information is available in the Reference Manual

### 3.4.2 To define the interfaces:

1. Right-click on the plate representing the diaphragm wall.

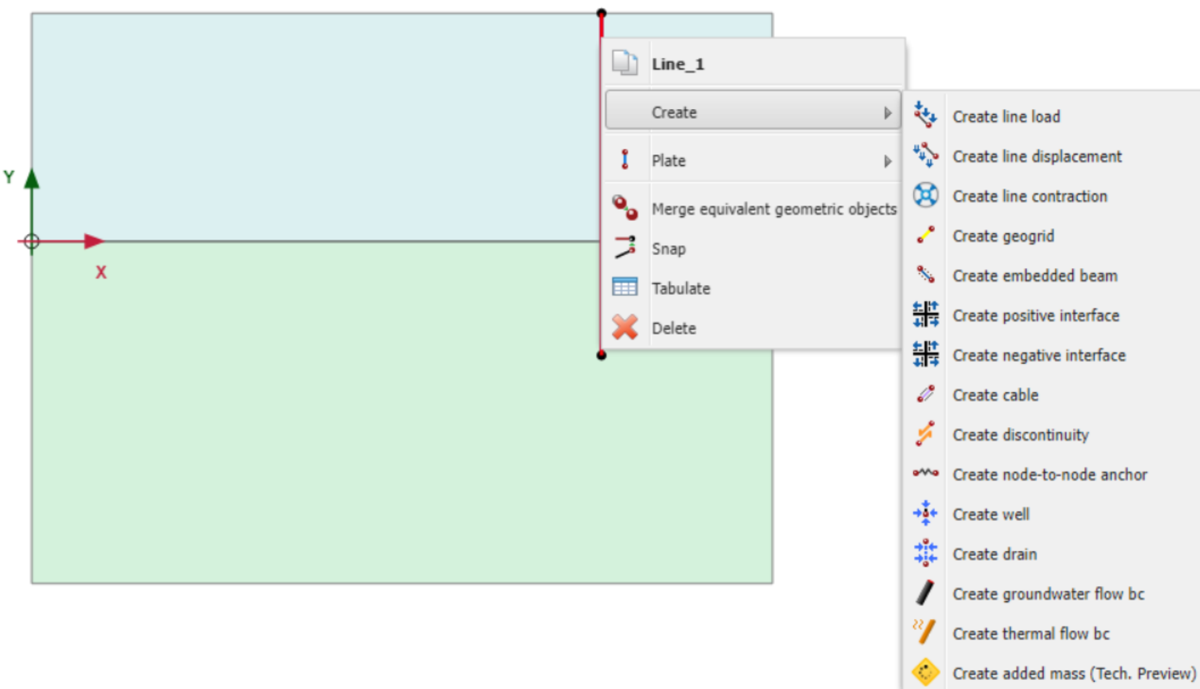



*Figure 44: Create positive interface*

2. Select **Create** > **Positive interface**.
3. In the same way assign a negative interface as well as shown in Figure 44 (on page 58).

> **Note:**
>
> - In order to identify interfaces at either side of a geometry line, a positive sign ( ⊕ ) or negative sign ( ⊖ ) is added. This sign has no physical relevance or influence on the results.

- A **Virtual thickness factor** can be defined for interfaces. This is a purely numerical value, which can be used to optimise the numerical performance of the interface. To define it, select the interface in the drawing area and specify the value to the **Virtual thickness factor** parameter in the **Selection explorer**. Non-experienced users are advised not to change the default value. For more information about interface properties see the Reference Manual - *Chapter 5 - Advanced Geometric Modelling options.*

## 3.4.3 To define the excavation levels:

1. Click the **Create line** button in the side toolbar.
2. To define the first excavation stage move the cursor to position (50 18) at the wall and click. Move the cursor 15 m to the right (65 18) and click again. Click the right mouse button to finish drawing the first excavation stage.
3. To define the second excavation stage move the cursor to position (50 10) and click. Move to (65 10) and click again. Click the right mouse button to finish drawing the second excavation stage.
4. The third excavation stage is automatically defined as it corresponds to the boundary between the soil layers (y = 0).

## 3.4.4 To define the strut:

You will define a strut with the following material properties shown in Table 6 (on page 59) .

**Table 6: Material properties of the strut (anchor)**

| Property | Name | Strut | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| **Mechanical** | | | |
| Out-of-plane spacing | $L_{spacing}$ | 5 | m |
| Axial stiffness | $EA$ | $2 \cdot 10^{6}$ | kN |

1. Click the **Create structure** button in the side toolbar and select the **Create fixed-end anchor** in the expanded menu.
2. Move the cursor to (50 19) and click the left mouse button. A fixed-end anchor is added, being represented by a rotated T with a fixed size.

3. Click the **Show materials** button in the side toolbar. Set the **Set type** parameter in the **Material sets** window to **Anchors** and click the **New** button. Enter `Strut` as an **Identification** of the data set and enter the properties as given in Table 6 (on page 59). Click **OK** to close the data set.
4. Click **OK** to close the **Material sets**.
5. Make sure that the fixed-end anchor is selected in the drawing area.
6. In the **Selection explorer** assign the material data set to the strut by selecting the corresponding option in the **Material** drop-down menu.
   The anchor is oriented in the model according to the $Direction_x$ and $Direction_y$ parameters in the **Selection explorer** as shown in Figure 45 (on page 60). The default orientation is valid in this tutorial.

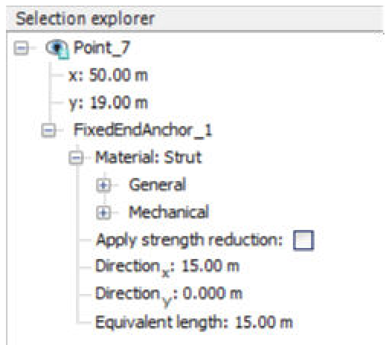



*Figure 45: Parameters for fixed-end anchors in the Selection explorer*

7. Enter an **Equivalent length** of 15m corresponding to half the width of the excavation.

**Note:** The **Equivalent length** is the real distance between the connection point and the fixed end point. For an excavation problem that is typically half the width of the excavation as the axis of symmetry in the middle of the excavation is considered fixed.

## 3.4.5 To define the distributed load:

1. Click the **Create load** button in the side toolbar.
2. Select the **Create line load** option in the expanded menu to define a distributed load.
3. Move the cursor to (43 20) and click, then move the cursor 5m to the right to (48 20) and click again. Right-click to finish the drawing.
4. In the **Selection explorer** assign a value of -5 kN/m/m to the y-component of the load ($q_{y,start,ref}$) as shown in Figure 46 (on page 61).

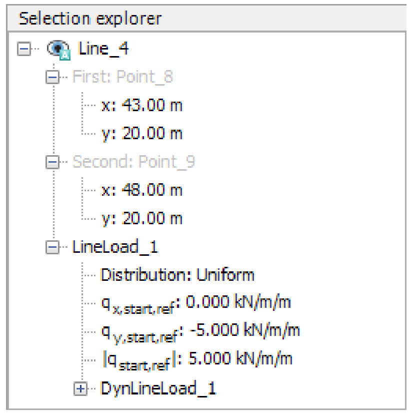



*Figure 46: Components of the distributed load in the Selection explorer*

## 3.5 Generate the mesh

In order to generate the mesh, follow these steps:

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 47 (on page 62).

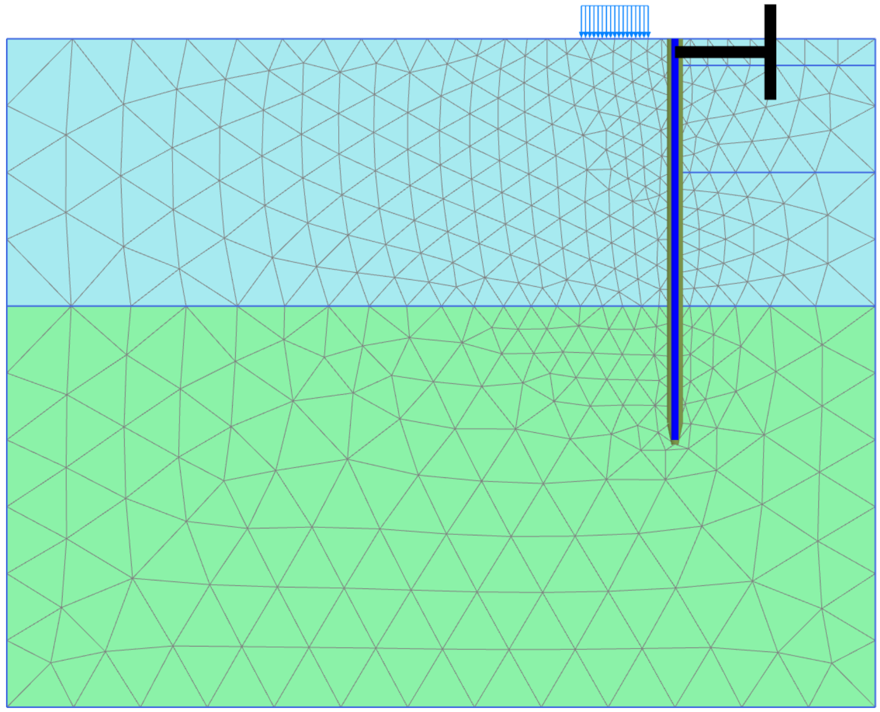


*Figure 47: The generated mesh*

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

## 3.6 Define and perform the calculation

In practice, the construction of an excavation is a process that can consist of several phases. First, the wall is installed to the desired depth. Then some excavation is carried out to create space to install an anchor or a strut. Then the soil is gradually removed to the final depth of the excavation. Special measures are usually taken to keep the water out of the excavation. Props may also be provided to support the retaining wall.

In PLAXIS 2D, these processes can be simulated with the **Staged construction** loading type available in the **General** subtree of the **Phases** window. It enables the activation or deactivation of weight, stiffness and strength of selected components of the finite element model. Note that modifications in the **Staged construction** mode of the program are possible only for this type of loading. The current tutorial explains the use of this powerful calculation option for the simulation of excavations.

### 3.6.1 Initial phase

1. Click the **Staged construction** tab to proceed with the definition of calculation phases.
   The initial phase has already been created.

2. Keep its calculation type as **K0 procedure.** Make sure all the soil volumes are active and all the structural elements and load are inactive.

### 3.6.2 Phase 1: External load

1. In the **Phases explorer** click the **Add phase** button to create a new phase.
The default settings are valid for this phase. In the model the full geometry is active except for the wall, interfaces, strut and load.
2. Click the **Select multiple objects** button in the side toolbar. In the appearing menu select **Select line** > **Select plates** as shown in Figure 48 (on page 63).

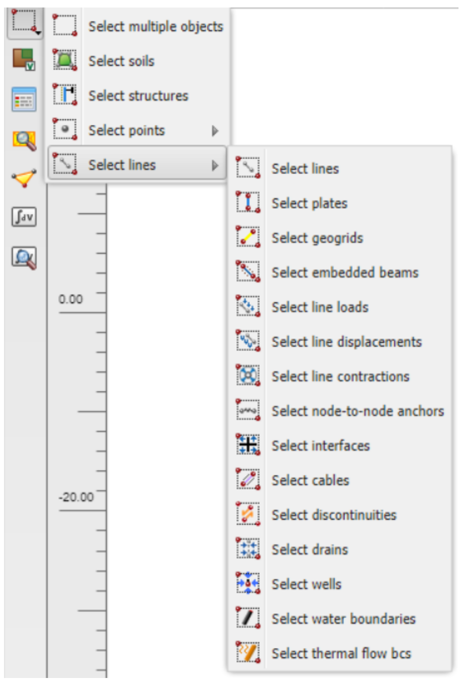



*Figure 48: Select multiple objects*

3. In the drawing area define a rectangle that includes all the plate elements as shown in Figure 49 (on page 64).
4. Right-click the wall in the drawing area and select the **Activate option** from the context menu.

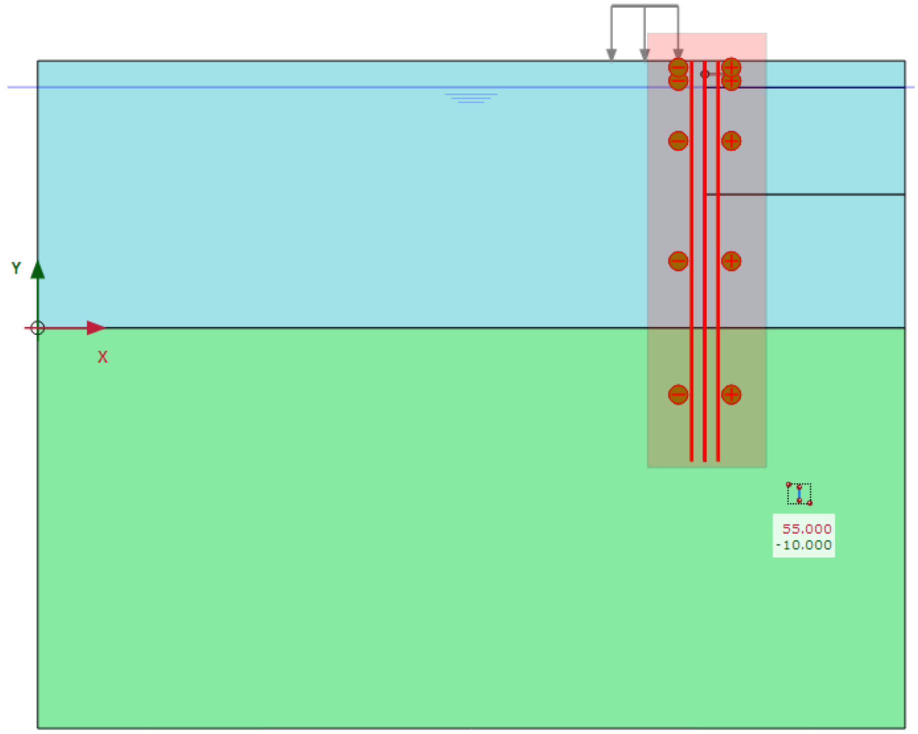



*Figure 49: Selected multiple plate elements in the model*

When the wall is unselected it can be seen that wall is now visible in the color that is specified in the material dataset.

5. Right-click the distributed load to activate it and select the **Activate option** from the appearing menu. The load has been defined in the **Structures mode** as -5 kN/m/m. The value can be checked in the **Selection explorer**.
6. Make sure all the interfaces in the model are active.

**Note:** The selection of an interface is done by right-clicking the corresponding geometry line and subsequently selecting the corresponding interface (positive or negative) from the context menu.

### 3.6.3 Phase 2: First excavation stage

1. Click the **Add phase** button in the **Phases explorer**.
   A new calculation phase appears in the **Phases explorer**. Note that the program automatically presumes that the current phase should start from the previous one and that the same objects are active.

   **Note:** To copy the settings of the parent phase, select the phase in the **Phases** explorer and then click the **Add phase** button. Note that the settings of the parent phase are not copied when it is specified by selecting it in the **Start from phase** drop-down menu in the **Phases** window.

2. The default settings are valid for this phase. In the **Staged construction mode** all the structure elements except the fixed-end anchor are active.
3. In the drawing area right-click the top right cluster and select the **Deactivate** option in the appearing menu.

The model for the first excavation phase is shown in Figure 50 (on page 65):

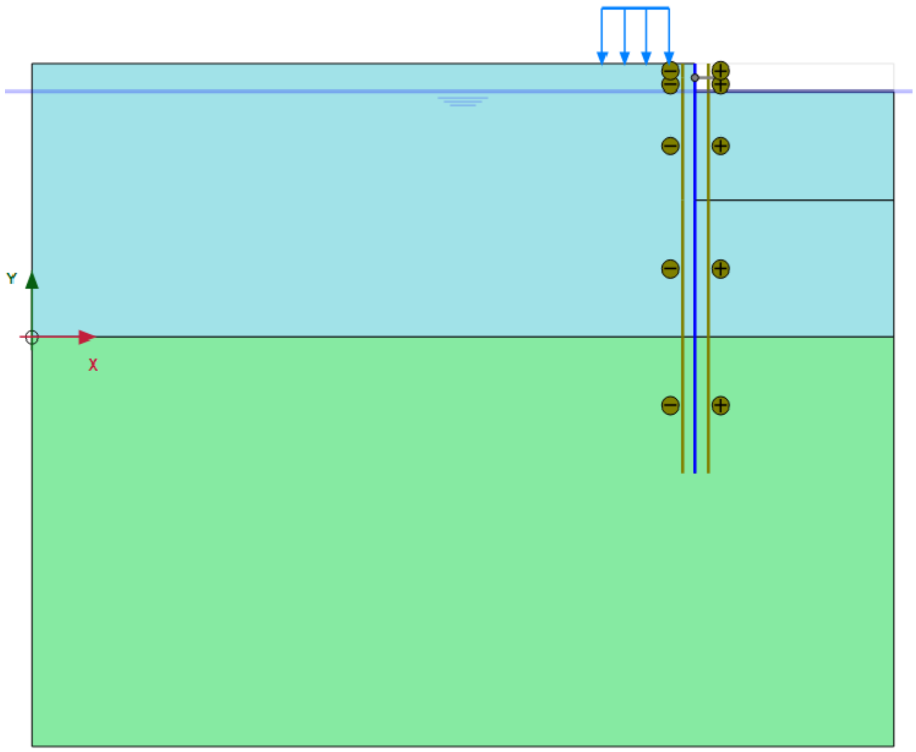



*Figure 50: Model for the first excavation phase*

### 3.6.4 Phase 3: Installation of a strut

1. Click the **Add phase** button in the **Phases explorer**.
2. Activate the strut.
   The strut turns black to indicate it is active.

### 3.6.5 Phase 4: Second (submerged) excavation stage

1. Click the **Add phase** button to add a new phase.

2. Deactivate the second cluster from the top on the right side of the mesh. It should be the topmost active cluster.

> **Note:** Note that in PLAXIS 2D the pore pressures are not automatically deactivated when deactivating a soil cluster. Hence, in this case, the water remains in the excavated area and a submerged excavation is simulated.

The resulting model is shown in Figure 51 (on page 66) :

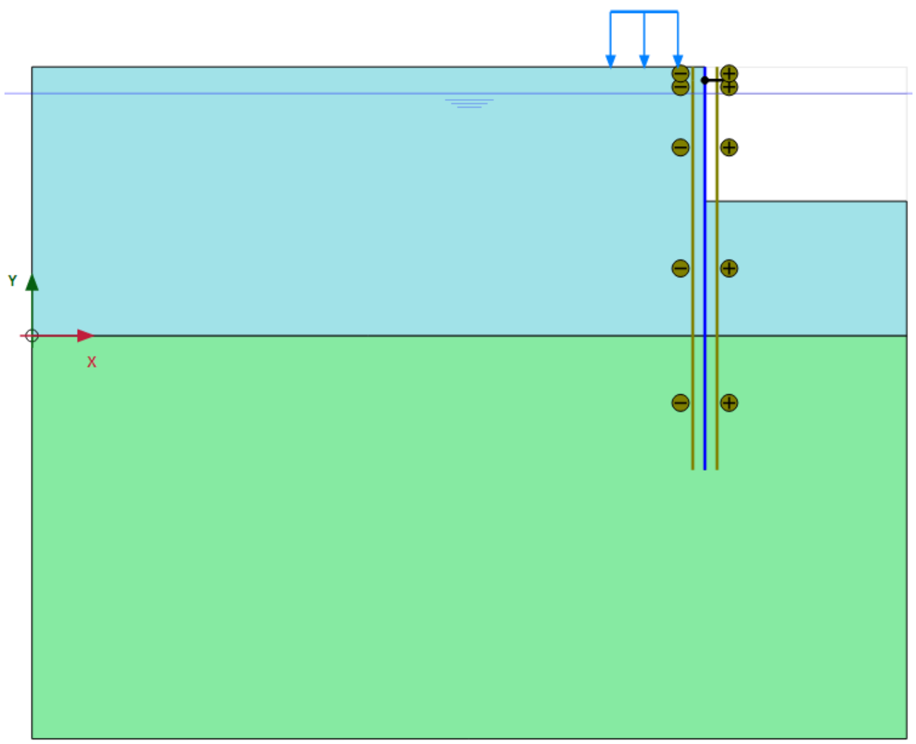



*Figure 51: Model for the second excavation phase*

## 3.6.6 Phase 5: Third excavation stage

In the final calculation stage the excavation of the last clay layer inside the pit is simulated.

1. Click the **Add phase** button to add a new phase.
2. Deactivate the third cluster from the top on the right hand side of the mesh.
   The model for the phase is shown in Figure 52 (on page 67):

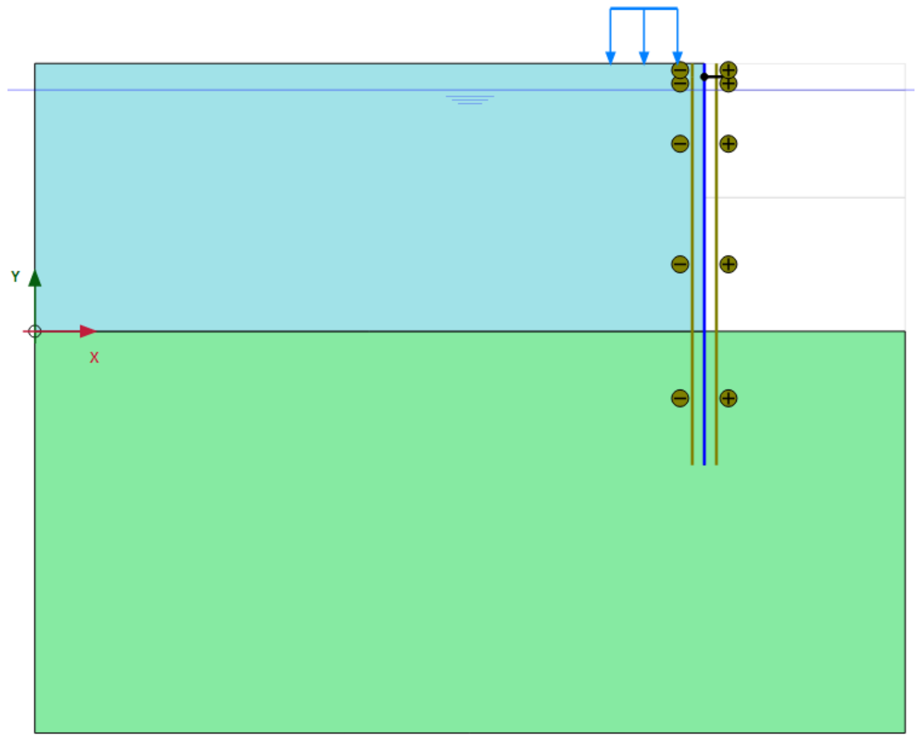



*Figure 52: Model for the third excavation phase*

The calculation definition is now complete.

## 3.6.7 Execute the calculation

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow the steps given below.

1. Click the **Select points for curves** button in the side toolbar.
   The connectivity plot is displayed in the Output program and the **Select points** window is activated.
2. Select some nodes on the wall at points where large deflections can be expected (e.g. 50 10). The nodes located near that specific location are listed. Select the convenient one by checking the box in front of it in the list.
3. Click on the **Update** tab at the top left to save the selected points, close the Output program and go back to the **Input** program.
4. Click the **Calculate** button to calculate the project.

During a **Staged construction** calculation phase, a multiplier called ΣMstage is increased from 0.0 to 1.0. This parameter is displayed in the calculation info window. As soon as ΣMstage has reached the value 1.0, the construction stage is completed and the calculation phase is finished. If a **Staged construction** calculation finishes while ΣMstage is smaller than 1.0, the program will give a warning message. The most likely reason for not finishing a construction stage is that a failure mechanism has occurred, but there can be other causes as well. See the Reference Manual for more information about **Staged construction**.

## 3.7 View the calculation results

In addition to the displacements and the stresses in the soil, the Output program can be used to view the forces in structural objects. To examine the results of this project, follow these steps:

### 3.7.1 Displacements and stresses

To view displacements and stresses, follow these steps:

1. Click the final calculation phase in the **Calculations** window.
2. Click the **View calculation results** button  on the toolbar.
   As a result, the Output program is started, showing the deformed mesh (scaled up) as shown in Figure 53 (on page 68) at the end of the selected calculation phase, with an indication of the maximum displacement:

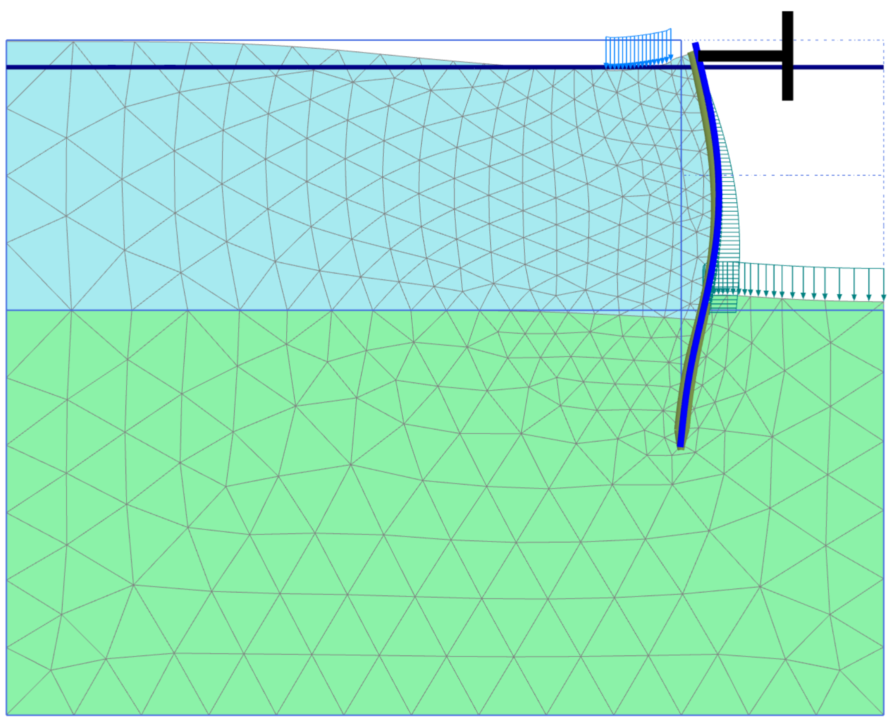


*Figure 53: Deformed mesh after the third excavation stage*

**Note:**

In the Output program, the display of the loads, fixities and prescribed displacements applied in the model can be toggled on/off by clicking the corresponding options in the **Geometry** menu.

3. Select the menu **Deformations** > **Incremental displacements** > **|Δu|**.
The plot shows colour shadings of the displacement increments, which indicates the forming of a mechanism of soil movement behind the wall.
4. Click the **Arrows** button in the toolbar.
The plot shows the displacement increments of all nodes as arrows. The length of the arrows indicates the relative magnitude.
5. Select the menu **Stresses** > **Principal effective stresses** > **Effective principal stresses**.

The plot in Figure 54 (on page 69) shows the effective principal stresses at the three middle stress points of each soil element with an indication of their direction and their relative magnitude. Note that the **Center principal stresses** button is selected in the toolbar. The orientation of the principal stresses indicates a large passive zone under the bottom of the excavation and a small passive zone behind the strut.

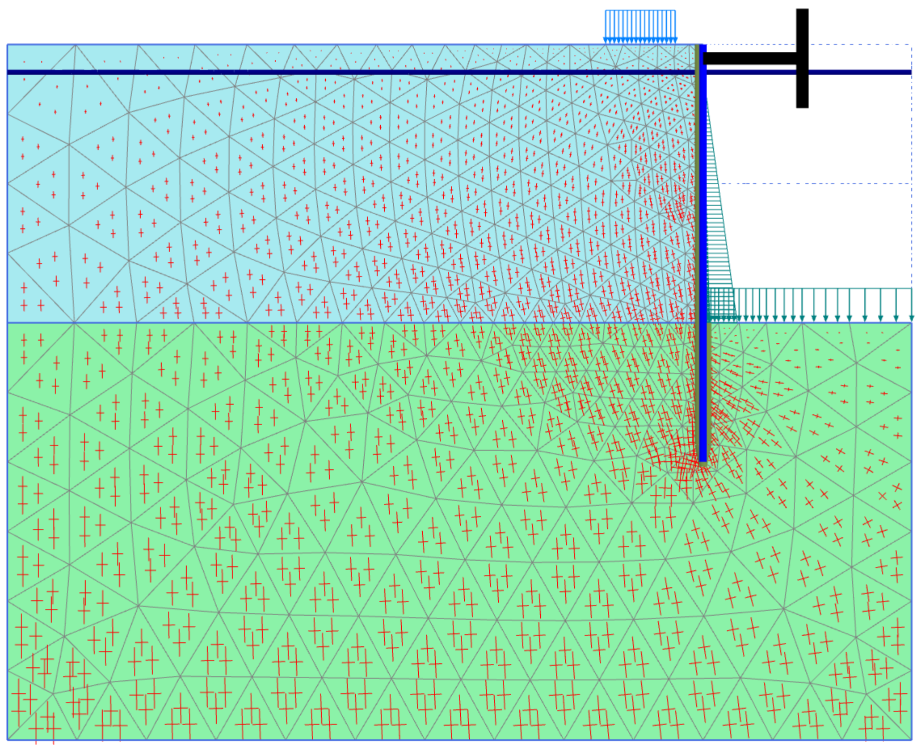


*Figure 54: Principal stresses after excavation*

## 3.7.2 Shear forces and bending moments

To plot the shear forces and bending moments in the wall follow the steps given below.

1. Double-click the wall.

A new window is opened showing the axial force.

2. Select the menu **Forces** > **bending moment M.**
   The bending moment in the wall is displayed in Figure 55 (on page 70) with an indication of the maximum moment:

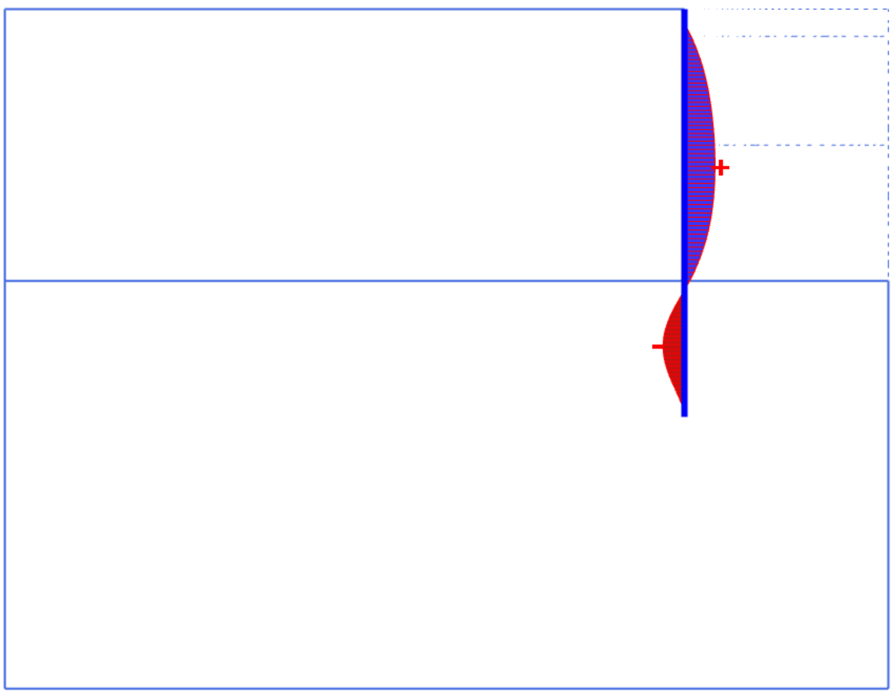


*Figure 55: Bending moments in the wall*

3. Select **Shear forces Q** from the **Forces** menu.

   The plot now shows the shear forces in the wall.

> **Note:** The **Window** menu may be used to switch between the window with the forces in the wall and the stresses in the full geometry. This menu may also be used to **Tile** or **Cascade** the two windows, which is a common option in a Windows environment.

4. Select the first window (showing the effective stresses in the full geometry) from the **Window** menu. Double-click the strut.
   The strut force (in kN) is shown in the displayed table.
5. Click the **Curves manager** button on the toolbar.
   As a result, the **Curves manager** window pops up.
6. Click **New** to create a new chart.
   The **Curve generation** window pops up.
7. For the x-axis select the point A from the drop-down menu. In the tree select **Deformations - Total displacements - |u|**.
8. For the y-axis keep the **Project** option in the drop-down menu. In the tree select **Multiplier - ΣMstage**.
9. Click **OK** to accept the input and generate the load-displacement curve.
   As a result the curve is plotted as shown in Figure 56 (on page 71) :

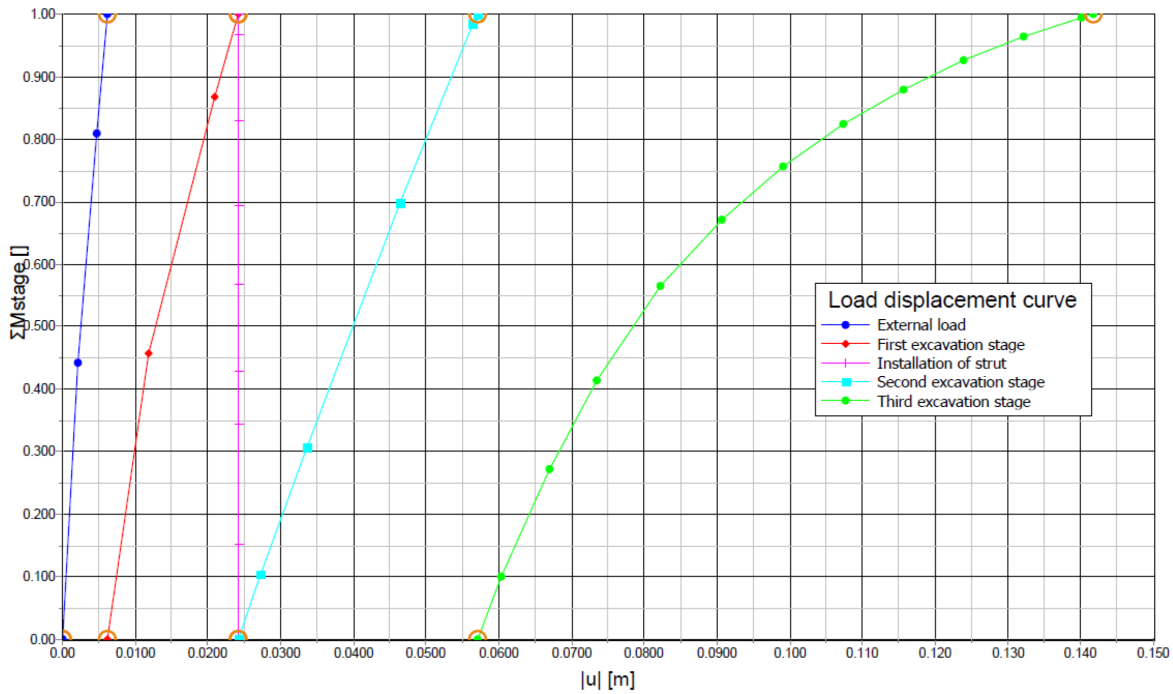



*Figure 56: Load-displacement curve of deflection of wall*

The curve shows the construction stages. For each stage, the parameter ΣMstage changes from 0.0 to 1.0. The decreasing slope of the curve in the last stage indicates that the amount of plastic deformation is increasing. The results of the calculation indicate, however, that the excavation remains stable at the end of construction.

# 4 Settlements due to tunnel construction [GSE]

In this tutorial the construction of a shield tunnel in medium soft soil and the influence on a pile foundation is considered. A shield tunnel is constructed by excavating soil at the front of a tunnel boring machine (TBM) and installing a tunnel lining behind it. In this procedure the soil is generally over-excavated, which means that the cross sectional area occupied by the final tunnel lining is always less than the excavated soil area. Although measures are taken to fill up this gap, one cannot avoid stress re-distributions and deformations in the soil as a result of the tunnel construction process. To avoid damage to existing buildings or foundations on the soil above, it is necessary to predict these effects and to take proper measures. Such an analysis can be performed by means of the finite element method. This tutorial shows an example of such an analysis.

**Objectives**

- Modelling of the tunnel boring process
- Modelling undrained behaviour using the **Undrained (B)** option

**Geometry**

The tunnel considered in this tutorial has a diameter of 5 m and is located at an average depth of 17 m. The geometry of the tunnel is shown in Figure 57 (on page 73) .

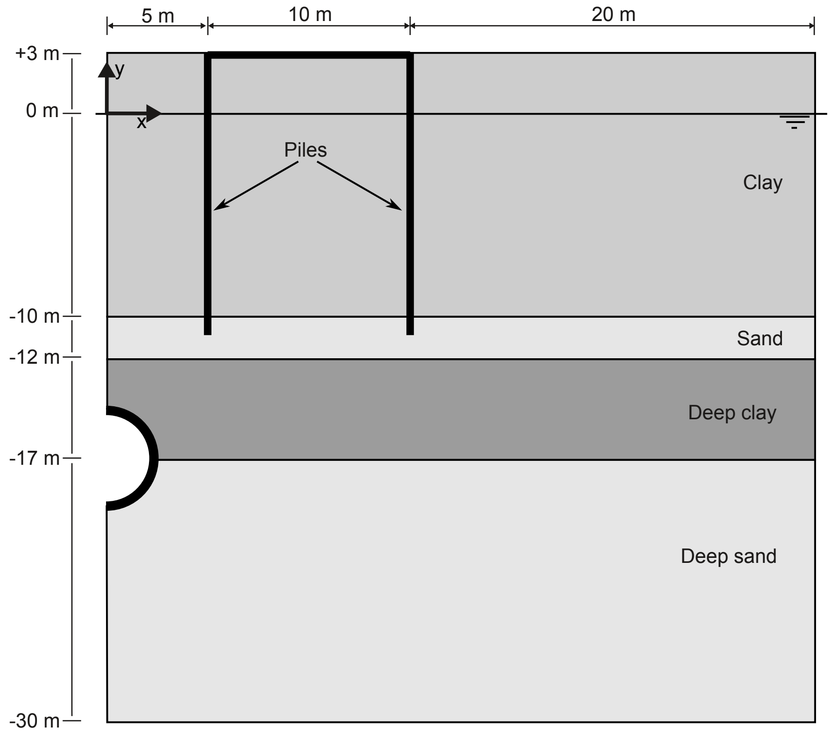



*Figure 57: Geometry of the tunnel project with an indication of the soil layers*

## 4.1 Create new project

To create the new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** 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-Noded**).
4. Set the model **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 35 m, $y_{min}$ = -30 m and $y_{max}$ = 3 m.
5. Keep the default values for units and constants and press **OK** to close the **Project properties** window.

## 4.2 Define the soil stratigraphy

The soil profile indicates four distinct layers: The upper 13 m consists of soft clay type soil with stiffness that increases approximately linearly with depth. Under the clay layer there is a 2 m thick fine sand layer. This layer is used as a foundation layer for old wooden piles on which traditional brickwork houses were built. The pile foundation of such a building is modelled next to the tunnel. Displacements of these piles may cause damage to the building, which is highly undesirable. Below the sand layer there is a 5 m thick deep loamy clay layer.

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.

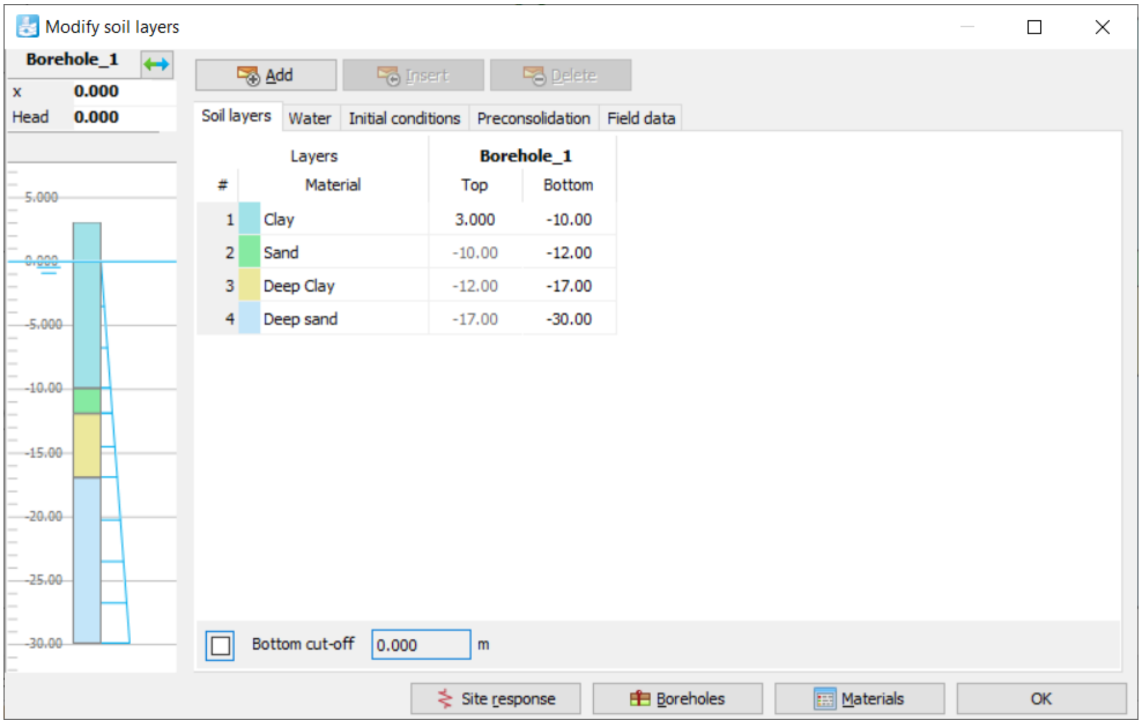



*Figure 58: The soil stratigraphy in the **Modify soil layers** window*

2. Create the soil stratigraphy as shown in Figure 58 (on page 74).
3. Keep the **Head** in the borehole to 0 m.

## 4.2.1 Create and assign material data sets

Four data sets need to be created for the clay and sand layers.

For the upper clay layer the stiffness and shear strength increase with depth. Therefore values for $E'_{inc}$ and $s_{u,inc}$ are entered in the **Advanced** subtree. The values of $E'_{ref}$ and $s_{u,ref}$ become the reference values at the reference level $y_{ref}$. Below $y_{ref}$ the actual values of E' and $s_u$ increase with depth according to:

$$E'(y) = E_{ref}' + E_{inc}'(y_{ref} - y)$$

$$s_u(y) = s_{u,ref} + s_{u,inc}(y_{ref} - y)$$

The data sets of the two lower soil layers include appropriate parameters for the tunnel interfaces. In the other data sets the interface properties just remain at their default values. Enter four data sets with the properties as listed in Table 7 (on page 75) and Table 8 (on page 76) , then assign them to the corresponding clusters in the geometry model.

The layers have the following properties:

**Table 7: Material properties of the clay layers**

| Property | Name | Clay | Deep clay | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | Mohr-Coulomb | Mohr-Coulomb | - |
| Drainage type | - | Undrained (B) | Undrained (B) | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 15 | 16 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 18 | 18.5 | $kN/m^3$ |
| **Mechanical** | | | | |
| Young's modulus at reference level | $E'_{ref}$ | $3.4 \cdot 10^3$ | $9.0 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu (nu)$ | 0.33 | 0.33 | - |
| Young's modulus increment | $E'_{inc}$ | 400 | 600 | $kN/m^3$ |
| Reference level | $y_{ref}$ | 3.0 | -12 | m |
| Undrained shear strength at reference level | $s_{u,ref}$ | 5 | 40 | $kN/m^2$ |
| Undrained shear strength inc. | $s_{u,inc}$ | 2 | 3 | $kN/m^3$ |
| **Groundwater** | | | | |
| Data set | - | Standard | Standard | - |
| Soil type | - | Coarse | Coarse | - |
| Use defaults | - | None | None | - |
| Permeability in horizontal direction | $k_x$ | $0.1 \cdot 10^{-3}$ | 0.01 | m/day |
| Permeability in vertical direction | $k_y$ | $0.1 \cdot 10^{-3}$ | 0.01 | m/day |
| **Interfaces** | | | | |
| Strength determination | - | Rigid | Manual | - |
| Interface reduction factor | $R_{inter}$ | 1.0 | 0.7 | - |

| **Initial** | | | | |
|---|---|---|---|---|
| $K_0$ determination | - | Manual | Manual | - |
| Lateral earth pressure coefficient | $K_{0,x}$ | 0.6 | 0.7 | - |

**Table 8: Material properties of the sand layers**

| Property | Name | Sand | Deep sand | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | HS small | HS small | - |
| Drainage type | - | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16.5 | 17 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | 21 | kN/m$^3$ |
| **Mechanical** | | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}^{ref}$ | $25 \cdot 10^3$ | $42 \cdot 10^3$ | kN/m$^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}^{ref}$ | $25 \cdot 10^3$ | $42 \cdot 10^3$ | kN/m$^2$ |
| Unloading / reloading stiffness | $E_{ur}^{ref}$ | $75 \cdot 10^3$ | $126 \cdot 10^3$ | kN/m$^2$ |
| Poisson's ratio | $\nu_{ur}$ | 0.2 | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | 0.5 | - |
| Shear modulus at very small strains | $G_0^{ref}$ | $80 \cdot 10^3$ | $110 \cdot 10^3$ | kN/m$^2$ |
| Shear strain at which $G_s$ = 0.722 $G_0$ | $\gamma_{0.7}$ | $0.2 \cdot 10^{-3}$ | $0.13 \cdot 10^{-3}$ | - |
| Cohesion | $c'_{ref}$ | 0 | 0 | kN/m$^2$ |
| Friction angle | $\varphi'$ | 31 | 35 | ° |

| **Mechanical** | | | | |
|---|---|---|---|---|
| Dilatancy angle | $\psi$ | 1 | 5 | ° |
| **Groundwater** | | | | |
| Classification type | - | Standard | Standard | - |
| Soil class | - | Coarse | Coarse | - |
| Use defaults | - | None | None | - |
| Permeability in horizontal direction | $k_x$ | 1.0 | 0.5 | m/day |
| Permeability in vertical direction | $k_y$ | 1.0 | 0.5 | m/day |
| **Interfaces** | | | | |
| Strength determination | - | Rigid | Manual | - |
| Interface reduction factor | $R_{inter}$ | 1.0 | 0.7 | - |
| **Initial** | | | | |
| $K_0$ determination | - | Automatic | Automatic | - |
| Pre-overburden pressure | $POP$ | 0.0 | 0.0 | - |
| Over-consolidation ratio | $OCR$ | 1.0 | 1.0 | - |

To create the material sets, follow these steps:

1. Click the **Materials** button  in the **Modify soil layers** window and create the data sets.

## 4.3 Define the structural elements

The tunnel and the building are defined as structural elements.

### 4.3.1 Define the tunnel

The tunnel considered here is the right half of a circular tunnel. After generating the basic geometry, follow these steps to design the circular tunnel:

1. In the **Structures mode** click the **Create tunnel** button in the side toolbar and click at (0 -17) in the drawing area.
   The **Tunnel designer** window pops up displaying the **General** tabsheet of the **Cross section mode** .
2. Select the **Circular** option in the **Shape type** drop-down menu.
3. Select the **Define right half** option in the **Whole or half tunnel** drop-down menu.
4. In the **Offset to begin point** group set **Axis 2** to -2.5. No change is required for the orientation axes.
5. Click the **Segments** tab to proceed to the corresponding tabsheet.
   A segment is automatically created. A new box is shown under the segment list where the properties of the segment can be defined.
6. In the **Segment** box set **Radius** to 2.5 m.
   The generated segment is shown in Figure 59 (on page 78) .

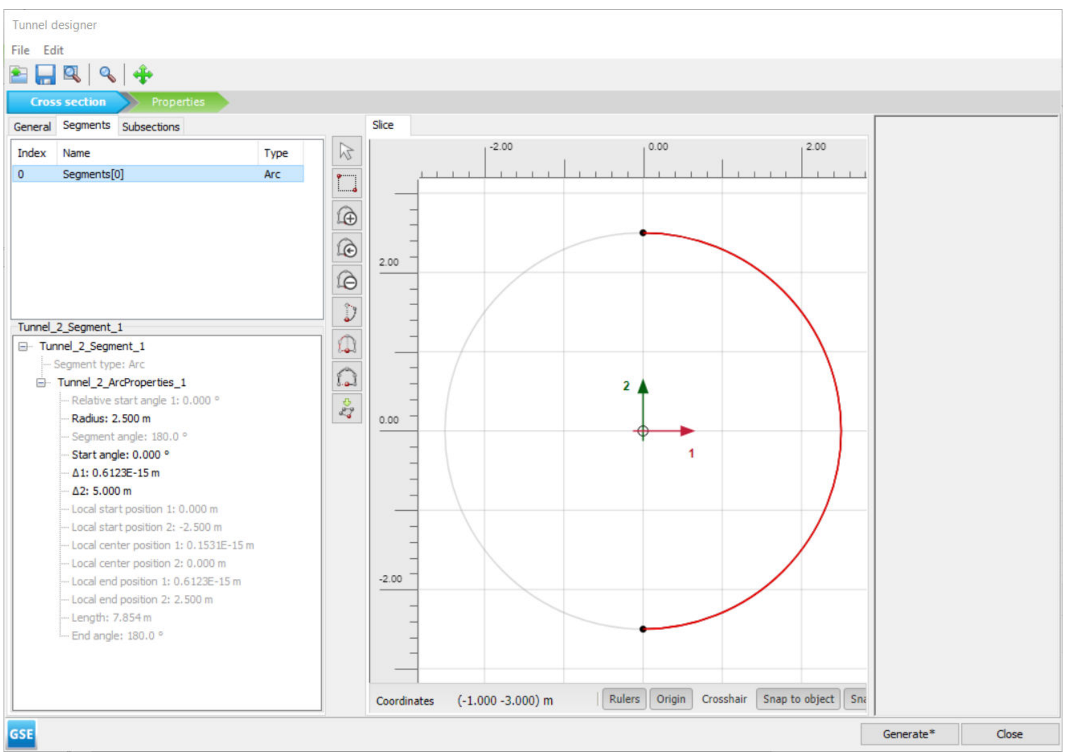



*Figure 59: The geometry of the tunnel segment*

**Note:** In the tunnel as considered here, the segments do not have a specific meaning as the tunnel lining is homogeneous and the tunnel will be constructed at once. In general, the meaning of segments becomes significant when:

- It is desired to excavate or construct the tunnel (lining) in different stages.
- Different tunnel segments have different lining properties.
- One would consider hinge connections in the lining (hinges can be added after the design of the tunnel, in the general drawing area).
- The tunnel shape is composed of arcs with different radii (for example NATM tunnels).

7. Click the **Properties** tab to proceed to the corresponding mode.
8. Right-click on the segment in the display area and select the menu **Create** > **Create plate** option in the appearing menu.
9. In Tunnel Designer go to the Material property in the **Selection explorer** and click the plus button [+] to create a new material dataset. Specify the material parameters for the TBM according to the Table 9 (on page 79):

**Table 9: Material properties of the plates**

| Parameter | Name | TBM | Lining | Building | Unit |
|---|---|---|---|---|---|
| **General** | | | | | |
| Material type | - | Elastic | Elastic | Elastic | - |
| Unit Weight | $w$ | 17.7 | 8.4 | 25 | kN/m/m |
| Prevent punching | - | No | No | No | - |
| **Mechanical** | | | | | |
| Isotropic | - | Yes | Yes | Yes | - |
| Axial stiffness | $EA_1$ | $63 \cdot 10^6$ | $14 \cdot 10^6$ | $1 \cdot 10^{10}$ | kN/m |
| Bending stiffness | $EI$ | $472.5 \cdot 10^3$ | $143 \cdot 10^3$ | $1 \cdot 10^{10}$ | $kNm^2/m$ |
| Poisson's ratio | $\nu$ | 0 | 0.15 | 0 | - |

**Note:** A tunnel lining consists of curved plates (shells). The lining properties can be specified in the material database for plates. Similarly, a tunnel interface is nothing more than a curved interface.

10. Right-click the segment in the display area and select the **Create negative interface** option in the appearing menu.
11. Right-click the segment in the display area and select the **Create line contraction** option in the appearing menu. In the polycurve properties specify a value of 0.5% for $C_{ref}$. The tunnel model is shown in Figure 60 (on page 80) .

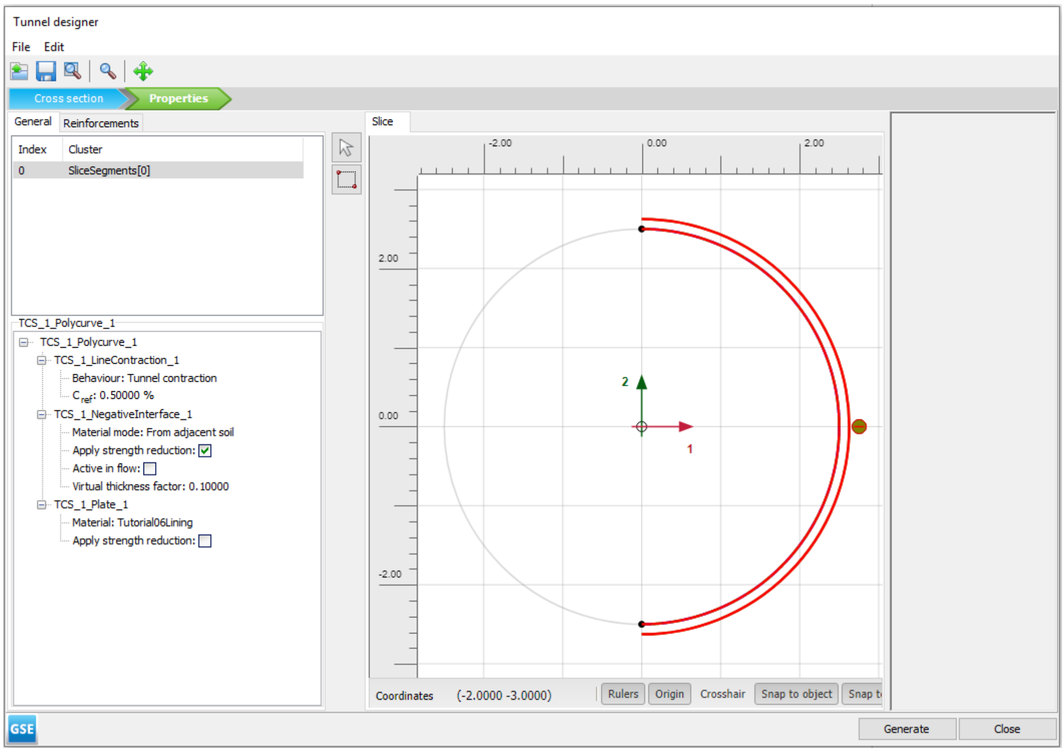



*Figure 60: Tunnel model in the **Properties** tab*

**Note:**

- A $C_{ref}$ value of 0.5% corresponds to a volume loss of 0.5% of the tunnel volume. The actual strain that is applied to the line is half the applied contraction. Hence, the resulting liner contraction is 0.25%.
- The entered value of contraction is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

12. Click on **Generate** to include the defined tunnel in the model.
13. Close the **Tunnel designer** window.

## 4.3.2 Define building

The building itself will be represented by a stiff plate founded on piles.

1. From the side bar, select **Create line** > **Create plate** and draw a plate from (5 3) to (15 3), representing the building.
2. Create a material set for the building according to Table 9 (on page 79) and assign it to the plate. Do not forget to also create the material set for the tunnel lining.
3. From the side bar, select **Create line** > **Create embedded beam** and draw two piles (embedded beam rows) from (5 3) to (5 -11) and from (15 3) to (15 -11).

4. Create a material set for the foundation piles according to Table 10 (on page 81) and assign it to the foundation piles.

**Table 10: Material properties of piles**

| Parameter | Name | Foundation piles | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| Unit weight | $\gamma$ | 7.0 | $kN/m^3$ |
| **Mechanical** | | | |
| Pile spacing | $L_{spacing}$ | 3.0 | m |
| Cross section type | - | Predefined | - |
| Predefined cross section type | - | Solid circular beam | - |
| Diameter | - | 0.25 | m |
| Stiffness | $E$ | $10 \cdot 10^6$ | $kN/m^2$ |
| Axial skin resistance | Axial skin resistance | Linear | - |
| | $T_{skin, start, max}$ | 1.0 | kN/m |
| | $T_{skin, end, max}$ | 100.0 | kN/m |
| Lateral resistance | Lateral resistance | Unlimited | - |
| Base resistance | $F_{max}$ | 100.0 | kN |
| Interface stiffness factors | Default values | Yes | - |

**Note:** With the **Default fixities** used, a plate that extends to a geometry boundary that is fixed in at least one direction obtains fixed rotations, whereas a plate that extends to a free boundary obtains a free rotation.

## 4.4 Generate the mesh

The default global coarseness parameter (**Medium**) can be accepted in this case. Note that the structural elements (plate and embedded beams) are internally automatically refined by a factor of 0.25.

1. Proceed to the **Mesh mode**.

2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 61 (on page 82) .

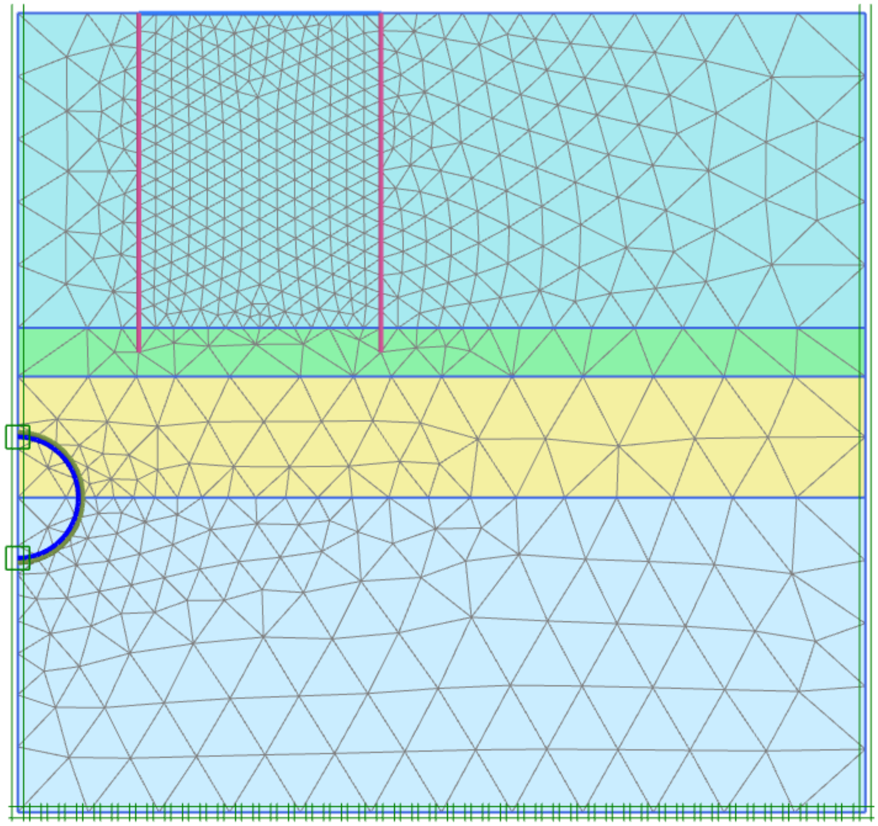


*Figure 61: The generated mesh*

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

## 4.5 Define and perform the calculation

To simulate the construction of the tunnel it is clear that a staged construction calculation is needed.

### 4.5.1 Initial phase

1. Proceed to the **Staged construction** mode to proceed with the definition of the calculation phases.

2. The initial phase has already been introduced. Keep its calculation type as **K0 procedure**. The water pressures can be generated on the basis of a general phreatic level at a level of y = 0.0 m as already defined in the borehole. Make sure that the building, foundation piles and tunnel lining are deactivated.

### 4.5.2 Phase 1: Building

The first calculation phase is used to activate the building.

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window rename the Phase ID as `Building`.
3. In the **Deformation control parameters** subtree select the **Ignore undr. behaviour (A,B)** option. The default values of the remaining parameters are valid for this phase.
4. In the drawing area activate the plate and the foundation piles.

### 4.5.3 Phase 2: TBM

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window select the **Reset displacements to zero** option in the **Deformation control parameters** subtree.
3. In **Staged construction** multi-select the clusters inside the tunnel. In the **Selection explorer** deactivate the two soil clusters and set the **Water conditions** to **Dry**.
4. Activate the circular plate that in this phase represents the Tunnel Boring Machine (and thus has the TBM material set) as well as the negative interfaces. Note that contraction is not active in this phase.

### 4.5.4 Phase 3: TBM conicity

1. Click the **Add phase** button to create a new phase.
2. Multi-select the plates. In the **Selection explorer** activate the contraction.

**Note:**

- The contraction represents the effects of the cone shape of the TBM (cutter head has larger diameter than the tail).
- The contraction of the tunnel lining by itself does not introduce forces in the tunnel lining. Eventual changes in lining forces as a result of the contraction procedure are due to stress redistributions in the surrounding soil or to changing external forces.

### 4.5.5 Phase 4: Tail void grouting

At the tail of the tunnel boring machine (TBM), grout is injected to fill up the gap between the TBM and the final tunnel lining. The grouting process is simulated by applying a pressure on the surrounding soil.

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** mode deactivate the TBM (plates, negative interfaces and contraction).
3. Multi-select the clusters inside the tunnel. In the **Selection explorer** activate **WaterConditions**.
4. Select the **User-defined** option in the **Condition** drop-down menu and set $p_{ref}$ to -230 kN/m$^2$. The pressure distribution in the tunnel is constant.

### 4.5.6 Phase 5: Lining installation

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** set the clusters inside the tunnel to **Dry**.
3. Activate the plates and the negative interfaces of the tunnel.
4. Since the plates now represent the final lining of the tunnel, assign the *Lining* material set to the plate elements.

### 4.5.7 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select some characteristic points for load-displacement curves (for example the corner point at the ground surface above the tunnel node 6130 and the corner points of the building node 9647 ).
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button .

## 4.6 Results

After the calculation, select the last calculation phase and click the **View calculation results** button. The **Output** program is started, showing the deformed meshes at the end of the calculation phases which is shown in Figure 62 (on page 85):

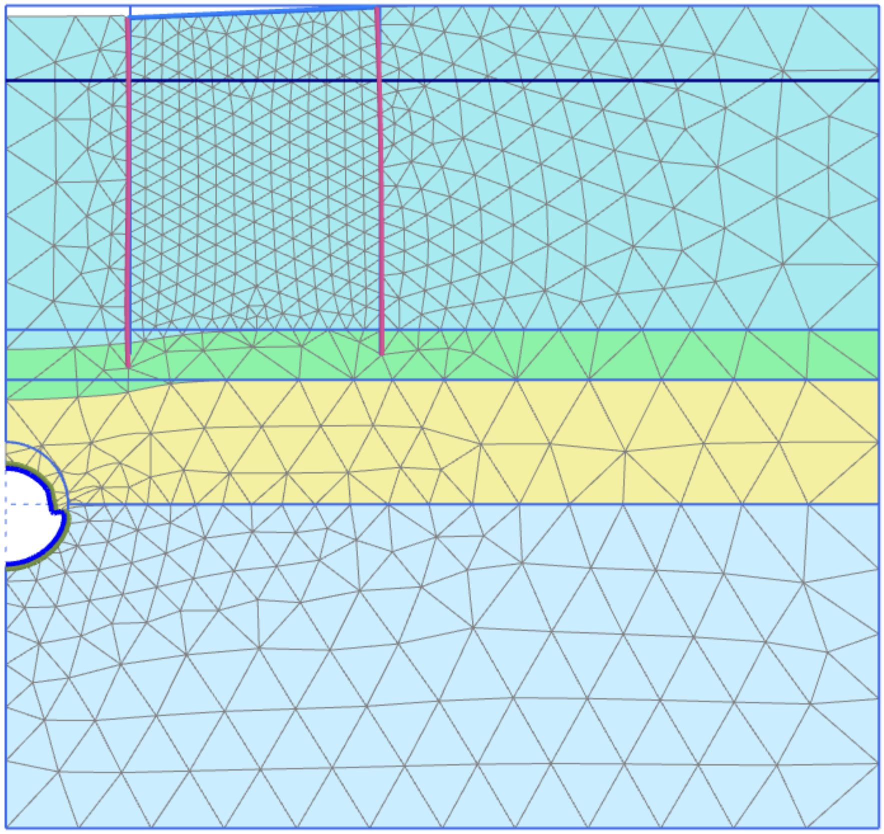


*Figure 62: Deformed mesh after construction of the tunnel (Phase 5; scaled up manually 20 times)*

As a result of the second calculation phase (removing soil and water out of the tunnel) there is some settlement of the soil surface and the tunnel lining shows some deformation. In this phase the axial force in the lining is the maximum axial force that will be reached. The lining forces can be viewed by double clicking the lining and selecting force related options from the **Force** menu. The plots of the axial forces and bending moment are scaled by factors of $5 \cdot 10^{-3}$ and 0.2 respectively which is shown in Figure 63 (on page 86).

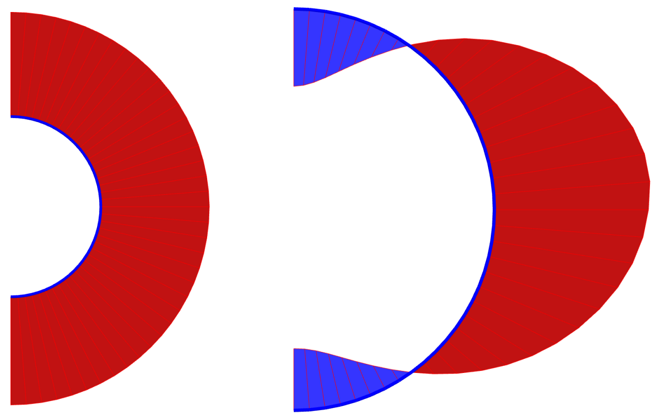


*Figure 63: Axial forces and Bending moments in the lining after the second phase*

The plot of effective stresses as shown in Figure 64 (on page 86), shows that arching occurs around the tunnel. This arching reduces the stresses acting on the tunnel lining. As a result, the axial force in the final phase is lower than that after the second calculation phase.

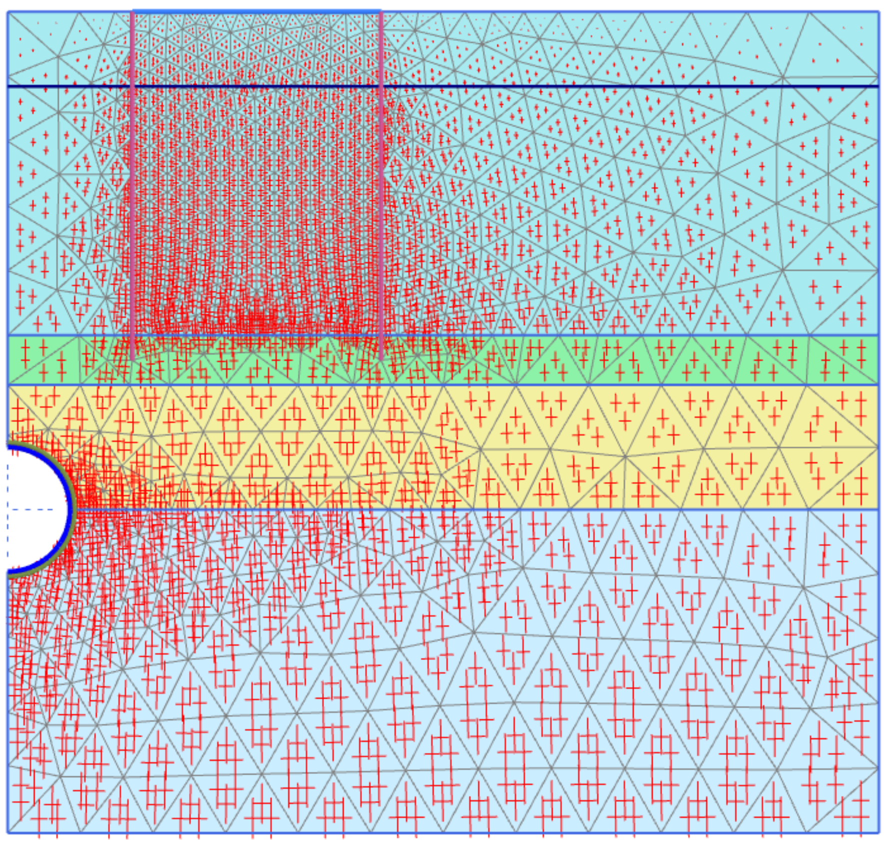


*Figure 64: Effective principal stresses after the construction of the tunnel(Phase 2 TBM)*

To display the tilt of the structure at the end of lining installation phase:

1. Click the **Distance measurement** button in the side toolbar.
2. Click the node located at the left corner of the structure (5 3).
3. Click the node located at the right corner of the structure (15 3).

The **Distance measurements information** window is displayed in Figure 65 (on page 87) ,where the resulting tilt of the structure is given:

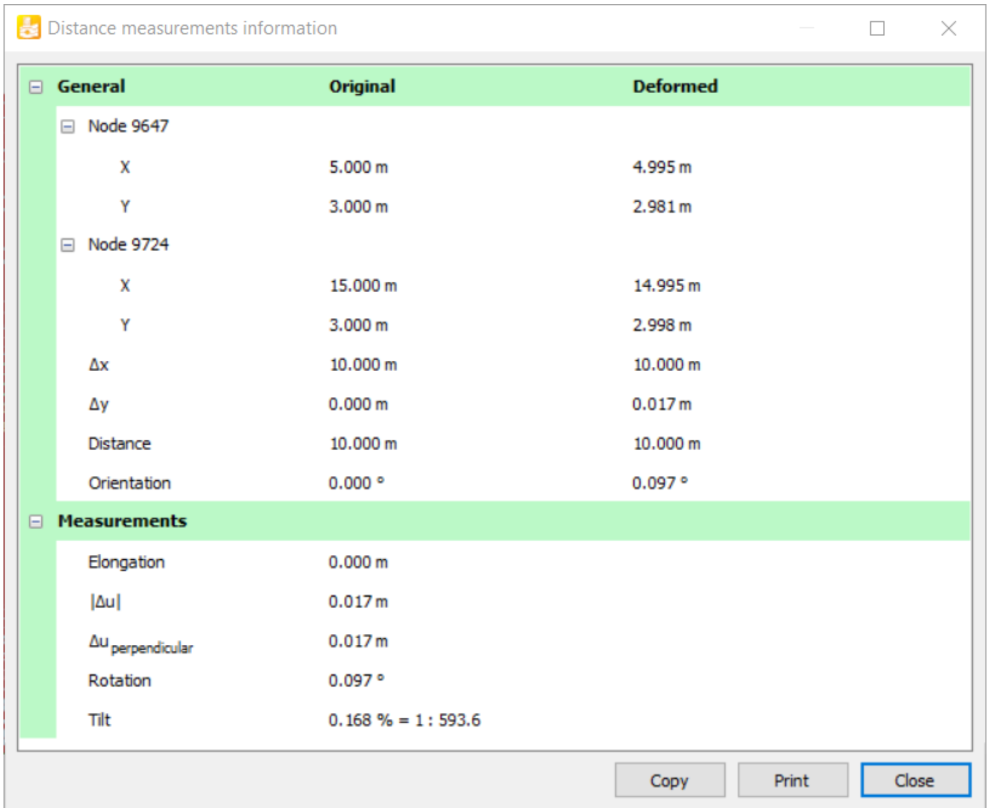



*Figure 65: Distance measurement information window*

# 5 Excavation of an NATM tunnel [GSE]

This tutorial illustrates the use of PLAXIS 2D for the analysis of the construction of a NATM tunnel. The NATM is a technique in which ground exposed by excavation is stabilised with shotcrete to form a temporary lining.

**Objectives**

- Modelling the construction of an NATM tunnel using the **Deconfinement** method.
- Using **Gravity loading** to generate initial stresses.

**Geometry**

The geometry of the tunnel is shown in Figure 66 (on page 88) :

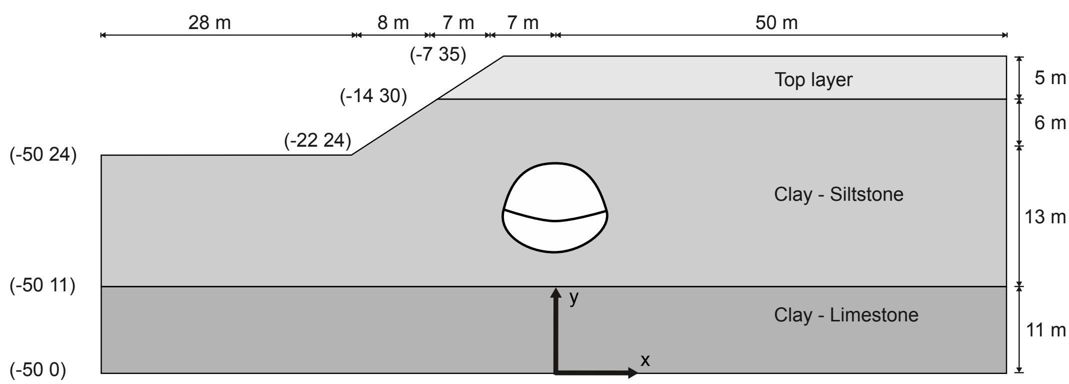



*Figure 66: Geometry of the project*

## 5.1 Create a new project

To create a new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet make sure that **Model** is set to **Plane strain** and that **Elements** is set to **15-Noded**.
4. Define the limits for the soil contour as $x_{min}$ = -50 m, $x_{max}$ = 50 m, $y_{min}$ = 0 m and $y_{max}$ = 35 m.

## 5.2 Define the soil stratigraphy

The basic stratigraphy will be created using the **Borehole** feature. In the model 11 m of the Clay-limestone layer is considered. The bottom of this layer is considered as reference in y direction ($y_{min}$ = 0).

To define the soil stratigraphy:

1. Click the **Create borehole** button and create the first borehole at x = -22 m.
2. In the **Modify soil layers** window create three soil layers.
   a. The layer number 1 has a depth equal to zero in Borehole_1. Assign 24 to **Top** and **Bottom**.
   b. The layer number 2 lies from **Top** = 24 to **Bottom** = 11.
   c. The layer number 3 lies from **Top** = 11 to **Bottom** = 0.
3. Click the **Boreholes** button at the bottom of the **Modify soil layers** window.
4. In the appearing menu select the **Add** option.
   The **Add borehole** window pops up.
5. Specify the location of the second borehole (x = -14).
6. Note that the soil layers are available for Borehole_2.
   a. The layer number 1 has a depth equal to zero in Borehole_2. However as the depth of layer 2 is higher, assign 30.00 to **Top** and **Bottom** of the layer 1.
   b. The layer number 2 lies from **Top** = 30 to **Bottom** = 11.
   c. The layer number 3 lies from **Top** = 11 to **Bottom** = 0.
7. Create a new borehole (Borehole_3) at x = -7.
8. In Borehole_3:
   a. The layer number 1 has a non-zero thickness and lies from **Top** = 35 to **Bottom** = 30.
   b. The layer number 2 lies from **Top** = 30 to **Bottom** = 11.
   c. The layer number 3 lies from **Top** = 11 to **Bottom** = 0.
9. In all the boreholes the water level is located at y = 0 m.
10. Specify the soil layer distribution as shown in Figure 67 (on page 90).

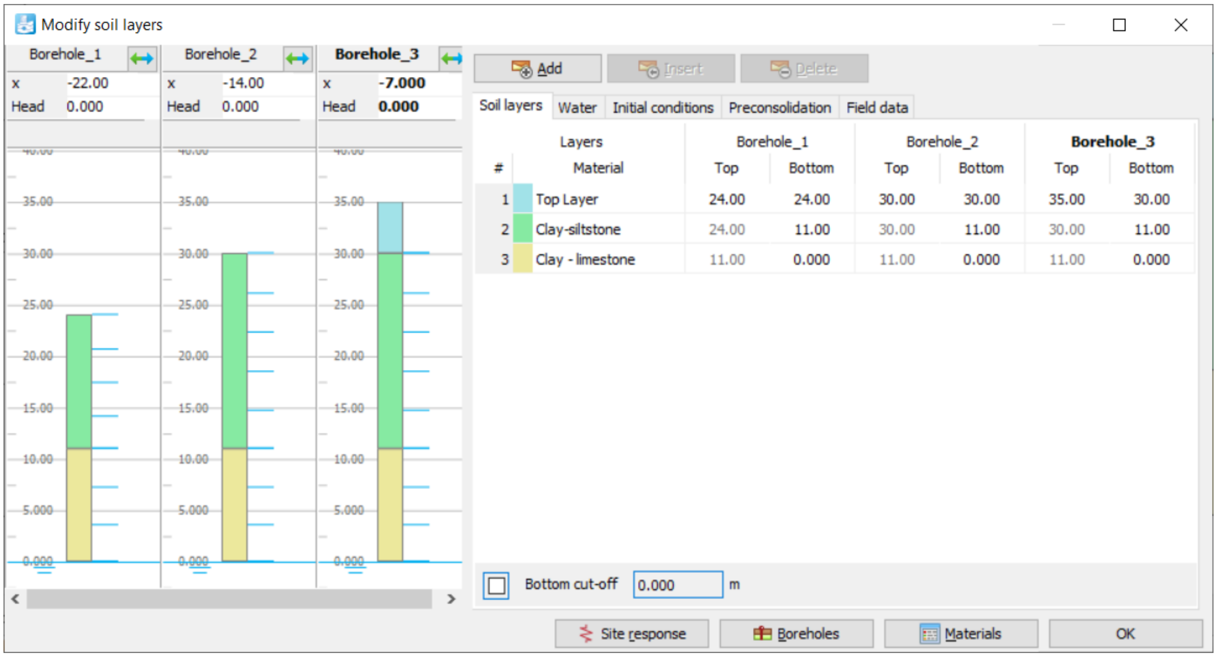



*Figure 67: Soil layer distribution*

## 5.3 Create and assign material data sets

Three material sets need to be created for the soil layers.

Note that the layering of the model left from the first borehole is based on Borehole_1 and the layering right from the last borehole is based on Borehole_3. Hence, no borehole is needed at x = -50 m or x = 50 m.

The layers have the following properties as shown in Table 11 (on page 90):

**Table 11: Material properties of the soil layer**

| Property | Name | Top layer | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | - | Hardening soil | - |
| Drainage type | - | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 20 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 22 | kN/m$^3$ |

| | | | |
|---|---|---|---|
| **Mechanical** | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}^{ref}$ | $40 \cdot 10^3$ | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}^{ref}$ | $40 \cdot 10^3$ | $kN/m^2$ |
| Unloading / reloading stiffness | $E_{ur}^{ref}$ | $120 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu_{ur}$ | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | - |
| Cohesion | $c'_{ref}$ | 10 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 30 | ° |
| **Interfaces** | | | |
| Strength determination | - | Rigid | - |
| Interface reduction factor | $R_{inter}$ | 1.0 | - |

**Table 12: Material properties of the soft rock layers**

| Parameter | Name | Clay-silt stone | Clay-limestone | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | Hoek-Brown | Hoek-Brown | - |
| Type of material behaviour | - | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 25 | 24 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 25 | 24 | $kN/m^3$ |
| **Mechanical** | | | | |
| Young's modulus | $E_{rm}$ | $1.0 \cdot 10^6$ | $2.5 \cdot 10^6$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 0.25 | 0.25 | - |

| Mechanical | | | | |
|---|---|---|---|---|
| Uniaxial compressive strength | $\mid \sigma_{ci} \mid$ | $25 \cdot 10^3$ | $50 \cdot 10^3$ | $kN/m^2$ |
| Material constant for the intact rock | $m_i$ | 4 | 10 | - |
| Geological Strength Index | $GSI$ | 40 | 55 | - |
| Disturbance factor | $D$ | 0.2 | 0.0 | - |
| Dilatancy parameter | $\psi_{max}$ | 30 | 35 | ° |
| Dilatancy parameter | $\sigma_{\psi}$ | 400 | 1000 | $kN/m^2$ |

| Interfaces | | | | |
|---|---|---|---|---|
| Strength determination | - | Manual | Rigid | - |
| Interface reduction factor | $R_{inter}$ | 0.5 | 1.0 | - |

1. Create soil material data sets according to Table 11 (on page 90) and assign them to the corresponding layers Figure 67 (on page 90) and while assigning the values for soft rock layers as per Table 12 (on page 91) ,find the analysis for various parameters after expanding the window one of which is shown in Figure 68 (on page 93).
2. Close the **Modify soil layers** window and proceed to the **Structures mode** to define the structural elements.

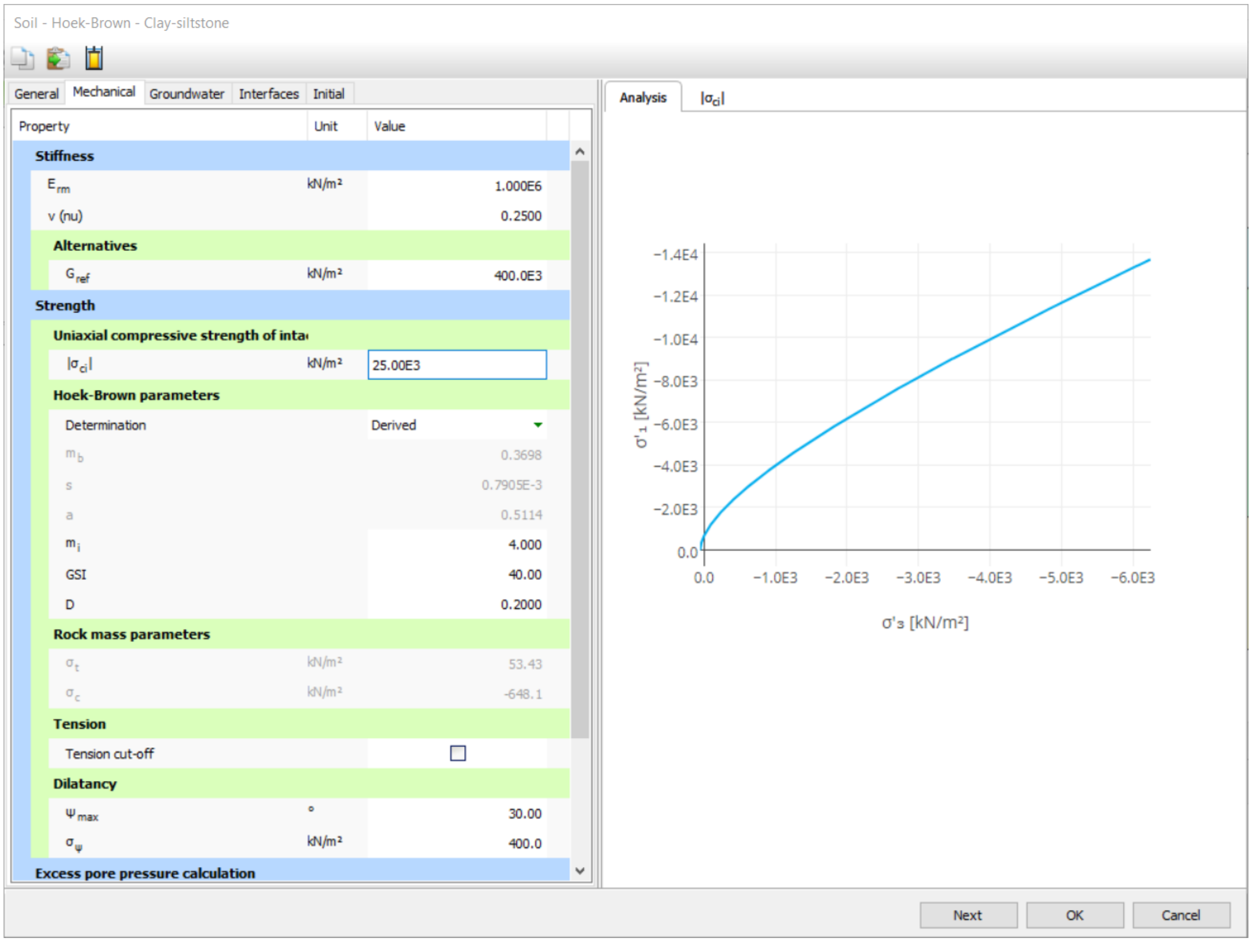



*Figure 68: Mechanical Parameter*

## 5.4 Define the tunnel

1. In the **Structures mode** click the **Create tunnel** button in the side toolbar and click on (0 16) in the drawing area to specify the location of the tunnel.
   The **Tunnel designer** window pops up.
2. The default shape option (**Free**) will be used. The default values of the rest of the parameters defining the location of the tunnel in the model are valid as well.
3. Click on the **Segments** tab.
4. Click the **Add section** button in the side toolbar. In the segment info box
   a. Set the **Segment type** to **Arc**.
   b. Set **Radius** to 10.4 m
   c. Set the **Segment angle** to 22°.
5. The default values of the remaining parameters are valid.

6. Click the **Add section** button to add a new arc segment.
   a. Set **Radius** to 2.4 m.
   b. Set the **Segment angle** to 47°.
   c. The default values of the remaining parameters are valid.
7. Click the **Add section** button to add a new arc segment.
   a. Set **Radius** to 5.8 m.
   b. Set the **Segment angle** to 50°.
   c. The default values of the remaining parameters are valid.
8. Click the **Extend to symmetry axis** option to complete the right half of the tunnel.
   A new arc segment is automatically added closing the half of the tunnel.
9. Click the **Symmetric close** button to complete the tunnel. Four new arc segment are automatically added closing the tunnel.
10. Click on the **Subsections** tab.
11. Click the **Add** button to add a new subsection. This subsection will be used to separate the top heading (upper excavation cluster) from the invert (lower excavation cluster).
    a. Set **Offset 2** to 3 m.
    b. From the **Segment type** drop-down menu select **Arc** option
    c. Set **Radius** to 11 m.
    d. **Segment angle** to 360°.
12. Click the **Select multiple objects** button and select all the geometric entities in the slice.
13. Click the **Intersect** button.
14. Delete the part of the subsection outside of the slice by selecting it in the display area and clicking the **Delete** button in the side toolbar.

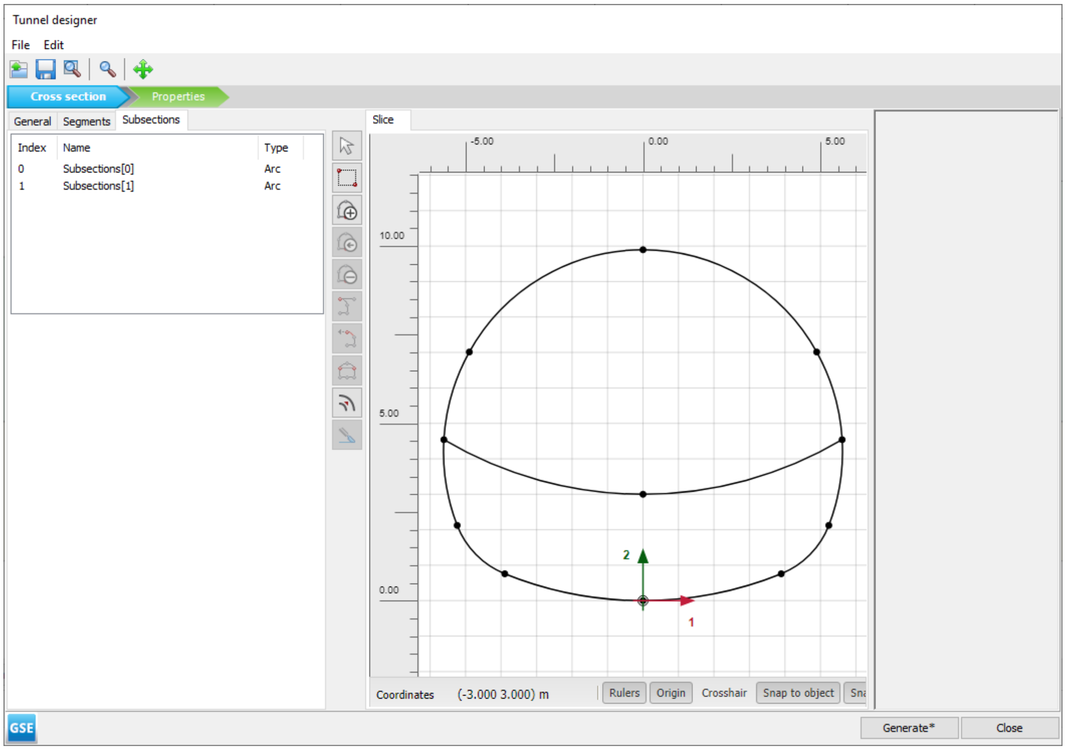



*Figure 69: Segments in the tunnel cross section*

15. Proceed to the **Properties mode** .
16. Multi-select the polycurves in the display area and select the **Create plate** option in the appearing menu.
17. The various segments in the tunnel cross section can be seen in Figure 69 (on page 95).
18. Press **<Ctrl + M>** to open the **Material sets** window. Create a new material dataset for the created plates according to Table 13 (on page 95).

**Table 13: Material properties of the plates**

| Parameter | Name | Lining | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| Unit Weight | w | 5 | kN/m/m |
| Prevent punching | - | No | - |
| **Mechanical** | | | |
| Isotropic | - | Yes | - |
| Axial stiffness | $EA_1$ | $6.0 \cdot 10^6$ | kN/m |

| **Mechanical** | | | |
|---|---|---|---|
| Bending stiffness | EI | $20{\cdot}10^3$ | $kNm^2/m$ |
| Poisson's ratio | ν | 0.15 | - |

**19.** Multi-select the created plates and in the **Selection explorer**, assign the material **Lining** to the selected plates.

**20.** Assign negative interfaces to the lines defining the shape of the tunnel (not the excavation levels). The final tunnel view in the **Tunnel designer** window is shown in Figure 70 (on page 96):

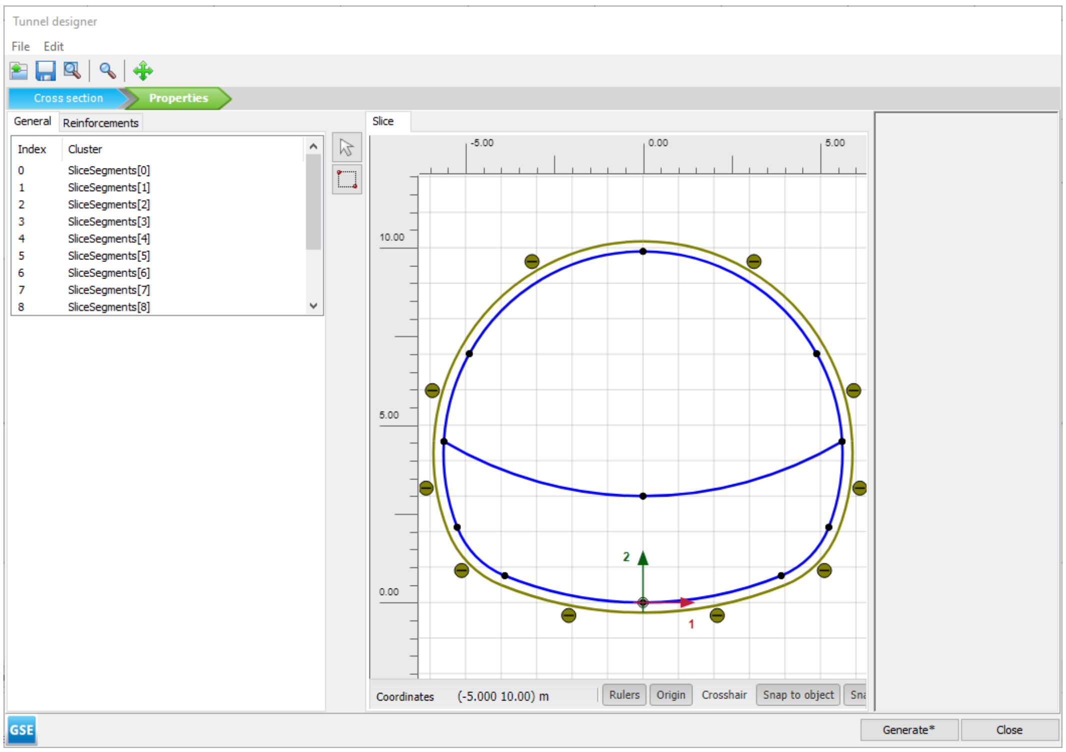



*Figure 70: Final tunnel*

**21.** Click on **Generate** to update the tunnel in the model and click **Close**.

## 5.5 Generate the mesh

The default global coarseness parameter (**Medium**) can be accepted in this case.

1. Proceed to the **Mesh mode**.
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 71 (on page 97).

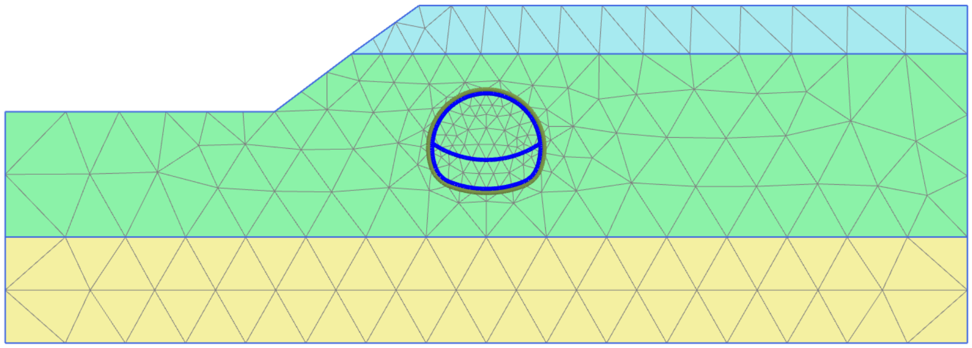


*Figure 71: The generated mesh*

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

## 5.6 Define and perform the calculation

To simulate the construction of the tunnel a staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. The calculation phases are **Plastic** analyses, **Staged construction**. The three-dimensional arching effect is emulated by using the so-called β-method. The idea is that the initial stresses $p_k$ acting around the location where the tunnel is to be constructed are divided into a part (1-β) $p_k$ that is applied to the unsupported tunnel and a part **Deconfinement** method that is applied to the supported tunnel.

To apply this method in PLAXIS 2D, one can use the **Deconfinement** option, which is available for each de-activated soil cluster in the model explorer. **Deconfinement** is defined as the aforementioned factor (1-β). For example, if 60% of the initial stresses in a de-activated soil cluster should disappear in the current calculation phase (so the remaining 40% is to be considered later), it means that the **Deconfinement** (1-β) parameter of that inactive cluster should be set to 60%. The value of **Deconfinement** can be increased in subsequent calculation phases until it reaches 100%.

To define the calculation process follow these steps:

### 5.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with the definition of the calculation phases.
2. The initial phase has already been introduced. Note that the soil layers are not horizontal. It is not recommended in this case to use the **K0 procedure** to generate the initial effective stresses. Instead **Gravity loading** will be used. This option is available in the **General** subtree of the **Phases** window.
3. Water will not be considered in this example. The general phreatic level should remain at the model base.
4. Make sure that the tunnel is inactive.

### 5.6.2 Phase 1: First tunnel excavation (deconfinement)

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** mode deactivate the upper cluster in the tunnel. Do NOT activate the tunnel lining.
3. While the de-activated cluster is still selected, in the **Selection explorer** set **Deconfinement(1 - β)** to 60 %.

The model for Phase 1 is displayed in Figure 72 (on page 98).

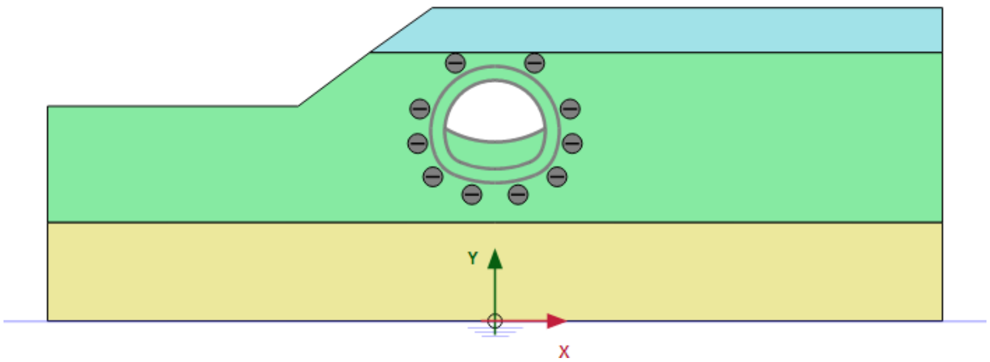



*Figure 72: Configuration of Phase 1*

### 5.6.3 Phase 2: First (temporary) lining

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction mode** , activate the lining and interfaces of the part of the tunnel excavated in the previous phase.
3. Select the de-activated cluster. In the **Selection explorer** set **Deconfinement** to 100 % and the model for phase 2 is shown in Figure 73 (on page 98).

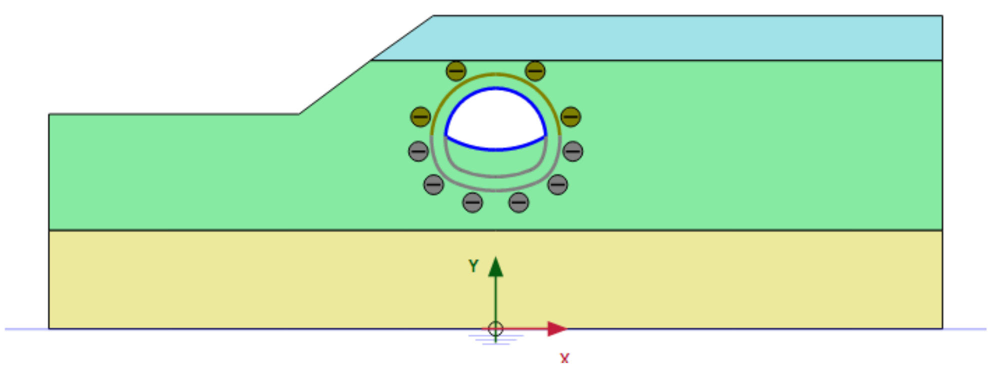



*Figure 73: Configuration of Phase 2*

## 5.6.4 Phase 3: Second tunnel excavation (deconfinement)

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction mode** deactivate the lower cluster (invert) and the temporary lining in the middle of the tunnel.
3. While the lower de-activated cluster is still selected, set in the **Selection explorer Deconfinement** to 60%.
4. The model for phase 3 can be seen in Figure 74 (on page 99).

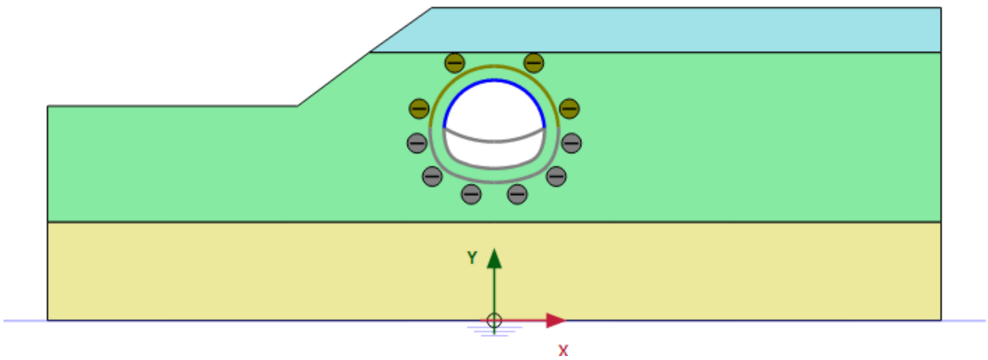



*Figure 74: Configuration of Phase 3*

## 5.6.5 Phase 4: Second (final) lining

1. Click the **Add phase** button to create a new phase.
2. Activate the remaining lining and interfaces.
   All the plates and interfaces around the full tunnel are active.
3. Select the lower de-activated cluster. In the **Selection explorer** set **Deconfinement** to 100 %.
4. The model for phase 4 can be seen in Figure 75 (on page 99).

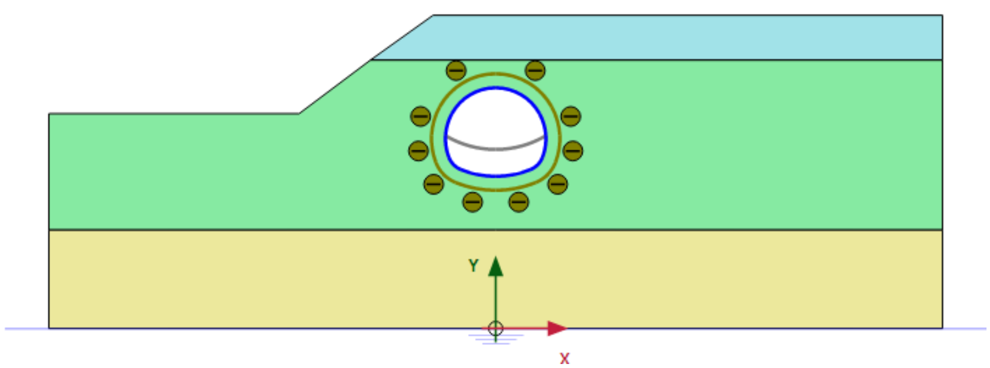



*Figure 75: Configuration of Phase 4*

### 5.6.6 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select a node at the slope crest point and the tunnel crest. These points might be of interest to evaluate the deformation during the construction phases.
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button.

## 5.7 Results

After the calculation, select the last calculation phase and click the **View calculation results** button. The **Output** program is started, showing the deformed mesh at the end of the calculation phases as shown in Figure 76 (on page 100):

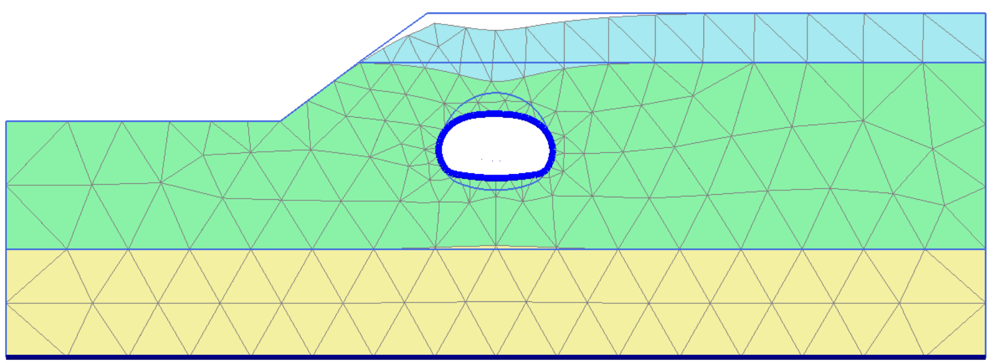


*Figure 76: The deformed mesh at the end of the final calculation phase*

To display the bending moments resulting in the tunnel:

1. To select the lining of all the tunnel sections, click the corresponding button in the side toolbar and drag the mouse to define a rectangle where all the tunnel sections are included. Select the **Plate** option in the appearing window as shown in Figure 77 (on page 101):

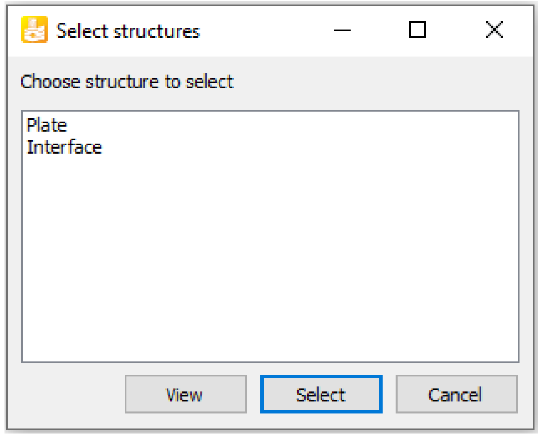



*Figure 77: Select structures window*

2. Click **View**.

   Note that the tunnel lining is displayed in the **Structures** view.
3. From the **Forces** menu select the **Bending moment M** option. The result, scaled by a factor of 0.5 is displayed in Figure 78 (on page 101).

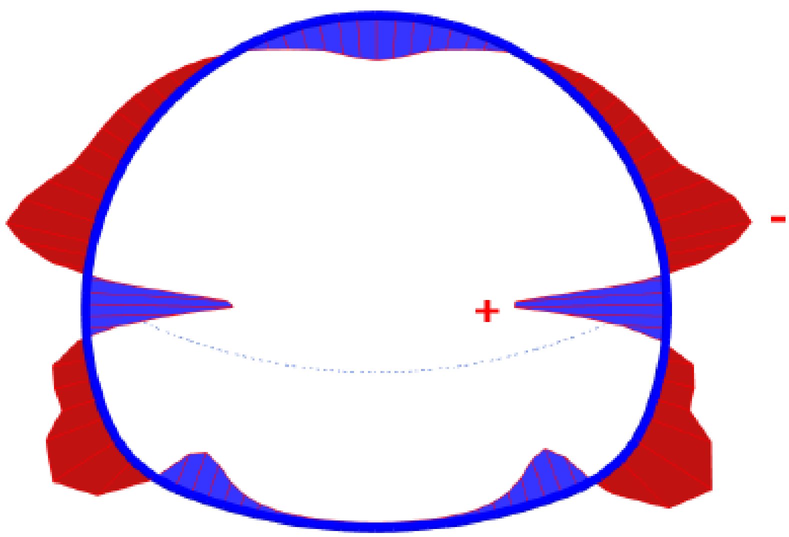



*Figure 78: Resulting bending moments in the NATM tunnel*

# 6 Dry excavation using a tie back wall [ADV]

This example involves the dry construction of an excavation. The excavation is supported by concrete diaphragm walls. The walls are tied back by prestressed ground anchors.

PLAXIS 2D allows for detailed modelling of this type of problem. It is demonstrated in this example how ground anchors are modelled and how prestressing is applied to the anchors. Moreover, the dry excavation involves a groundwater flow calculation to generate the new water pressure distribution. This aspect of the analysis is explained in detail.

**Objectives**

- Modelling ground anchors.
- Generating pore pressures with a groundwater flow calculation.
- Displaying the contact stresses and resulting forces in the model.
- Scaling the displayed results.

**Geometry**

The excavation is 20 m wide and 10 m deep. 16 m long concrete diaphragm walls of 0.35 m thickness are used to retain the surrounding soil. Two rows of ground anchors are used at each wall to support the walls. The anchors have a total length of 14.5 m and an inclination of 33.7°(2:3). On the left side of the excavation a surface load of 10 kN/m$^2$ is taken into account.

The relevant part of the soil consists of three distinct layers. From the ground surface to a depth of 3 m there is a fill of relatively loose fine sandy soil. Underneath the fill, down to a minimum depth of 15 m, there is a more or less homogeneous layer consisting of dense well-graded sand. This layer is particularly suitable for the installation of the ground anchors. The underlying layer consists of loam and lies to a large depth. 15 m of this layer is considered in the model as shown in Figure 79 (on page 102).

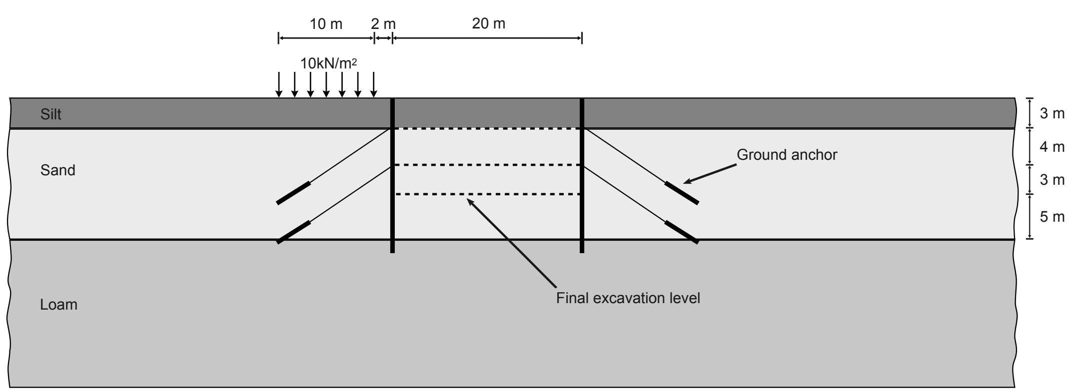



*Figure 79: Excavation supported by tie back walls*

## 6.1 Create new project

1. Start the Input program and select **Start a new project** from the **Quick start** 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 **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 100 m, $y_{min}$ = 0 m, $y_{max}$ = 30 m.
5. Keep the default values for units and the constants and press **OK** to close the **Project properties** window.

## 6.2 Define the soil stratigraphy

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Add three soil layers to the borehole. Locate the ground level at y = 30 m by assigning 30 to the **Top** level of the uppermost layer. The bottom levels of the layers are located at 27, 15 and 0 m, respectively.
3. Set the **Head** to 23 m.
   The layer stratigraphy is shown in Figure 80 (on page 103):

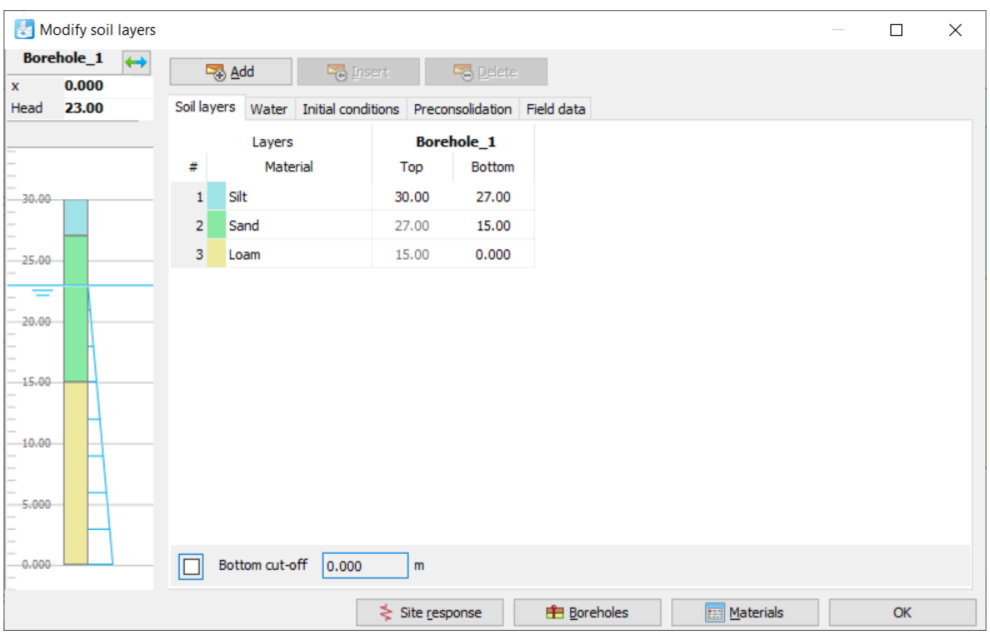



*Figure 80: The Modify soil layers window*

## 6.3 Create and assign material data sets

Three data sets need to be created. The materials have the following properties as shown in Table 14 (on page 104):

**Table 14: Soil and interface properties**

| Parameter | Name | Silt | Sand | Loam | Unit |
|---|---|---|---|---|---|
| **General** | | | | | |
| Soil model | Model | Hardening soil | Hardening soil | Hardening soil | - |
| Drainage type | Type | Drained | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 17 | 17 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | 20 | 19 | $kN/m^3$ |
| **Mechanical** | | | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $20 \cdot 10^3$ | $30 \cdot 10^3$ | $12 \cdot 10^3$ | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $20 \cdot 10^3$ | $30 \cdot 10^3$ | $8 \cdot 10^3$ | $kN/m^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $60 \cdot 10^3$ | $90 \cdot 10^3$ | $36 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu_{ur}$ | 0.2 | 0.2 | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | 0.5 | 0.8 | - |
| Cohesion (constant) | $c'_{ref}$ | 1 | 0 | 5 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 30 | 34 | 29 | ° |
| Dilatancy angle | $\psi$ | 0 | 4 | 0 | ° |
| $K_0$-value for normal consolidation | $K_0{}^{nc}$ | 0.5 | 0.4408 | 0.5152 | - |
| **Groundwater** | | | | | |
| Classification type | - | USDA | USDA | USDA | - |

| **Groundwater** | | | | | |
|---|---|---|---|---|---|
| SWCC fitting method | - | Van Genuchten | Van Genuchten | Van Genuchten | - |
| Soil class | - | Silt | Sand | Loam | - |
| < 2μm | - | 6.0 | 4.0 | 20.0 | % |
| 2μm - 50μm | - | 87.0 | 4.0 | 40.0 | % |
| 50μm - 2mm | - | 7.0 | 92.0 | 40.0 | % |
| Flow parameters - Use defaults | - | From data set | From data set | From data set | - |
| Permeability in horizontal direction | $k_x$ | 0.5996 | 7.128 | 0.2497 | m/day |
| Permeability in vertical direction | $k_y$ | 0.5996 | 7.128 | 0.2497 | m/day |
| **Interfaces** | | | | | |
| Strength determination | - | Manual | Manual | Rigid | - |
| Interface reduction factor | $R_{inter}$ | 0.65 | 0.70 | 1.0 | - |
| Consider gap closure | - | Yes | yes | yes | |
| **Initial** | | | | | |
| $K_0$ determination | - | Automatic | Automatic | Automatic | - |
| Pre-overburden pressure | *POP* | 0 | 0 | 25 | $kN/m^2$ |
| Over-consolidation ratio | *OCR* | 1.0 | 1.0 | 1.0 | - |

1. Define three data sets for soil and interfaces with the parameters given in Table 14 (on page 104).
2. Assign the material data sets to the corresponding soil layers (Figure 80 (on page 103)).

## 6.4 Define the structural elements

The creation of diaphragm walls, excavation levels, ground anchor and surface load is described below.

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .

### 6.4.1 To define the diaphragm wall and interfaces:

A diaphragm wall with the following material properties as shown in Table 15 (on page 106) has to be defined:

**Table 15: Material properties of the diaphragm wall (plate)**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| Material type | - | Elastic | - |
| Isotropic | - | Yes | - |
| Weight | $w$ | 8.3 | kN/m/m |
| Prevent punching | - | Yes | - |
| Axial stiffness | $EA_1$ | $12 \cdot 10^6$ | kN/m |
| Bending stiffness | $EI$ | $120 \cdot 10^3$ | kNm$^2$/m |
| Poisson's ratio | $\nu$ | 0.15 | - |

1. In the **Structures mode** , model the diaphragm walls as plates passing through (40 30) - (40 14) and (60 30) - (60 14).
2. Multi-select the plates in the model.
3. In the **Selection explorer** click on **Material**.

   The view will change displaying a drop-down menu and a plus button next to it as shown in Figure 81 (on page 106):

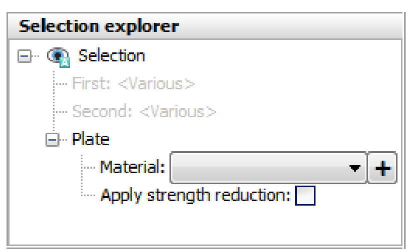



*Figure 81: Material assignment in the Selection explorer*

4. Click the Add button +.
   A new empty material set is created for plates.
5. Define the material data set for the diaphragm walls according to the properties are listed in Table 15 (on page 106). The concrete has a Young's modulus of 35 GN/m$^2$ and the wall is 0.35 m thick.
6. Assign positive and negative interfaces to the geometry lines created to represent the diaphragm walls.

### 6.4.2 To define the excavation levels:

The soil is excavated in three stages. The first excavation layer corresponds to the bottom of the silt layer and it is automatically created. To define the remaining excavation stages:

1. Define the second excavation phase by drawing a line through (40 23) and (60 23).
2. Define the third excavation phase by drawing a line through (40 20) and (60 20).

### 6.4.3 Defining the ground anchor

A ground anchor can be modelled by a combination of a node-to-node anchor and an embedded beam. The embedded beam simulates the grouted part of the anchor whereas the node-to-node anchor simulates the free length. In reality there is a complex three-dimensional state of stress around the grout body which cannot be simulated in a 2D model.

The coordinates of the anchor and grout body are listed in Table 16 (on page 107) , Table 17 (on page 107) and Table 18 (on page 108) and material properties of grout body are shown in Table 19 (on page 108).

**Table 16: Node to node anchor coordinates**

| Anchor location | Name | First point | Second point |
|---|---|---|---|
| Top | Left | (40 27) | (31 21) |
| | Right | (60 27) | (69 21) |
| Bottom | Left | (40 23) | (31 17) |
| | Right | (60 23) | (69 17) |

**Table 17: Properties of the anchor rod (node-to-node anchor)**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| Material type | - | Elastic | - |
| Axial stiffness | $EA$ | $500 \cdot 10^3$ | kN |
| Out-of-plane spacing | $L_{spacing}$ | 2.5 | m |

**Table 18: Grout coordinates**

| Anchor location | Name | First point | Second point |
|---|---|---|---|
| Top | Left | (31 21) | (28 19) |
| | Right | (69 21) | (72 19) |
| Bottom | Left | (31 17) | (28 15) |
| | Right | (69 17) | (72 15) |

**Table 19: Properties of the grout body (embedded beam )**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| **General** | | | |
| Material type | - | Elastic | - |
| Unit weight | $\gamma$ | 0 | $kN/m^3$ |
| **Mechanical** | | | |
| Pile spacing | $L_{spacing}$ | 2.5 | m |
| Beam type | - | Predefined | - |
| Predefined beam type | - | Solid circular beam | - |
| Diameter | $D$ | 0.3 | m |
| Stiffness | $E$ | $7.07 \cdot 10^6$ | $kN/m^2$ |
| Axial Skin resistance | Distribution | Linear | - |
| | $T_{skin,\ start,\ max}$ | 400 | kN/m |
| | $T_{skin,\ end,\ max}$ | 400 | kN/m |
| Lateral resistance | Lateral resistance | Unlimited | - |
| Base resistance | $F_{max}$ | 0 | kN |
| Interface stiffness factor | Default values | Yes | - |

1. Define the node-to-node anchors according to Table 16 (on page 107).
2. Create an **Anchor** material data set according to the parameters specified in Table 17 (on page 107).
3. Multi-select the anchors in the drawing area. Assign the material data set by selecting the corresponding option in the **Material** drop-down menu in the **Selection explorer**.

4. Define the grout body using the **Embedded beam** button according to Table 18 (on page 108).
5. Create the **Grout** material data set according to the parameters specified in Table 19 (on page 108) and assign it to the grout body.
6. Set the **Behaviour** of the embedded beam to **Grout body** as shown in Figure 82 (on page 109).

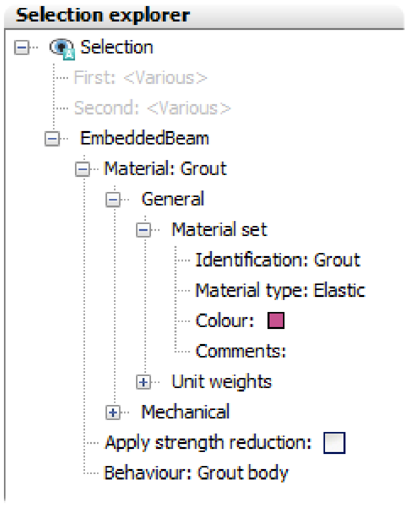



*Figure 82: Embedded beam in the Selection explorer*

The connection with the anchor will be automatically established.

7. Multi-select (keep **<Ctrl>** pressed while selecting) the top node-to-node anchors and embedded beams. Right-click on the selected region and select the **Group** option in the context menu.
8. In the **Model explorer** expand the **Groups subtree**.
   Note that a group is created composed of the elements of the top ground anchors.
9. Click on **Group_1** in the **Model explorer** and type a new name (e.g `GroundAnchor_Top`).
10. Follow the same steps to create a group and to rename the bottom ground anchors.

Although the precise stress state and interaction with the soil cannot be modelled with this 2D model, it is possible in this way to estimate the stress distribution, the deformations and the stability of the structure on a global level, assuming that the grout body does not slip relative to the soil. With this model it is certainly not possible to evaluate the pullout force of the ground anchor.

### 6.4.4 To define the distributed load:

1. Create a line load between (28 30) and (38 30).

## 6.5 Generate the mesh

In order to generate the mesh, follow these steps:

1. Proceed to the **Mesh mode**.
2. Click the **Generate mesh** button in the side toolbar. Use the default option for the **Element distribution** parameter (**Medium**).
3. Click the **View mesh** button to view the mesh as shown in Figure 83 (on page 110).

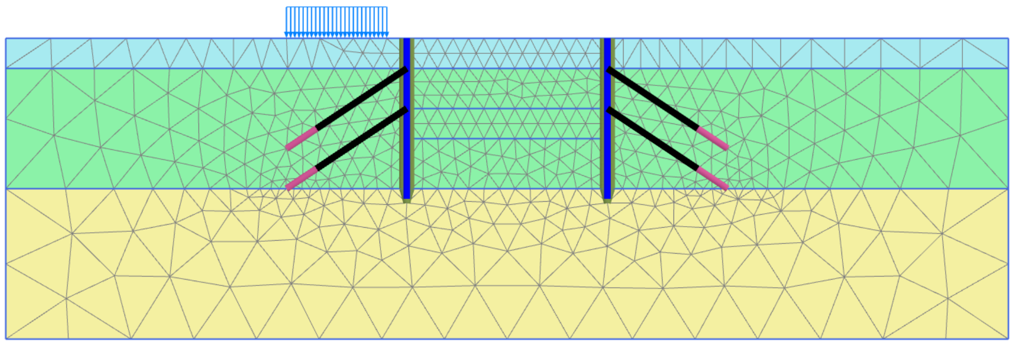


*Figure 83: The generated mesh*

## 6.6 Define and perform the calculation

The calculation of this project consists of six phases. In the initial phase (Phase 0), the initial stresses are generated. In Phase 1, the walls are constructed and the surface loads are activated. In Phase 2, the first 3 m of the pit is excavated without connection of anchors to the wall. At this depth the excavation remains dry. In Phase 3, the first anchor is installed and pre-stressed. Phase 4 involves further excavation to a depth of 7 m. At this depth the excavation still remains dry. In Phase 5, the second anchor is installed and pre-stressed. Phase 6 is a further excavation to the final depth of 10 m including the dewatering of the excavation.

Before defining the calculation phases, the water levels to be considered in the calculation can be defined in the **Flow conditions** mode. The water level is lowered in the final excavation phase. At the side boundaries, the groundwater head remains at a level of 23.0 m. The bottom boundary of the problem should be closed. The flow of groundwater is triggered by the fact that the pit is pumped dry. At the bottom of the excavation the water pressure is zero, which means that the groundwater head is equal to the vertical level (head = 20.0 m). This condition can be met by drawing a new general phreatic level and performing a groundwater flow calculation. Activating the interfaces during the groundwater flow calculation prevents flow through the wall.

### 6.6.1 Initial phase

The initial stress field is generated by means of the **K0 procedure** using the default $K_0$-values in all clusters defined automatically by the program.

1. Proceed to the **Staged construction mode** .

2. Initially, all structural components and loads are inactive. Hence, make sure that the plates, the node-to-node anchors, the embedded beam rows and the surface loads are deactivated.
3. In the **Phases explorer** double-click the initial phase. The default parameters for the initial phase will be used. The **Phreatic** option is selected as **Pore pressure calculation type**. Note that when the pore pressures are generated by phreatic level, the full geometry of the defined phreatic level is used to generate the pore pressures.
4. Click **OK** to close the **Phases** window.
5. In the **Model explorer** expand the **Model conditions** subtree.
6. Expand the **Water** subtree.
   The water level created according to the head value specified in the borehole, (**BoreholeWaterLevel_1**), is automatically assigned to **GlobalWaterLevel** .
7. The model for initial phase is shown in Figure 84 (on page 111).

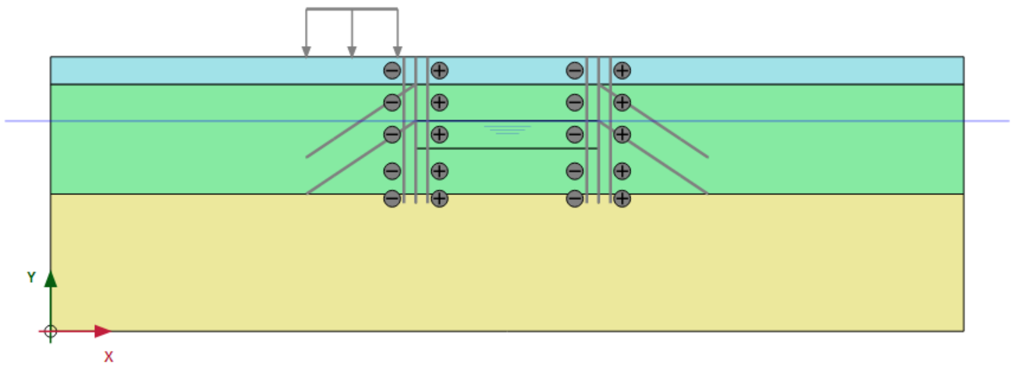



*Figure 84: Configuration of the initial phase*

## 6.6.2 Phase 1: Activation of wall and load

1. Click the **Add phase** button to create a new phase.
2. In the **Staged constructions mode** activate all walls and interfaces by clicking on the checkbox in front of them in the **Model explorer**.
   The active elements in the project are indicated by a green check mark.
3. Activate the distributed load.
4. After selecting the line load assign a value of -10 to $q_{y,start,ref}$ in the **Selection explorer** shown in Figure 85 (on page 111):

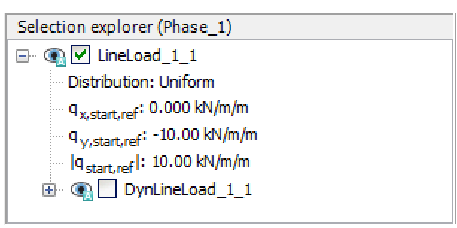



*Figure 85: Line load in the Selection explorer*

The model for the phase 1 in the **Staged construction mode** is displayed in Figure 86 (on page 112):

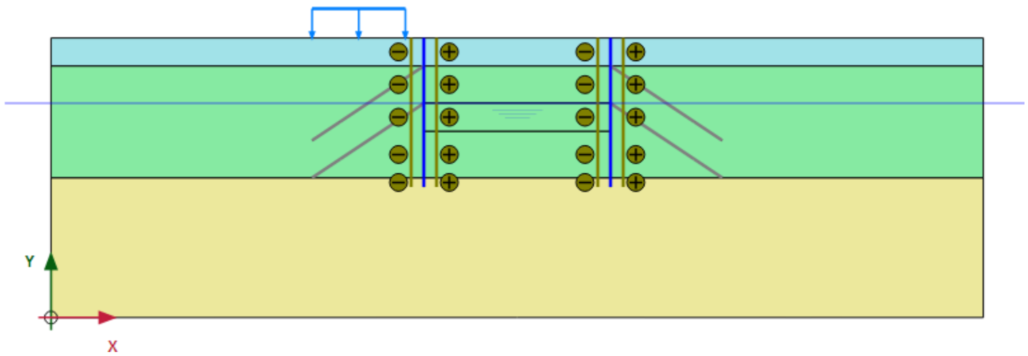


*Figure 86: Configuration of Phase 1 in the Staged construction mode*

### 6.6.3 Phase 2: First excavation

1. Click the **Add phase** button in the **Phases explorer** to add a new phase.
2. In the **Staged construction mode** de-activate the upper cluster of the excavation

The model for the first excavation phase is shown in Figure 87 (on page 112):

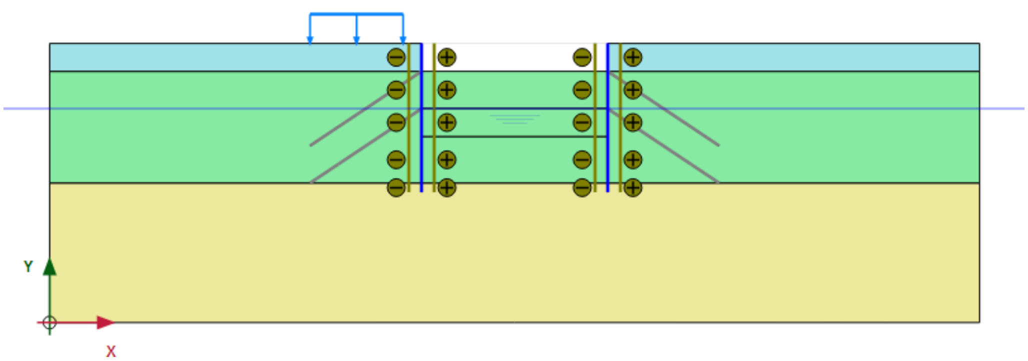


*Figure 87: Configuration of Phase 2 in the Staged construction mode*

### 6.6.4 Phase 3: First anchor row

1. Click the **Add phase** button in the **Phases explorer**.
2. Activate the upper ground anchors by clicking on the checkbox in front of **GroundAnchors_Top** under the **Groups** subtree in the **Model explorer**.
3. Multi-select the top node-to-node anchors.
4. In the **Selection explorer** set the **Adjust prestress** parameter to **True** and assign a pre-stress force of 500 kN.

> **Note:** A pre-stress force is exactly matched at the end of a finished staged construction calculation and turned into an anchor force. In successive calculation phases the force is considered to be just an anchor force and can therefore further increase or decrease, depending on the development of the surrounding stresses and forces.

The model for the phase 3 in the Staged construction mode is shown in Figure 88 (on page 113):

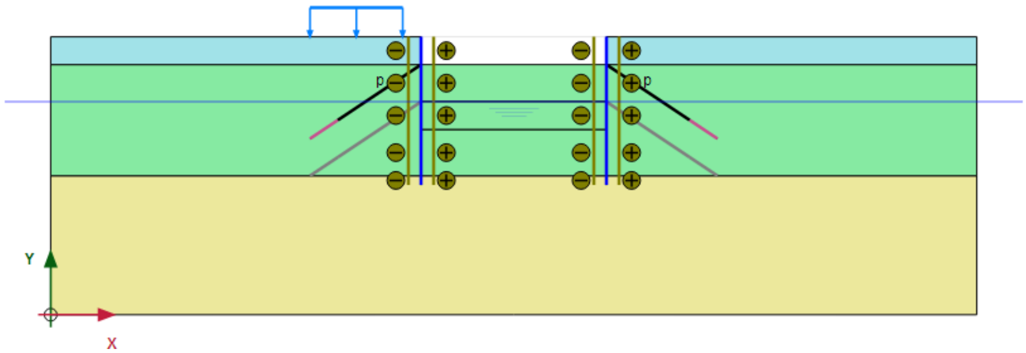



*Figure 88: Configuration of Phase 3 in the Staged construction mode*

## 6.6.5 Phase 4: Second excavation

1. Click the **Add phase** button to add a new phase.
2. Deactivate the second cluster of the excavation.

The model for the phase 4 in the **Staged construction** mode is displayed in Figure 89 (on page 113):

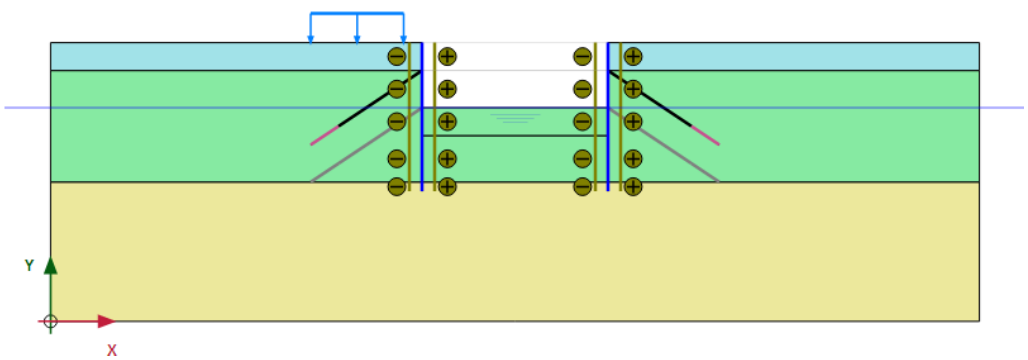



*Figure 89: Configuration of Phase 4 in the Staged construction mode*

Note that the anchors are not pre-stressed anymore.

### 6.6.6 Phase 5: Second anchor row

1. Click the **Add phase** button to add a new phase.
2. Activate the lower ground anchors.
3. Select the bottom node-to-node anchors.
4. In the **Selection explorer** set the **Adjust prestress** parameter to **True** and assign a pre-stress force of 1000 kN.

The model for the phase 5 in the **Staged construction** mode is displayed in Figure 90 (on page 114):

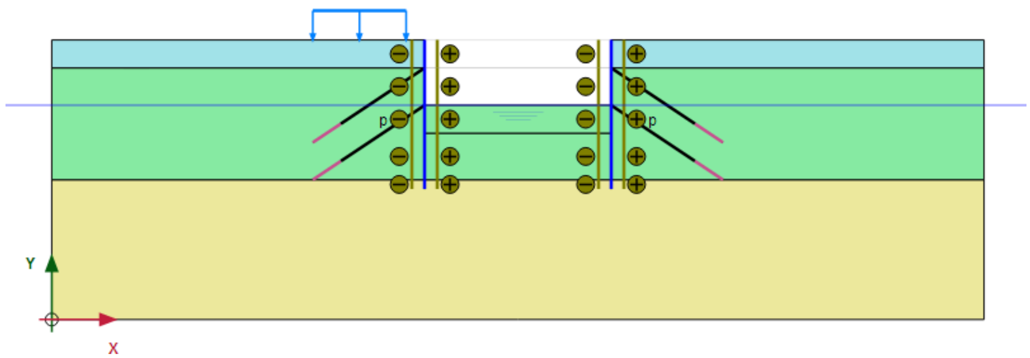



*Figure 90: Configuration of Phase 5 in the Staged construction mode*

### 6.6.7 Phase 6: Final excavation

1. Click the **Add phase** button to add a new phase.
2. In the **Phases** window, within **General** > **Pore pressure calculation type** select the **Steady state groundwater flow** option . The default values of the remaining parameters are valid.
3. Deactivate the third cluster of the excavation.
4. Click the **Flow conditions** tab to display the corresponding mode.
5. In the **Model explorer** expand the **Attributes library**.
6. Expand the **Water levels** subtree.
7. Click the **Create water level** button in the side toolbar and draw a new phreatic level. Start at (0 23) and draw the phreatic level through (40 20), (60 20) and end in (100 23).
8. In the **Model explorer** expand the **User water levels** subtree. Click on **UserWaterLevel_1** and type `LoweredWaterLevel` to rename the water level created in the **Flow conditions** mode as shown in Figure 91 (on page 115).

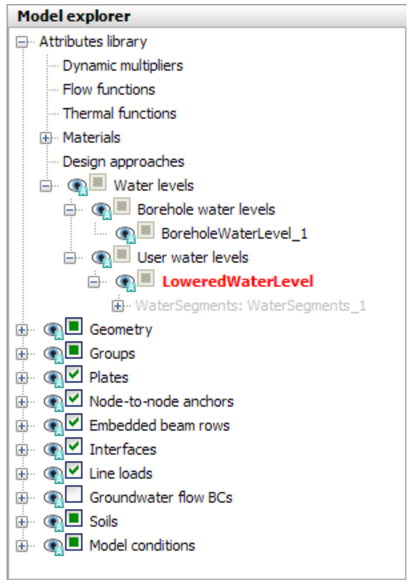



*Figure 91: Water levels in the Model explorer*

9. In the **Model explorer** expand **Model conditions** > **GroundwaterFlow**. The default boundary conditions are valid which is shown in Figure 92 (on page 115).

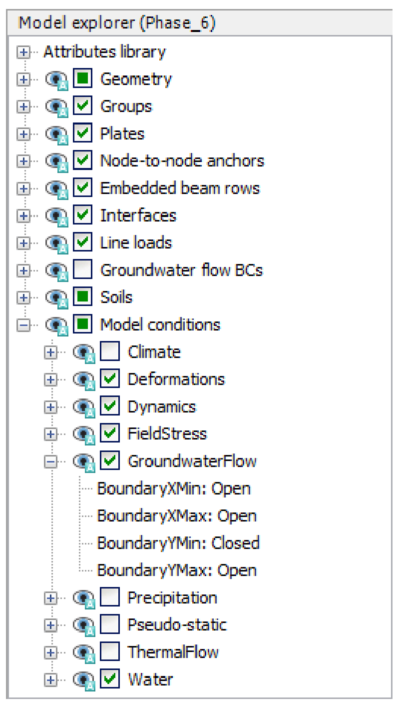



*Figure 92: The GroundwaterFlow subtree under the Model conditions in the Model explorer*

10. In the **Water** subtree assign the **LoweredWaterLevel** to **GlobalWaterLevel**.

The model and the defined water levels are displayed in Figure 93 (on page 116):

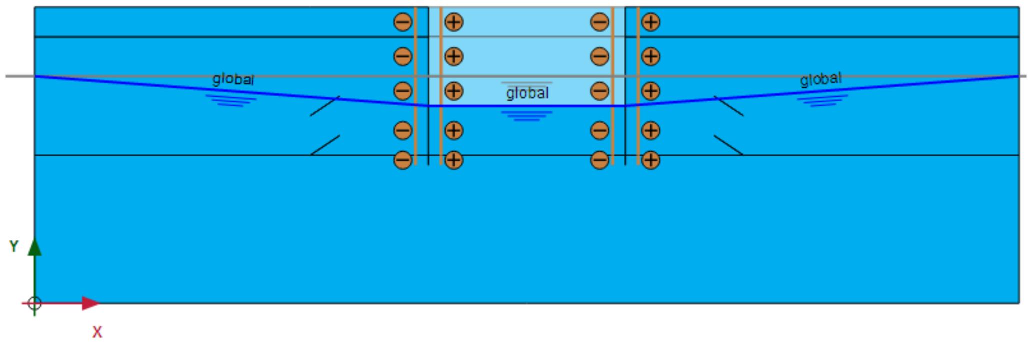



*Figure 93: Configuration of Phase 6 in the Flow conditions mode*

**Note:** Note that for **Groundwater flow** (steady or transient) the intersection points of the water level with the active model boundaries are important. The program calculates flow boundary conditions in terms of a groundwater head corresponding to the water level. The 'internal' part of the water level is not used and will be replaced by the phreatic level resulting from the groundwater flow calculation. Hence, the water level tool is just a convenient tool to create boundary conditions for a flow calculation.

### 6.6.8 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select some characteristic points for curves (for example the connection points of the ground anchors on the diaphragm wall, such as (40 27) and (40 23).
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button .

## 6.7 Results

The deformed meshes at the end of calculation phase 2 to phase 6 are shown in Figure 94 (on page 117), Figure 95 (on page 117), Figure 96 (on page 117), Figure 97 (on page 118) and Figure 98 (on page 118):

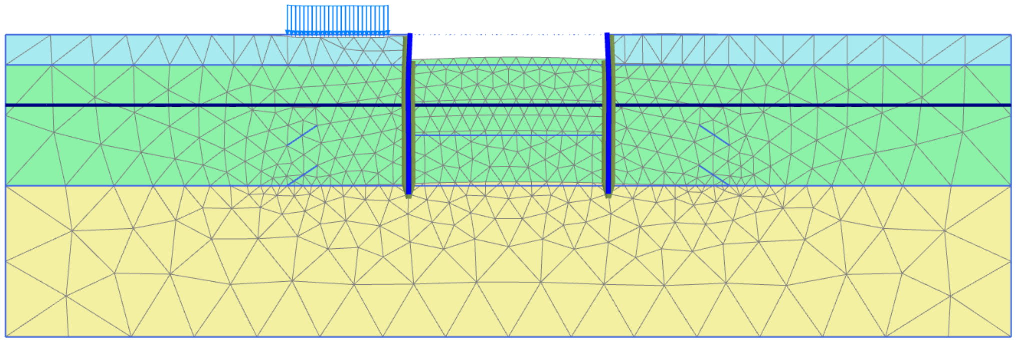


*Figure 94: Deformed mesh (scaled up 50.0 times) - Phase 2*

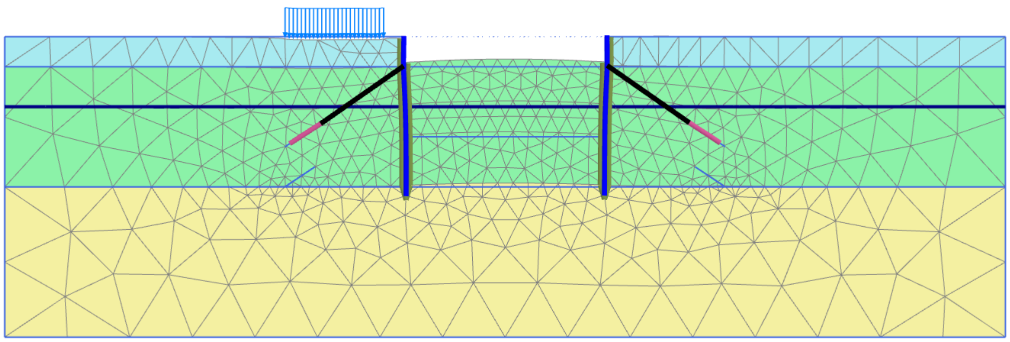


*Figure 95: Deformed mesh (scaled up 50.0 times) - Phase 3*

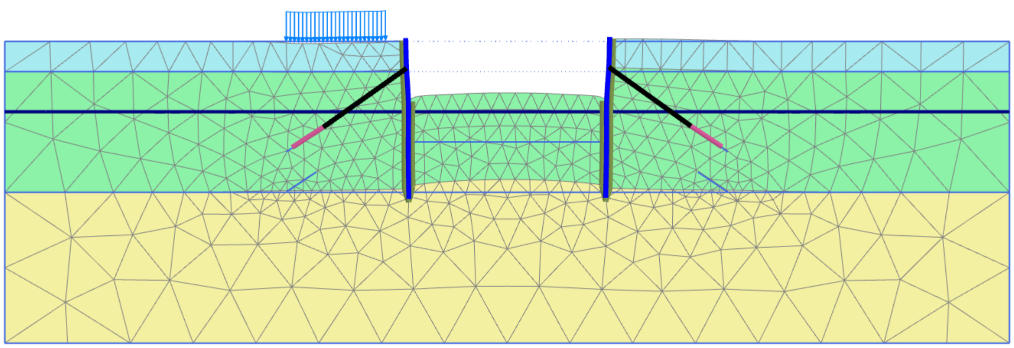


*Figure 96: Deformed mesh (scaled up 50.0 times) - Phase 4*

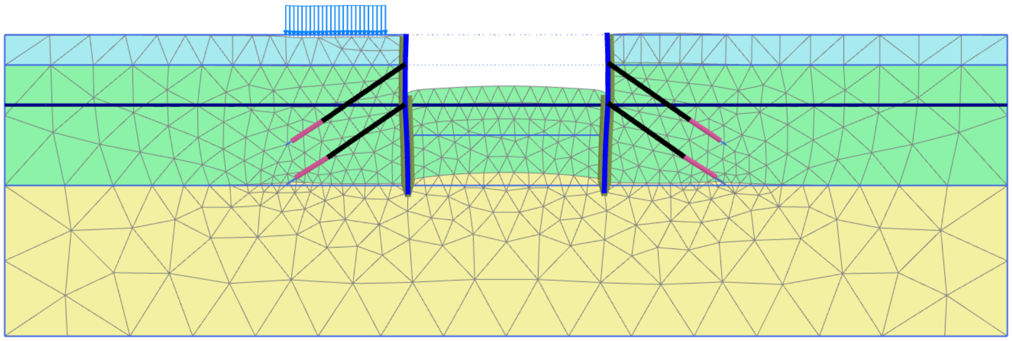


*Figure 97: Deformed mesh (scaled up 50.0 times) - Phase 5*

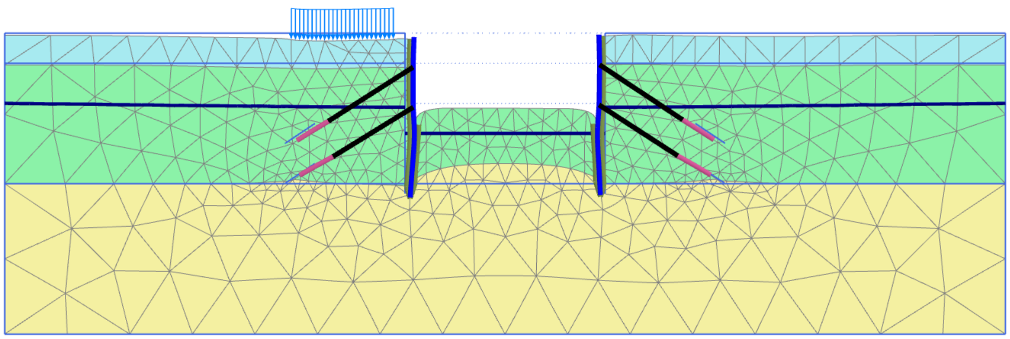


*Figure 98: Deformed mesh (scaled up 50.0 times) - Final phase*

Figure 99 (on page 118) shows the effective principal stresses in the final situation. The passive stress state beneath the bottom of the excavation is clearly visible. It can also be seen that there are stress concentrations around the grout anchors.

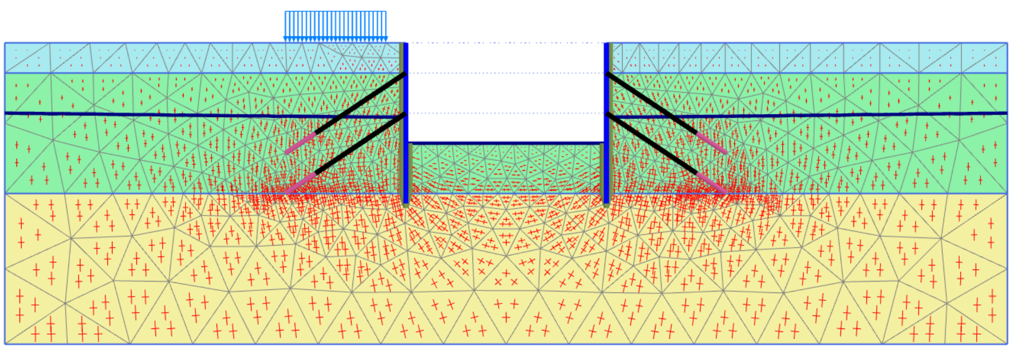


*Figure 99: Principal effective stresses (final stage)*

Figure 100 (on page 119) shows the bending moments in the diaphragm walls in the final state. The two dips in the line of moments are caused by the anchor forces.

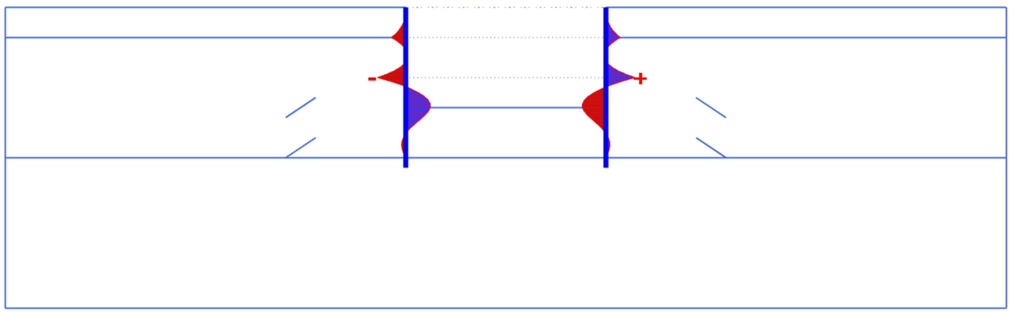


*Figure 100: Bending moments in the diaphragm walls in the final stage*

The anchor force can be viewed by double clicking the anchor. When doing this for the results of the third and the fifth calculation phase, it can be checked that the anchor force is indeed equal to the specified pre-stress force in the calculation phase they are activated. In the following phases this value might change due to the changes in the model.

# 7 Dry excavation using a tie back wall - ULS [ADV]

In this tutorial an Ultimate Limit State (ULS) calculation will be defined and performed for the dry excavation using a tie back wall (Dry excavation using a tie back wall [ADV] (on page 102)). The same geometry model will be used. The **Design approaches** feature is introduced in this example. This feature allows for the use of partial factors for loads and model parameters after a serviceability calculation has already been performed.

**Objective**

- Using **Design approaches**

## 7.1 Define the geometry

In order to define a design approach:

1. Open the project created in Dry excavation using a tie back wall [ADV] (on page 102) and save it under a different name.
2. Select the menu **Soil** > **Design approaches** or **Structures** > **Design approaches**.
   The corresponding window is displayed as shown in Figure 101 (on page 121).
3. Click the **Add** button.
   A new design approach is added in the list.
4. In this example the design approach 3 of the Eurocode 7 will be used. This design approach involves partial factors for loads and partial factors for materials (strength). Click the design approach in the list and specify a representative name (ex: 'Eurocode 7 - DA 3').

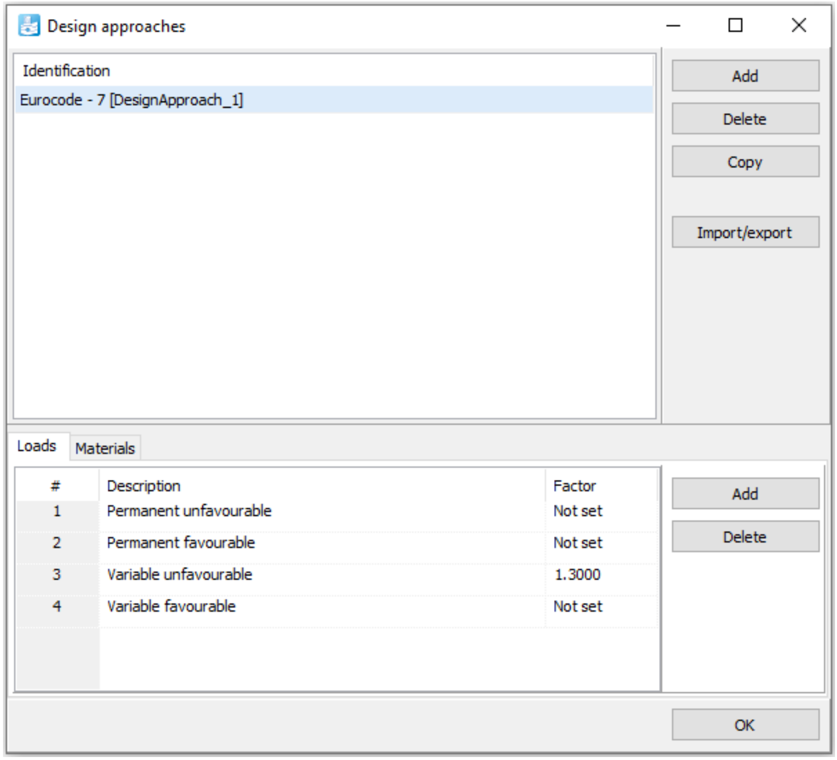



*Figure 101: Partial factors for loads*

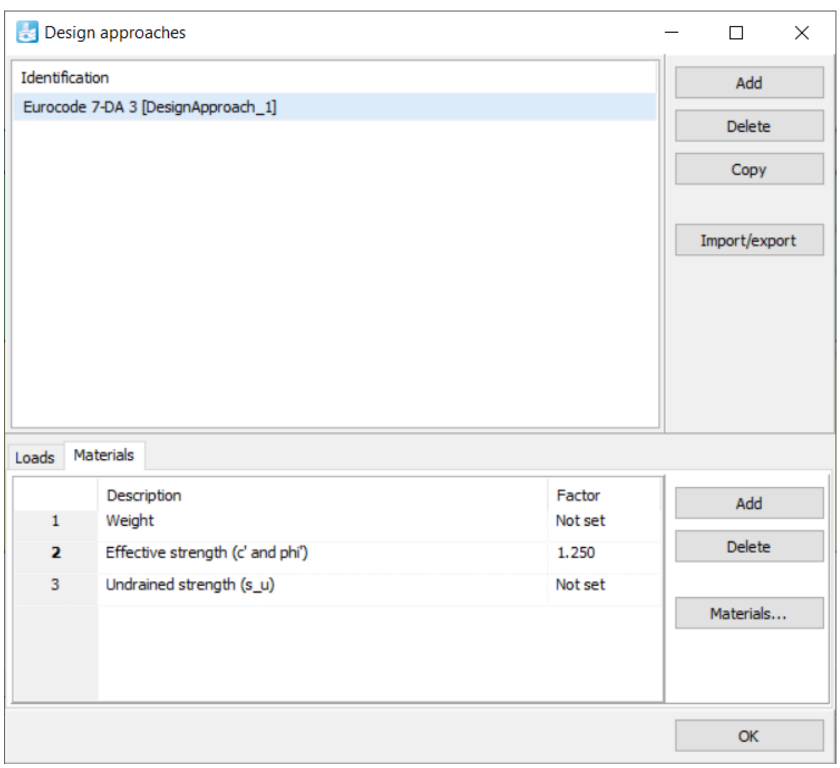



*Figure 102: Partial factors for materials*

5. In the lower part of the window the partial factors can be defined for loads and materials as shown in Figure 101 (on page 121). Set the partial factor for **Variable unfavourable** to 1.3.
6. Click the **Materials** tab.
7. Assign a value of 1.25 to **Effective strength (c' and phi')** as shown in Figure 102 (on page 121).
8. Click the **Materials...** button. The **Material sets** window pops up.
9. Open the **Loam** material data set. Note that the view has changed. In the current view it is possible to assign factors to different soil parameters, as well as to see the effect of these factors on the soil parameters.
10. Click the **Mechanical** tab. In the **Mechanical** tabsheet select the corresponding labels for $c'_{ref}$ and φ'.
11. Do the same for the remaining soil data sets.
12. Close the **Design approaches** window.

**Note:**

Note that a partial factor for $\phi$ and ψ applies to the tangent of $\phi$ and ψ respectively.

## 7.2 Define and perform the calculation

There are two main schemes to perform design calculations in relation to serviceability calculations (see Design approaches in the Reference Manual ). The first approach is used in this tutorial.

### 7.2.1 Changes to all phases

1. Proceed to the **Staged construction mode**.
2. In the **Phases explorer** click the phase Phase_1.
3. Add a new phase.
4. Double-click the newly added phase to open the **Phases** window.
5. In the **General** subtree of the **Phases** window select the defined design approach in the corresponding drop-down menu.
6. In the **Model explorer** expand the **Line loads** and all the subtrees under it.
7. Select the **Variable unfavourable** option in the **LoadFactorLabel** drop-down menu of the static component of the load.

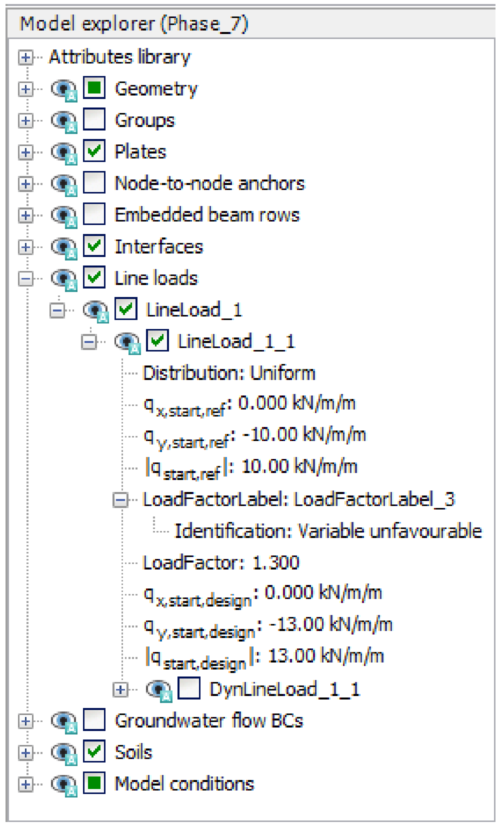



*Figure 103: Assignment of factor label to loads in the Selection explorer*

8. Follow the same steps to define ULS phases for all the remaining SLS phases. Make sure that the Phase 7 starts from Phase 1, Phase 8 from Phase 2, Phase 9 from Phase 3 and so on.

### 7.2.2 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select some characteristic points for curves (for example the connection points of the ground anchors on the diaphragm wall, such as (40 27) and (40 23)).
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button .

## 7.3 Results

The results obtained for the design approach phases can be evaluated in Output. Figure 104 (on page 124) displays the ΣMstage - |u| plot for the node located at (40.0 27.0).

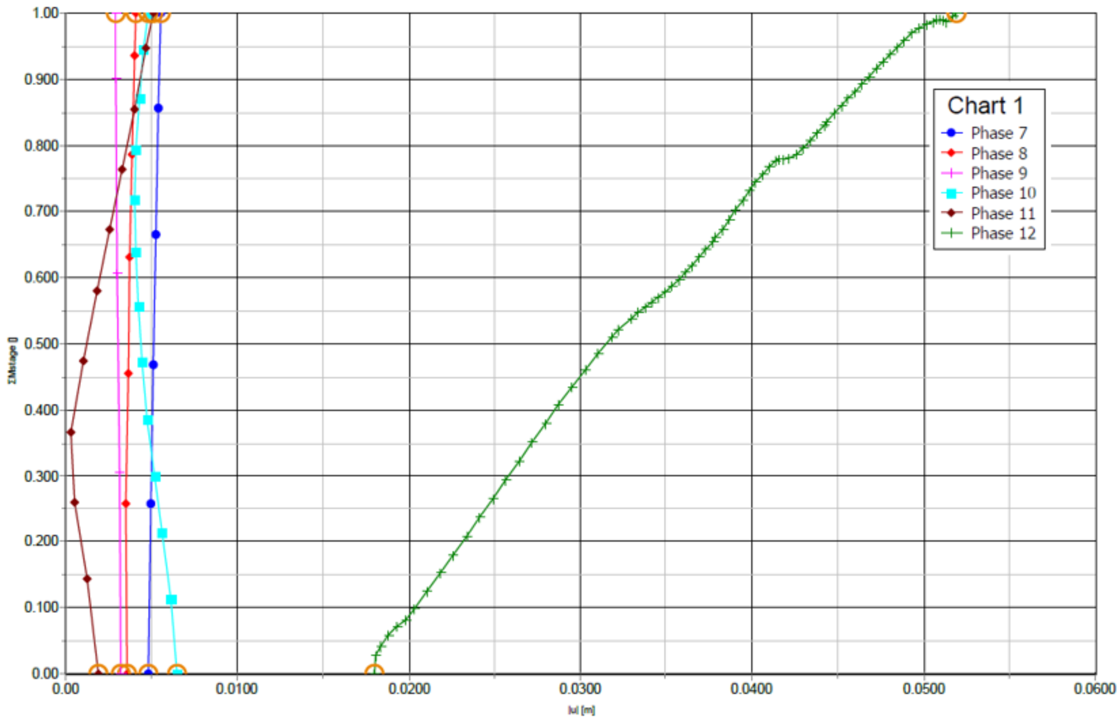



*Figure 104: ΣMstage - |u| plot for the ULS phases*

If the ULS calculations have successfully finished, the model complies with the corresponding design approach. If there are doubts about this due to excessive deformations, an additional **Safety** calculation may be considered using the same design approach, which should then result in a stable $\Sigma M_{sf}$ value larger than 1.0. Note that if partial factors have been used it is not necessary that $\Sigma M_{sf}$ also includes a safety margin. Hence, in this case $\Sigma M_{sf}$ just larger that 1.0 is enough.

Figure 105 (on page 125) displays the ΣMsf - |u| plot for the **Safety** calculations of the Phase 6 and the corresponding ULS phase (Phase 12). It can be concluded that the situation complies with the design requirements.

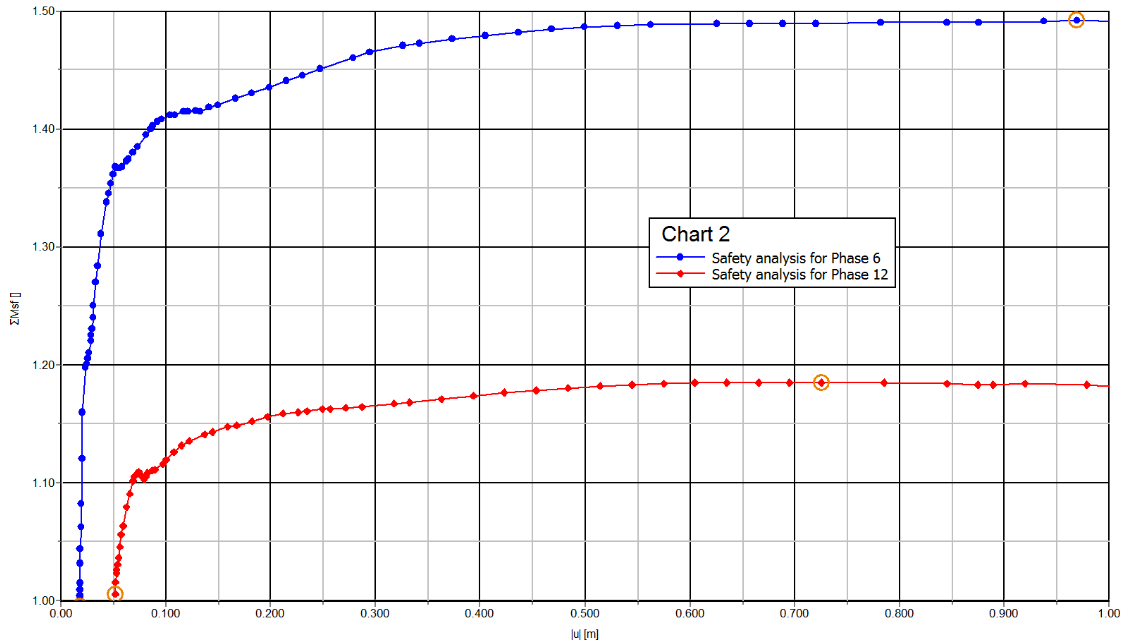



*Figure 105: ΣMsf - |u| plot for the last calculation phase and the corresponding ULS phase*

# 8 Construction of a road embankment [ADV]

The construction of an embankment on soft soil with a high groundwater level leads to an increase in pore pressure. As a result of this undrained behaviour, the effective stress remains low and intermediate consolidation periods have to be adopted in order to construct the embankment safely. During consolidation the excess pore pressures dissipate so that the soil can obtain the necessary shear strength to continue the construction process.

This tutorial concerns the construction of a road embankment in which the mechanism described above is analysed in detail. In the analysis three new calculation options are introduced, namely a consolidation analysis, an updated mesh analysis and the calculation of a safety factor by means of a safety analysis (strength reduction).

**Objectives**

- Consolidation analysis
- Modelling drains
- Change of permeability during consolidation
- Safety analysis (strength reduction)
- Updated mesh analysis (large deformations)

**Geometry**

The embankment is 16 m wide and 4 m high. The slopes have an inclination of 1:3. The problem is symmetric, so only one half is modelled (in this case the right half is chosen). The embankment itself is composed of loose sandy soil. The subsoil consists of 6 m of soft soil. The upper 3 m is peat and the lower 3 m is clay. The phreatic level is located 1 m below the original ground surface. Under the soft soil layers there is a dense sand layer of which 4 m are considered in the model which is shown in Figure 106 (on page 126).

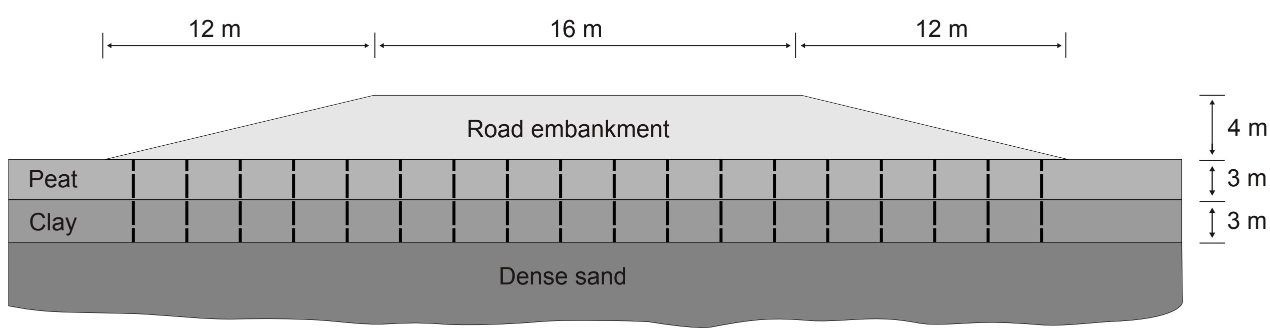



*Figure 106: Situation of a road embankment on soft soil*

# 8.1 Create new project

1. Start PLAXIS 2D by double clicking the icon of the Input program .
2. Click **Start a new project**.
3. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
4. In the **Model** tabsheet make sure that **Model** is set to **Plane strain** and that **Elements** is set to **15-Noded**.
5. Set the model **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 60 m, $y_{min}$ = -10 m and $y_{max}$ = 4 m.

# 8.2 Define the soil stratigraphy

The sub-soil layers are defined using a borehole. The embankment layers are defined in the **Structures mode** .

To define the soil stratigraphy:

1. Click the **Create borehole** button  and create a borehole at x = 0.
   The **Modify soil layers** window pops up as shown in Figure 107 (on page 127).

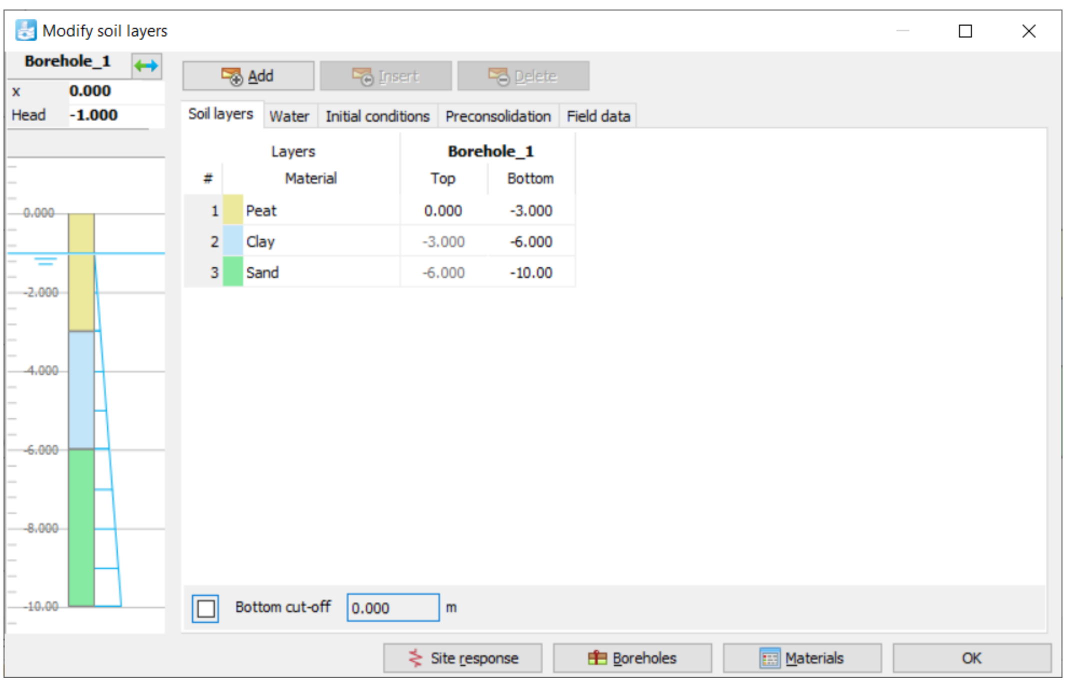



*Figure 107: Soil layer distribution*

2. Define three soil layers as shown in figure .
3. The water level is located at y = -1 m. In the borehole column specify a value of -1 to **Head**.

## 8.3 Create and assign material data sets

A number of material set are needed for this tutorial. The properties of the materials are as follows:

**Table 20: Material properties of the sand and clay layer and the interfaces**

| Parameter | Name | Embankment | Sand | Peat | Clay | Unit |
| --- | --- | --- | --- | --- | --- | --- |
| **General** | | | | | | |
| Soil model | - | Hardening soil | Hardening soil | Soft soil | Soft soil | - |
| Drainage type | - | Drained | Drained | Undrained (A) | Undrained (A) | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 17 | 8 | 15 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 19 | 20 | 12 | 18 | $kN/m^3$ |
| Initial void ratio | $e_{init}$ | 0.5 | 0.5 | 2.0 | 1.0 | - |
| **Mechanical** | | | | | | |
| Modified compression index | $\lambda^*$ | - | - | 0.15 | 0.05 | - |
| Modified swelling index | $\kappa^*$ | - | - | 0.03 | 0.01 | - |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $25 \cdot 10^3$ | $35 \cdot 10^3$ | - | - | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $25 \cdot 10^3$ | $35 \cdot 10^3$ | - | - | $kN/m^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $75 \cdot 10^3$ | $105 \cdot 10^3$ | - | - | $kN/m^2$ |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | 0.5 | - | - | - |
| Cohesion (constant) | $c'_{ref}$ | 1 | 0 | 2 | 1 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 30 | 33 | 23 | 25 | ° |
| Dilatancy angle | $\psi$ | 0 | 3 | 0 | 0 | ° |
| Miscellaneous: Set to default | | Yes | Yes | Yes | Yes | - |

| | | | | | | |
|---|---|---|---|---|---|---|
| ***Groundwater*** | | | | | | |
| Classification type | - | USDA | USDA | USDA | USDA | - |
| SWCC fitting method | - | Van Genuchten | Van Genuchten | Van Genuchten | Van Genuchten | - |
| Soil class | - | Loamy sand | Sand | Clay | Clay | - |
| < 2μm | - | 6.0 | 4.0 | 70.0 | 70.0 | % |
| 2μm - 50μm | - | 11.0 | 4.0 | 13.0 | 13.0 | % |
| 50μm - 2mm | - | 83.0 | 92.0 | 17.0 | 17.0 | % |
| Use defaults | - | From data set | From data set | None | From data set | - |
| Horizontal permeability | $k_x$ | 3.499 | 7.128 | 0.1 | 0.04752 | m/day |
| Vertical permeability | $k_y$ | 3.499 | 7.128 | 0.05 | 0.04752 | m/day |
| Change in permeability | $c_k$ | $1 \cdot 10^{15}$ | $1 \cdot 10^{15}$ | 1.0 | 0.2 | - |
| ***Interfaces*** | | | | | | |
| Strength determination | - | Rigid | Rigid | Rigid | Rigid | - |
| Interface reduction factor | $R_{inter}$ | 1 | 1 | 1 | 1 | - |
| ***Initial*** | | | | | | |
| $K_0$ determination | - | Automatic | Automatic | Automatic | Automatic | - |
| Pre-overburden pressure | *POP* | 0 | 0 | 5 | 0 | $kN/m^2$ |
| Overconsolidation ratio | *OCR* | 1.0 | 1.0 | 1.0 | 1.0 | - |

To create the material sets, follow these steps:

1. Click the **Materials** button to open the **Material sets** window.
2. Create soil material data sets according to Table 20 (on page 128) and assign them to the corresponding layers in the borehole (see Figure 107 (on page 127)).
3. Close the **Modify soil layers** window and proceed to the **Structures mode** to define the embankment and drains.

**Note:** The initial void ratio ($e_{init}$) and the change in permeability ($c_k$) should be defined to enable the modelling of a change in the permeability in a consolidation analysis due to compression of the soil. This option is recommended when using advanced models.

# 8.4 Define the construction

The embankment and the drains are defined in the **Structures mode** .

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .

## 8.4.1 To define the embankment:

1. Click the **Create soil polygon** button in the side toolbar and select the **Create soil polygon** option.
2. Define the embankment in the drawing area by clicking on (0 0), (0 4), (8 4) and (20 0).
3. Select and right click the created polygon and assign the **Embankment** data set to the soil polygon as shown in Figure 108 (on page 130).

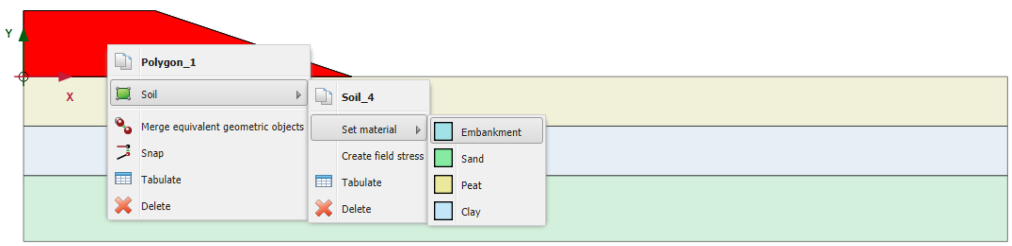



*Figure 108: Assignment of a material dataset to a soil cluster in the drawing area*

4. To define the embankment construction level click the **Cut polygon** button in the side toolbar and define a cutting line by clicking on (0 2) and (14 2).
   The embankment cluster is split into two sub-clusters.

## 8.4.2 To define the drains

In this project the effect of the drains on the consolidation time will be investigated by comparing the results with a case without drains. Drains will only be active for the calculation phases in the case with drains.

1. Click the **Create hydraulic conditions** button in the side toolbar and select the **Create drain** option in the appearing menu which is shown in Figure 109 (on page 130).

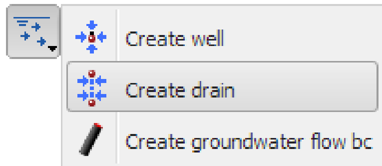



*Figure 109: Create Drain option*

2. Drains are defined in the soft layers (clay and peat; y = 0 m to y = -6 m). The distance between two consecutive drains is 2 m. Considering the symmetry, the first drain is located at 1 m distance from the model boundary. 10 drains will be created in total. The **head** is defined at 0.0 m. The geometry is shown in Figure 110 (on page 131).

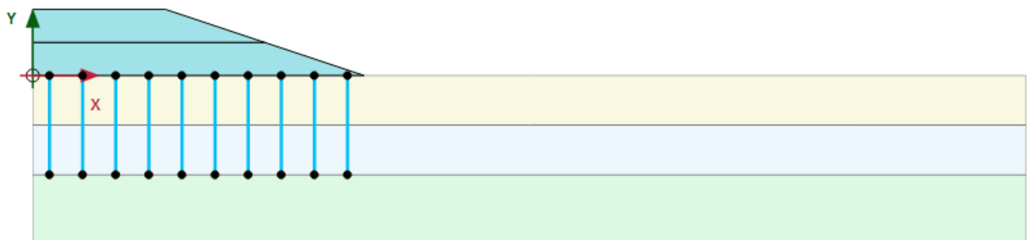



*Figure 110: Final geometry of the model*

**Note:**

The modelling of drains in a plane strain model actually involves the use of an equivalent (lateral) permeability in the surrounding soil based on the drain pattern. The latter has been omitted in this simplified example. More information can be in found in literature[1].

# 8.5 Generate the mesh

In order to generate the mesh, follow these steps:

1. Proceed to **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 111 (on page 132).

[1] Achtergronden bij numerieke modellering van geotechnische constructies, deel 2. CUR 191. Stichting CUR, Gouda Indraratna, B.N., Redana, I.W., Salim, W. (2000), Predicted and observed behaviour of soft clay foundations stabilised with vertical drains. Proc. GeoEng. 2000, Melbourne.

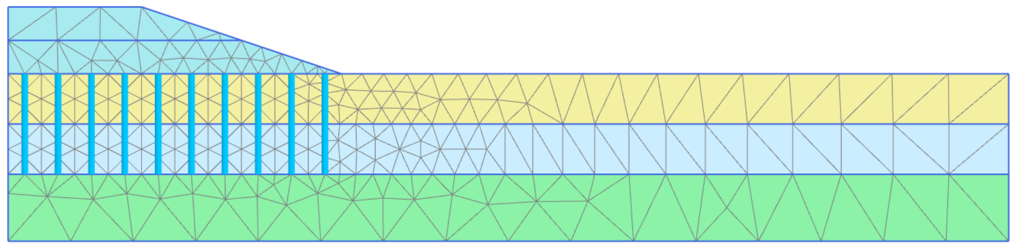


*Figure 111: The generated mesh*

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

# 8.6 Define and perform the calculation

The embankment construction is divided into two phases. After the first construction phase a consolidation period of 30 days is introduced to allow the excess pore pressures to dissipate. After the second construction phase another consolidation period is introduced from which the final settlements may be determined. Hence, a total of four calculation phases have to be defined besides the initial phase.

## 8.6.1 Initial phase: Initial conditions

In the initial situation the embankment is not present.

The remaining active geometry is horizontal with horizontal layers, so the **K0 procedure** can be used to calculate the initial stresses. The geometry of the model for initial phase is shown in Figure 112 (on page 132).

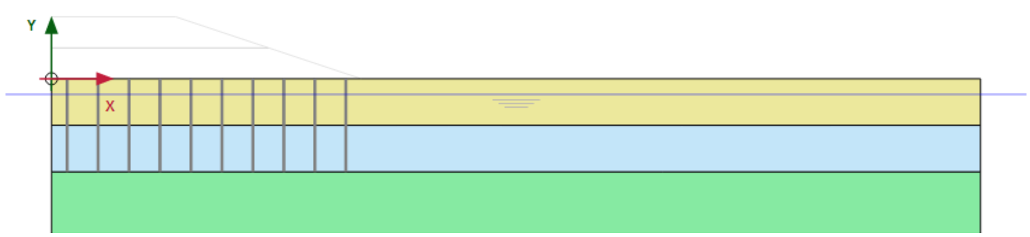



*Figure 112: Configuration of the initial phase*

The initial water pressures are fully hydrostatic and based on a general phreatic level located at y = -1 m. Note that a phreatic level is automatically created at y = -1 m, according to the value specified for **Head** in the borehole. In addition to the phreatic level, attention must be paid to the boundary conditions for the consolidation analysis that will be performed during the calculation process. Without giving any additional input, all boundaries except for the bottom boundary are draining so that water can freely flow out of these boundaries and excess pore pressures can dissipate. In the current situation, however, the left vertical boundary must be closed because this is a line of symmetry, so horizontal flow should not occur. The remaining boundaries are open because the excess pore pressures can be dissipated through these boundaries. In order to define the appropriate consolidation boundary conditions, follow these steps:

1. Go to the **Staged construction mode** and in the **Model explorer** expand the **Model conditions** subtree shown in Figure 113 (on page 133).
2. Expand the **GroundwaterFlow** subtree and set **BoundaryXMin** to **Closed** and **BoundaryYMin** to **Open**.

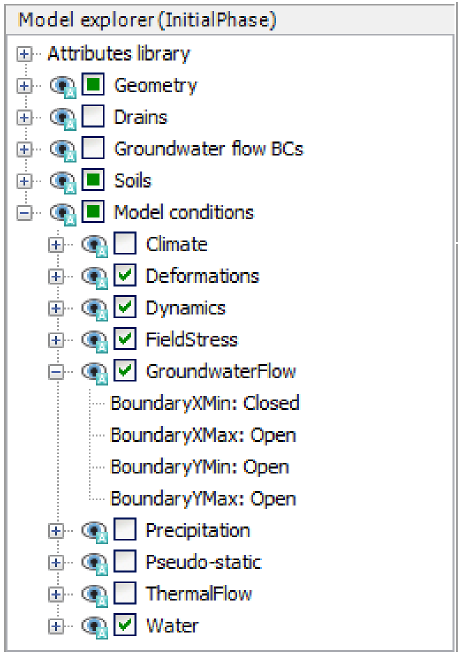



*Figure 113: The boundary conditions of the problem*

## 8.6.2 Consolidation analysis

A consolidation analysis introduces the dimension of time in the calculations. In order to correctly perform a consolidation analysis a proper time step must be selected. The use of time steps that are smaller than a critical minimum value can result in stress oscillations.

The consolidation option in PLAXIS 2D allows for a fully automatic time stepping procedure that takes this critical time step into account. Within this procedure there are three main possibilities:

| | |
|---|---|
|  | Consolidate for a predefined period, including the effects of changes to the active geometry (**Staged construction**). |
|  | Consolidate until all excess pore pressures in the geometry have reduced to a predefined minimum value (**Minimum excess pore pressure**). |
|  | Consolidate until a specified degree of saturation is reached (**Degree of consolidation**). |

The first two possibilities will be used in this exercise. To define the calculation phases, follow these steps:

## Phase 1: First embankment construction

The first calculation stage is a **Consolidation** analysis, **Staged construction**.

1. Click the **Add phase** button to create a new phase and double click.
2. In the **Phases** window go to the **General** subtree and from the **Calculation type** drop-down menu select the **Consolidation** option.
3. Make sure that for the **Loading type** the **Staged construction** option is selected .
4. Enter a **Time interval** of 2 days. The default values of the remaining parameters will be used.
5. In the **Staged construction mode** activate the first part of the embankment. The model for phase 1 is shown in Figure 114 (on page 134).

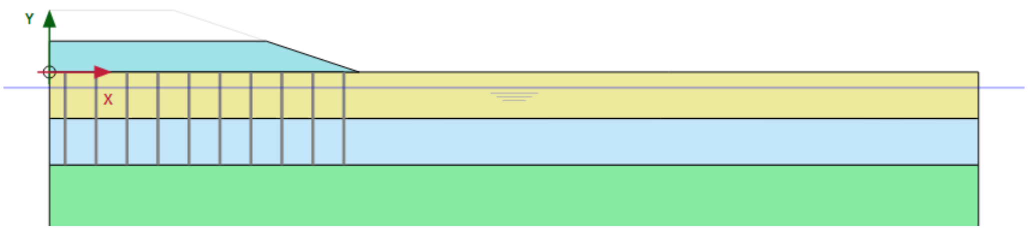



*Figure 114: Configuration of the phase 1*

## Phase 2: First consolidation period

The second phase is also a **Consolidation** analysis, **Staged construction**. In this phase no changes to the geometry are made as only a consolidation analysis to ultimate time is required.

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window go to the **General** subtree and from the **Calculation type** drop-down menu select the **Consolidation** option.
3. For the **Loading type** make sure that the **Staged construction** option is selected.
4. Enter a **Time interval** of 30 days. The default values of the remaining parameters will be used.

## Phase 3: Second embankment construction

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window go to the **General** subtree and from the **Calculation type** drop-down menu select the **Consolidation** option.
3. For the **Loading type** make sure that the **Staged construction** option is selected.
4. Enter a **Time interval** of 1 day. The default values of the remaining parameters will be used.
5. In the **Staged construction mode** activate the second part of the embankment. The model for phase 3 is shown in Figure 115 (on page 135).

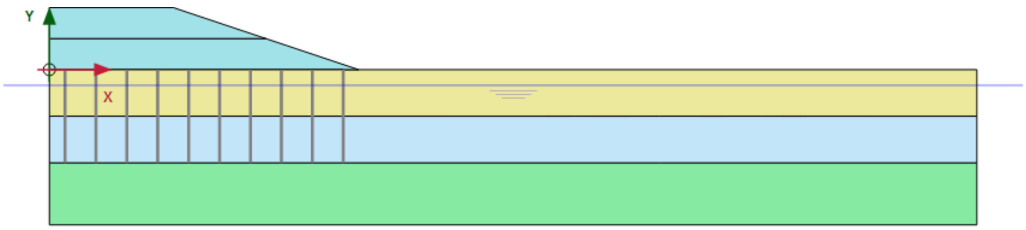



*Figure 115: Configuration of the phase 3*

### Phase 4: End of consolidation

The fourth phase is a **Consolidation** analysis to a minimum excess pore pressure.

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window go to the **General** subtree and from the **Calculation type** drop-down menu select the **Consolidation** option.
3. In the **Loading type** drop-down menu select the **Minimum excess pore pressure** option and accept the default value of 1 kN/m$^2$ for the minimum pressure. The default values of the remaining parameters will be used.

## 8.6.3 Safety analysis

In the design of an embankment it is important to consider not only the final stability, but also the stability during the construction. It is clear from the output results that a failure mechanism starts to develop after the second construction phase.

It is interesting to evaluate a global safety factor at this stage of the problem, and also for other stages of construction.

To calculate the global safety factor for the road embankment at different stages of construction, follow these steps:

1. Select Phase 1 in the **Phases** explorer.
2. Add a new calculation phase.
3. Double-click on the new phase to open the **Phases** window.
4. In the **Phases** window the selected phase is automatically selected in the **Start from phase** drop-down menu.
5. In the **General** subtree, select **Safety** as calculation type.
6. The **Incremental multipliers** option is already selected in the **Loading input** box. The first increment of the multiplier that controls the strength reduction process, **Msf**, is set to 0.1.
7. In order to exclude existing deformations from the resulting failure mechanism, select the **Reset displacements to zero** option in the **Deformation control parameters** subtree.
8. The first safety calculation has now been defined.

9. Follow the same steps to create new calculation phases that analyse the stability at the end of each consolidation phase. The various phases after defining safety calculation is shown in Figure 116 (on page 136).

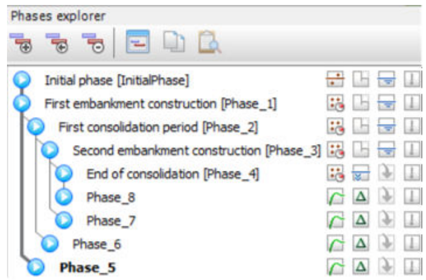



*Figure 116: Safety calculation phases*

## 8.6.4 Calculate

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow the steps given below.

1. Click the **Select points for curves** button in the side toolbar.
2. As the first point, select the toe of the embankment at (20 0).
3. The second point will be used to plot the development (and decay) of excess pore pressures. To this end, a point somewhere in the middle of the soft soil layers at the left side of the model is needed, hence underneath the middle of the embankment. Select for instance a precalculation point at (0 -3).
4. Click the **update** option.
5. Click the **Calculate** button to calculate the project.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window as shown in Figure 117 (on page 137).

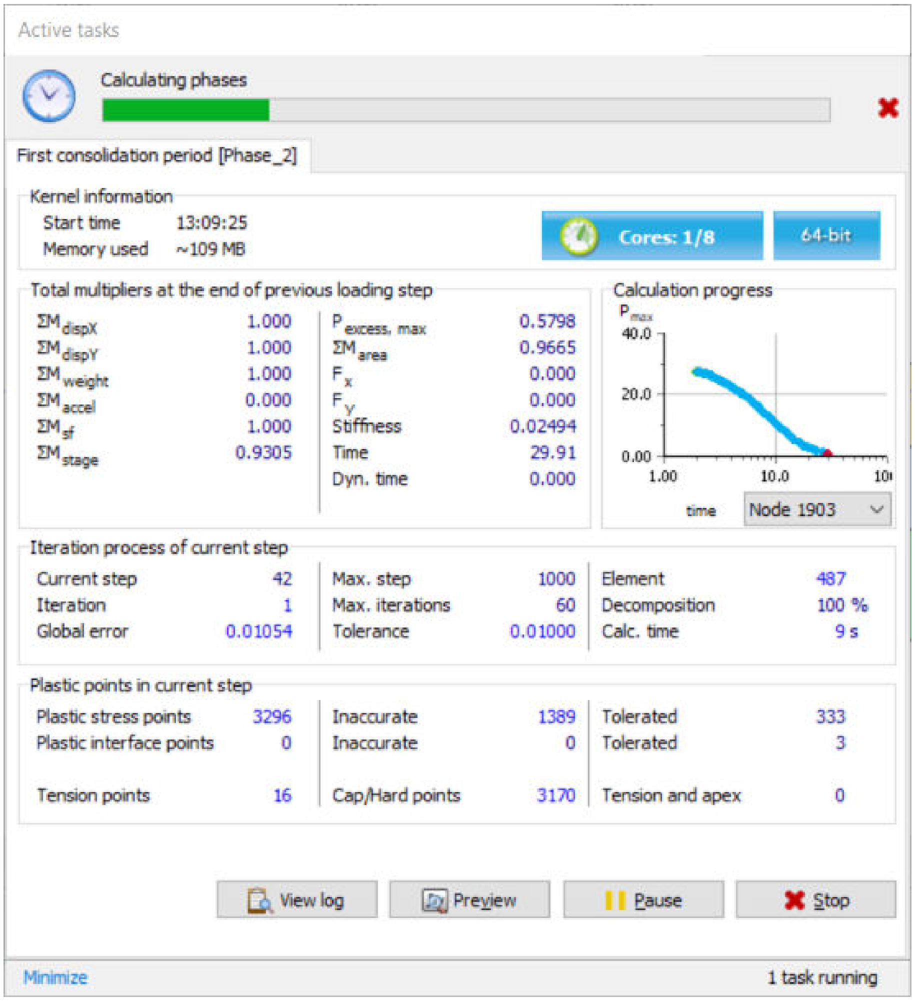



*Figure 117: Calculation progress displayed in the **Active tasks** window*

In addition to the multipliers, a parameter $P_{excess,max}$ occurs, which indicates the current maximum excess pore pressure. This parameter is of interest in the case of a **Minimum excess pore pressure** consolidation analysis, where all pore pressures are specified to reduce below a predefined value.

# 8.7 Results

## 8.7.1 Deformed mesh

After the calculation has finished, select the third phase and click the **View calculation results** button .

The **Output** window now shows the deformed mesh after the undrained construction of the final part of the embankment. Considering the results of the third phase, the deformed mesh as displayed in Figure 118 (on page 138) shows the uplift of the embankment toe and hinterland due to the undrained behaviour.

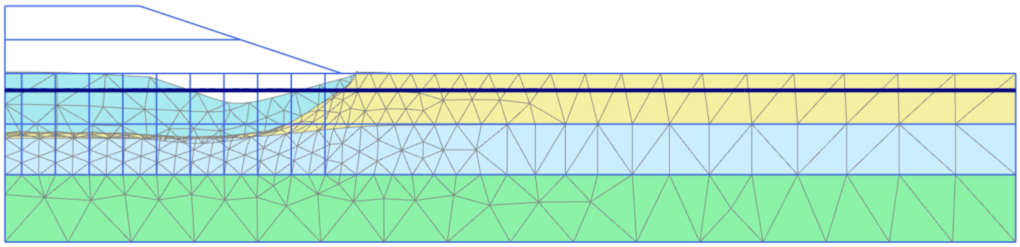


*Figure 118: Deformed mesh after undrained construction of embankment (Phase 3)*

## 8.7.2 Incremental displacements

1. Select the menu **Deformations** > **Incremental displacements** > **|Δ u|**.
2. Select the menu **View** > **Arrows** option in the menu or click the corresponding button in the toolbar to display the results arrows.

On evaluating the total displacement increments, it can be seen that a failure mechanism is developing shown in Figure 119 (on page 138):

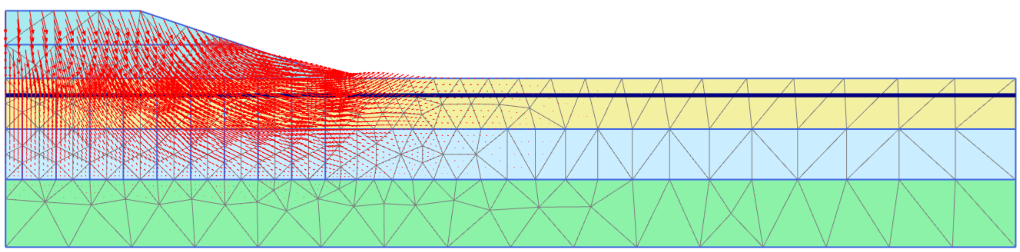


*Figure 119: Displacement increments after undrained construction of embankment*

## 8.7.3 Excess pore pressures

1. Press **<Ctrl + 7>** to display the developed excess pore pressures (see Appendix C of the Reference Manual for more shortcuts). They can also be displayed by selecting the menu **Stresses** > **Pore pressures** > **Pexcess**.
2. Click the **Center principal directions** button . The principal directions of excess pressures are displayed at the center of each soil element. The results are displayed in Figure 120 (on page 139).

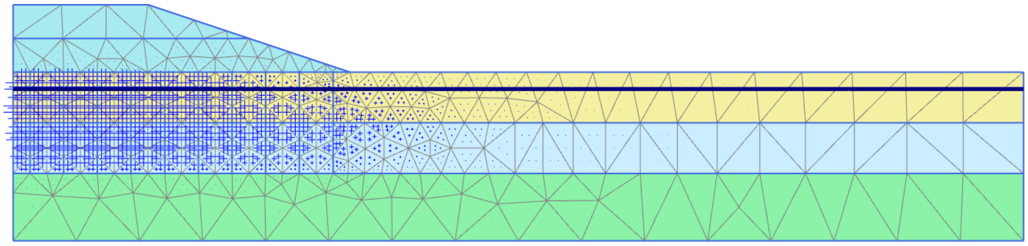


*Figure 120: Excess pore pressures after undrained construction of embankment*

It is clear that the highest excess pore pressure occurs under the embankment centre.

3. Select **Phase 4** in the drop down menu.
4. Click the **Contour lines** button in the toolbar to display the results as contours.
5. To show the **labels** of the contour lines on the soil profile click on the **Draw scanline** button (or the corresponding option in the **View** menu). Then on the geometry make a line by clicking on an initial point and dragging on the contours to be identified (Figure 121 (on page 139)).

By exploring the different phases, it can be seen that the settlement of the original soil surface and the embankment increases considerably during the **Phase 4**. This is due to the dissipation of the excess pore pressures (= consolidation), which causes further settlement of the soil. Figure 121 (on page 139) shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below 1.0 kN/m$^2$.

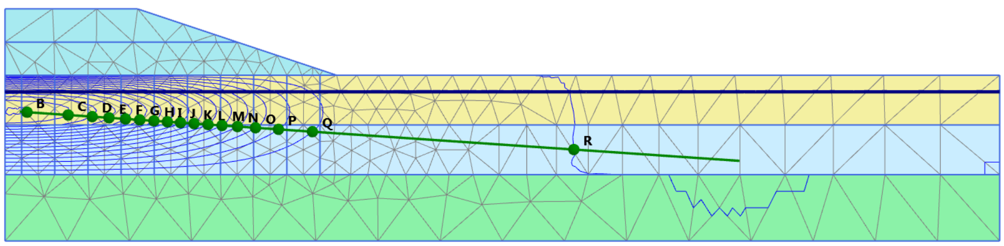



*Figure 121: Excess pore pressure contours after consolidation to $P_{excess}$ < 1.0 kN/m$^2$*

## 8.7.4 Development of excess pore pressure

The **Curves manager** can be used to view the development, with time, of the excess pore pressure under the embankment. In order to create such a curve, follow these steps:

1. In the **Curves manager** create a new curve by clicking the button .
2. For the x-axis, select the **Project** option from the drop-down menu and select **Time** in the tree.
3. For the y-axis select the point in the middle of the soft soil layers (defined as second pre-calculation point (on page 136)) from the drop-down menu. In the tree select **Stresses** > **Pore pressure** > **pexcess**.
4. Select the **Invert sign** option for the y-axis.
5. Click **OK**.
6. Open the **Curve settings** (F3) and go to the second tabsheet.
7. In the **Show** box click the **Phases** button. By default all phases are selected to show in the curve. For the clarity of the curve, hide the **Safety** phases (phases 5 - 8).

**8.** Click **OK** to close the **Curve settings** window.

A curve similar to the following one should appear as shown in Figure 122 (on page 140):

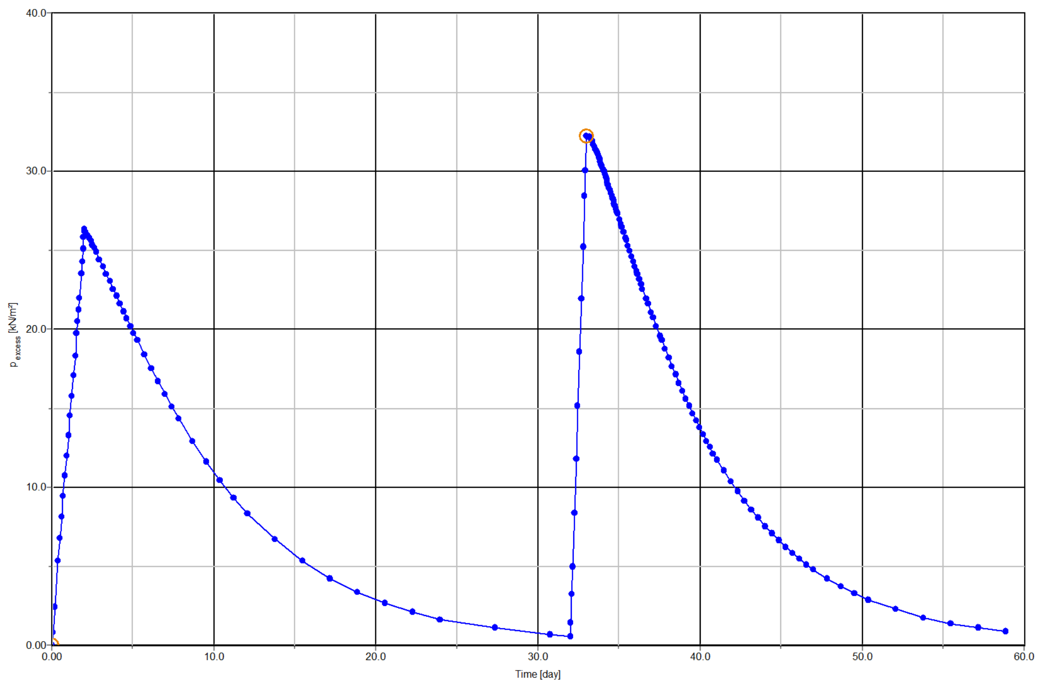



*Figure 122: Development of excess pore pressure under the embankment*

The figure clearly shows the four calculation phases. During the construction phases the excess pore pressure increases with a small increase in time while during the consolidation periods the excess pore pressure decreases with time. In fact, consolidation already occurs during construction of the embankment, as this involves a small time interval. From the curve it can be seen that more than 50 days are needed to reach full consolidation.

Save the chart before closing the Output program.

## 8.7.5 Safety analysis results

Additional displacements are generated during a **Safety** calculation. The total displacements do not have a physical meaning, but the incremental displacements in the final step (at failure) give an indication of the likely failure mechanism.

In order to view the mechanisms in the three different stages of the embankment construction:

1. Select one of these phases and click the **View calculation results** button.
2. Select the menu **Deformations** > **Incremental displacements** > **|Δu|**.

3. Change the presentation from **Arrows** to **Shadings** . The resulting plots shown in Figure 123 (on page 141) gives a good impression of the failure mechanisms. The magnitude of the displacement increments is not relevant.

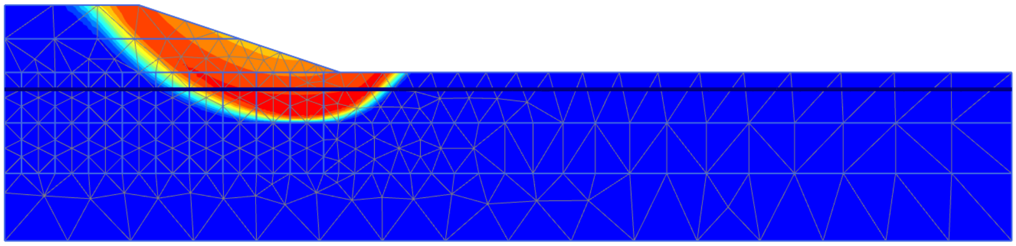


*Figure 123: Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage*

The safety factor can be obtained from the **Calculation info** option of the **Project** menu. The **Multipliers** tabsheet of the **Calculation information** window represents the actual values of the load multipliers. The value of ΣMsf represents the safety factor, provided that this value is indeed more or less constant during the previous few steps.

The best way to evaluate the safety factor, however, is to plot a curve in which the parameter ΣMsf is plotted against the displacements of a certain node. Although the displacements are not relevant, they indicate whether or not a failure mechanism has developed.

In order to evaluate the safety factors for the three situations in this way, follow these steps:

1. Click the **Curves manager** button in the toolbar.
2. Click **New** in the **Charts** tabsheet.
3. In the **Curve generation** window, select the embankment toe (Point A) for the x-axis. Select **Deformations** > **Total displacements** > **|u|**.
4. For the y-axis, select **Project** > **Multipliers** > **ΣMsf**. The **Safety** phases are considered in the chart.
5. Right-click on the chart and select the **Settings** option in the appearing menu. The **Settings** window pops up.
6. In the tabsheet corresponding to the curve click the **Phases** button.
7. In the **Select phases** window select Phase 5 shown in Figure 124 (on page 142):

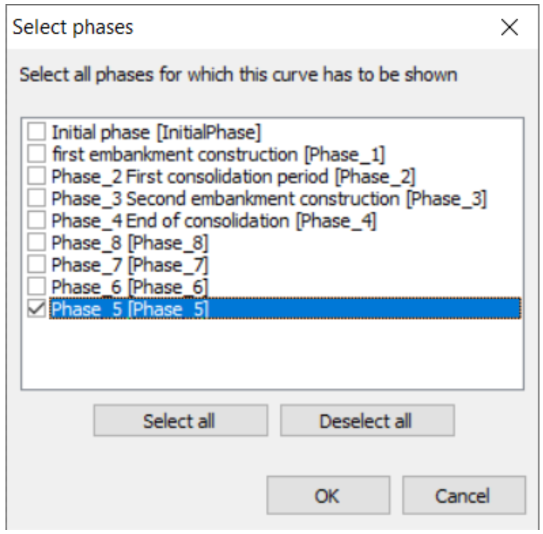



*Figure 124: The Select phases window*

8. Click **OK** to close the **Select phases** window.
9. In the **Settings** window change the titles of the curve in the corresponding tabsheet.
10. Click the **Add curve** button and select the **From current project...** option in the appearing menu. Define curves for phases 6, 7 and 8 by following the described steps.
11. In the **Settings** window click the **Chart** tab to open the corresponding tabsheet.
12. In the **Chart** tabsheet specify the chart name.
13. Set the scaling of the x-axis to **Manual** and set the value of **Maximum** to 1 as shown in Figure 125 (on page 143):

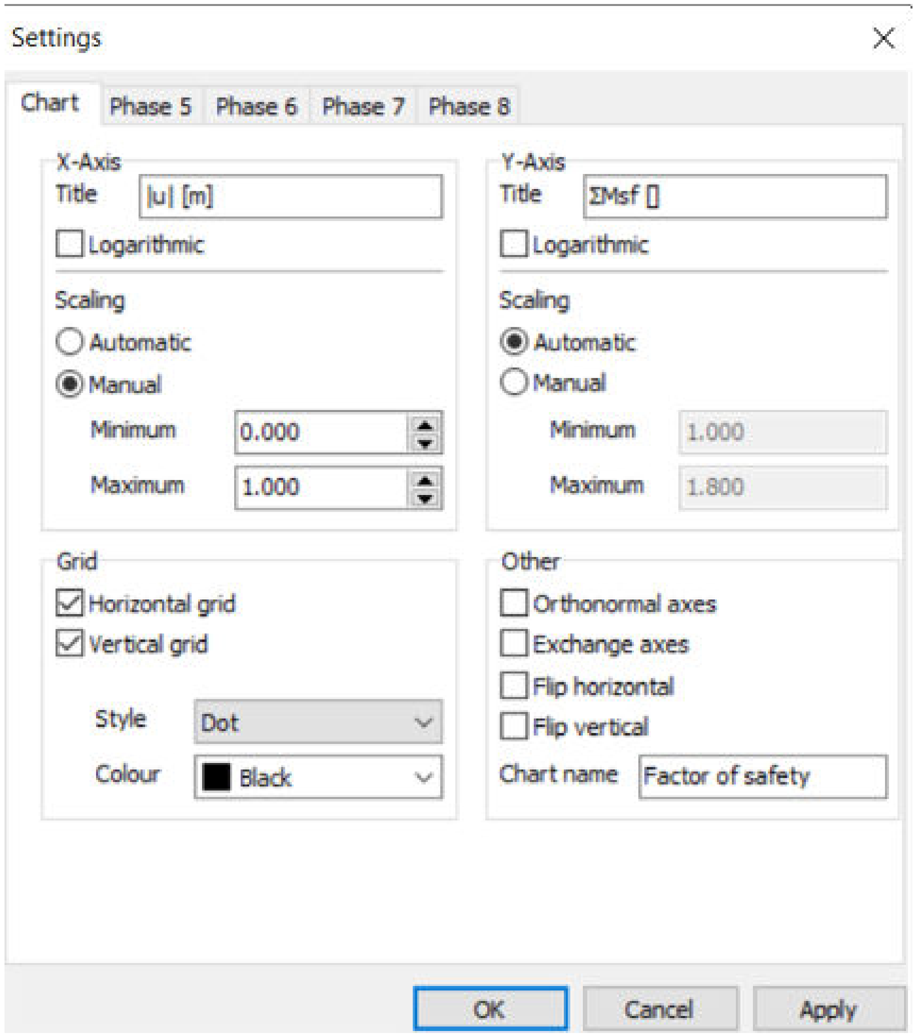



*Figure 125: The Chart tabsheet in the Settings window*

14. Click **Apply** to update the chart according to the changes made and click **OK** to close the **Settings** window.
15. To modify the location of the legend right-click on the legend.
16. In the context menu select **View** > **Legend in chart**.
17. The legend can be relocated in the chart by dragging it. The plot is shown in Figure 126 (on page 144):

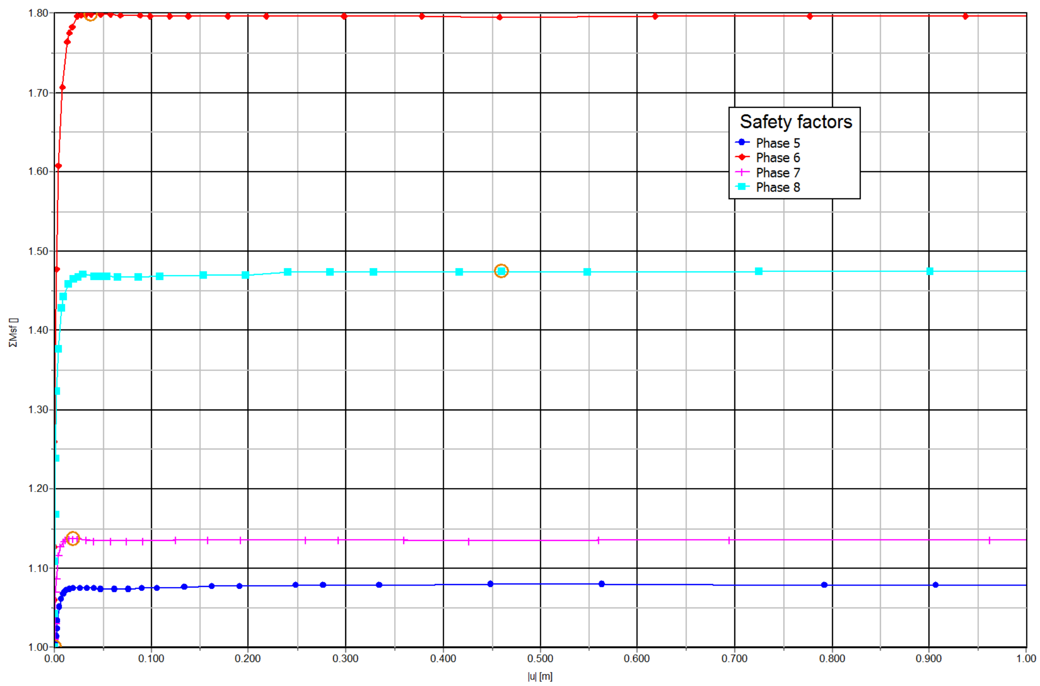



*Figure 126: Evaluation of safety factor*

The maximum displacements plotted are not relevant. It can be seen that for all curves a more or less constant value of ΣMsf is obtained. Hovering the mouse cursor over a point on the curves, a box showing the exact value of ΣMsf can be obtained.

## 8.8 Using drains

In this section the effect of the drains in the project will be investigated. Four new phases will be introduced having the same properties as the first four consolidation phases. The first of these new phases should start from the initial phase. The differences in the new phases are:

- The drains should be active in all the new phases. Activate them in the **Staged construction mode** .
- The **Time interval** in the first three of the consolidation phases (9 to 11) is 1 day. The last phase is set to **Minimum excess pore pressure** and a value of 1.0 kN/m$^2$ is assigned to the minimum pressure (|P-stop|).

Follow these steps:

1. After the calculation is finished, save the project, then select the last phase and click the **View calculation results** button. The **Output** window now shows the deformed mesh after the drained construction of the final part of the embankment. In order to compare the effect of the drains, the excess pore pressure dissipation in the second point can be used.

2. Click the **Curves manager** button to open the **Curves manager**.
3. In the **Chart** tabsheet double-click Chart 1 ($p_{excess}$ of the second point at (0 -3) versus time). The chart is displayed. Close the **Curves manager**.
4. Double-click the curve in the legend at the right of the chart. The **Settings** window pops up.
5. Click the **Add curve** button and select the **From current project ...** option in the appearing menu. The **Curve generation** window pops up.
6. Select the **Invert sign** option for y-axis and click **OK** to accept the selected options.
7. In the chart a new curve is added and a new tabsheet corresponding to it is opened in the **Settings** window. Click the **Phases** button. From the displayed window select the **Initial phase** and the last four phases (drains) and click **OK**.
8. In the **Settings** window change the titles of the curves in the corresponding tabsheets.
9. In the **Chart** tabsheet specify the chart name.
10. Click **Apply** to preview the generated curve and click **OK** to close the **Settings** window. The chart gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate as shown in Figure 127 (on page 145):

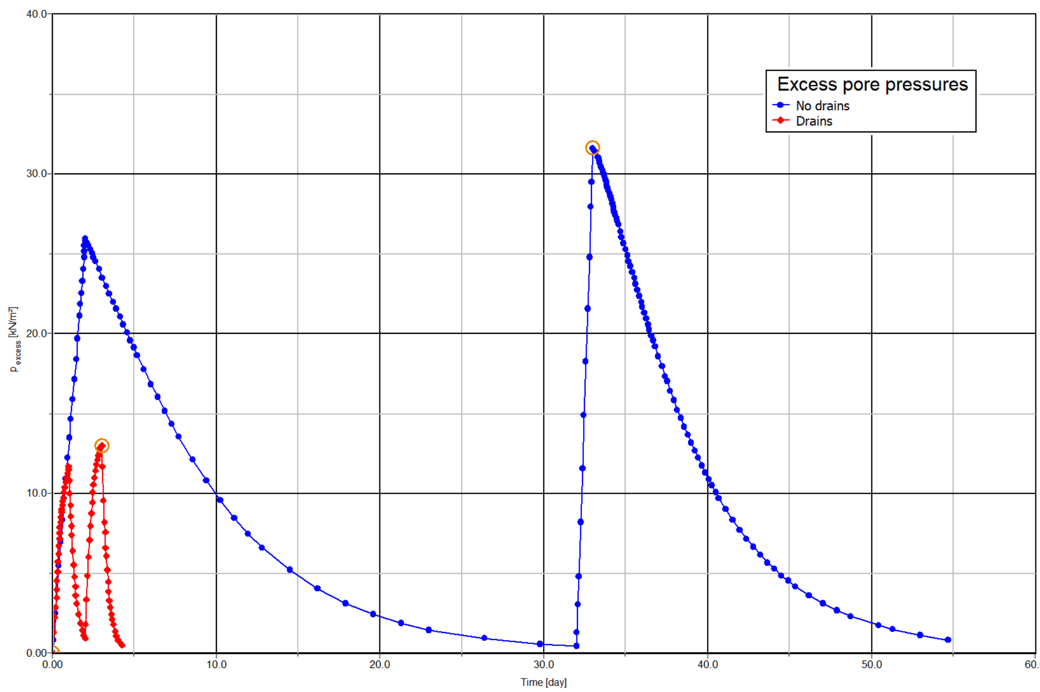



*Figure 127: Effect of drains*

**Note:**

Instead of adding a new curve, the existing curve can be regenerated using the corresponding button in the **Curves settings** window.

## 8.9 Updated mesh and updated water pressures analysis

As can be seen from the output of the **Deformed mesh** at the end of consolidation (stage 4), the embankment settles about one meter since the start of construction. Part of the sand fill that was originally above the phreatic level will settle below the phreatic level.

As a result of buoyancy forces the effective weight of the soil that settles below the water level will change, which leads to a reduction of the effective overburden in time. This effect can be simulated in PLAXIS 2D using the **Updated mesh** and **Updated water pressures** options. For the road embankment the effect of using these options will be investigated.

1. Select the initial phase in the **Phases** explorer.
2. Add a new calculation phase.
3. Define the new phase in the same way as Phase 1. In the **Deformation control parameters** subtree check the **Updated mesh** and **Updated water pressures** options.
4. Define the other 3 phases in the same way.

When the calculation has finished, compare the settlements for the two different calculation methods.

1. In the **Curve generation** window select time for the x-axis and select the vertical displacement ($u_y$) of the point in the middle of the soft soil layers at (0 -3) for the y-axis.
2. In this curve the results for Initial phase and phases from 1 to 4 will be considered.
3. Add a new curve to the chart.
4. In this curve the results for Initial phase and phases from 13 to 16 will be considered. The resulting chart is shown in Figure 128 (on page 147).

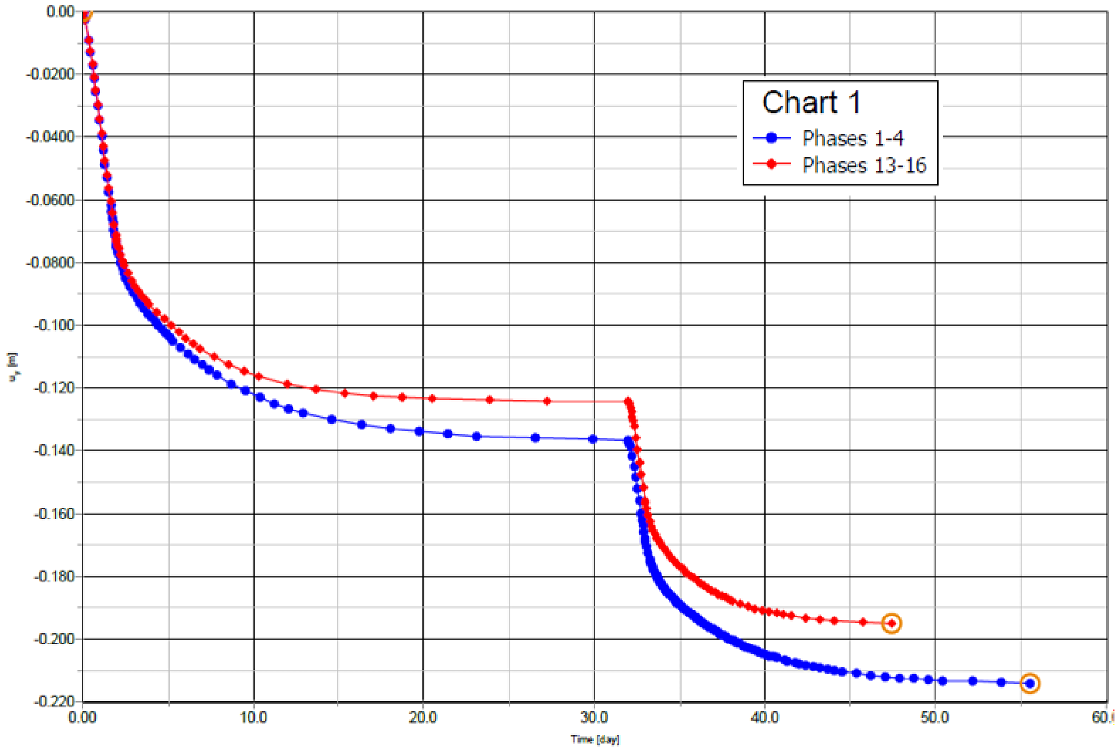



*Figure 128: Effect of updated mesh and water pressures analysis on resulting settlements*

It can be seen that the settlements are less when the **Updated mesh** and **Updated water pressures** options are used (red curve). This is partly because the **Updated mesh** procedure includes second order deformation effects by which changes of the geometry are taken into account, and partly because the **Updated water pressures** procedure results in smaller effective weights of the embankment. This last effect is caused by the buoyancy of the soil settling below the (constant) phreatic level. The use of these procedures allows for a realistic analysis of settlements, taking into account the positive effects of large deformations.

# 9 Excavation and dewatering [ADV]

In this tutorial lowering the groundwater level and the flow around a sheetpile wall will be analysed. The geometry model of the tutorial Dry excavation using a tie back wall [ADV] (on page 102) will be used . The **Well** feature is introduced in this example.

## 9.1 Create and assign material data set

The material parameters remain unchanged from the original project as shown in Table 14 (on page 104).

To create the project:

1. Open the project defined in the tutorial Dry excavation using a tie back wall [ADV] (on page 102).
2. Save the project under a different name (e.g. 'Flow around a sheet pile wall').

## 9.2 Define the structural elements

1. In the **Structures mode** click the **Create hydraulic conditions** button in the side toolbar.
2. Select the **Create well** option in the appearing menu.
3. Draw the first well by clicking on (42 20) and (42 17).
4. Draw the second well by clicking on (58 20) and (58 17).

## 9.3 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Select the cluster and two wells as shown in the Figure 129 (on page 149) . In **Selection Explorer** specify a **Coarseness factor** of 0.25.

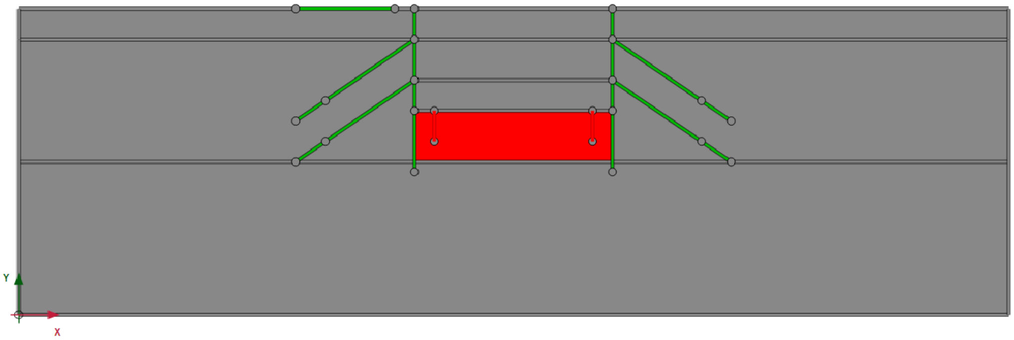


*Figure 129: Indication of the local refinement of the mesh in the model*

3. Click the **Generate mesh** button to generate the mesh. Use the default option for the **Element distribution** parameter (Medium).
4. Click the **View mesh** button to view the mesh as shown in Figure 130 (on page 149).

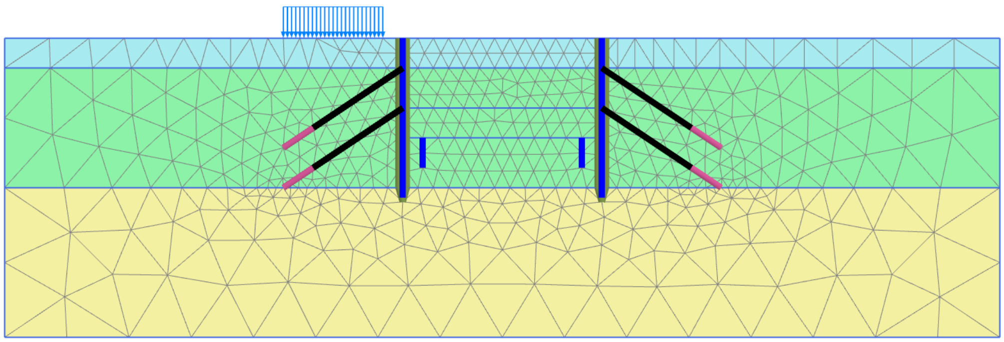


*Figure 130: The generated mesh*

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

## 9.4 Define and perform the calculation

- Proceed to the **Staged construction mode** . The groundwater flow analysis performed in phase 6 has to be modified.

### 9.4.1 Phase 6: Dewatering

In this phase the wells will be used to lower the phreatic level in the excavation down to y = 17m. This corresponds to 3m below the final excavation level.

1. Multi-select the wells in the model and activate them.

2. In the **Selection explorer** the behaviour of the wells is by default set to **Extraction**.
3. Set the discharge value to 1.5m$^3$/day/m.
4. Set the $h_{min}$ value to 17m. This means that water will be extracted as long as the groundwater head at the wall location is at least 17m.

   Figure 131 (on page 150) shows the parameters assigned to the wells in the **Selection explorer**.

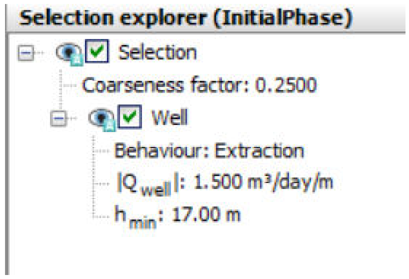



*Figure 131: Well properties*

## 9.4.2 Execute the calculation

The definition of the calculation process is complete.

1. Click the **Calculate** button to calculate the project.
2. Save the project after the calculation has finished.

# 9.5 Results

To display the flow field:

1. Select the Phase 6 in the drop down menu.
2. Click the menu **Stresses** > **Groundwater flow** > **|q|**.

   A scaled representation of the results (scale factor = 5.0 ) is shown in Figure 132 (on page 151):

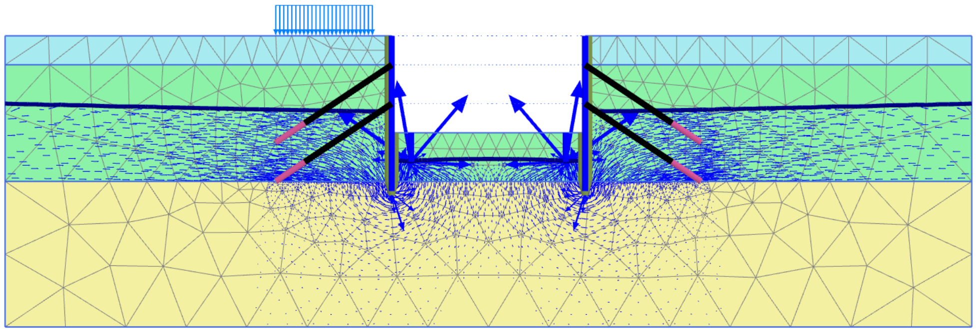


*Figure 132: The resulting flow field at the end of Phase 6*

Click the menu **Stresses** > **Pore pressures** > **Groundwater head.** Compare the results with the ones of the Phase 6 of the project defined in the tutorial Dry excavation using a tie back wall [ADV] (on page 102).

In Figure 133 (on page 151), the resulting groundwater head with and without the wells are displayed .

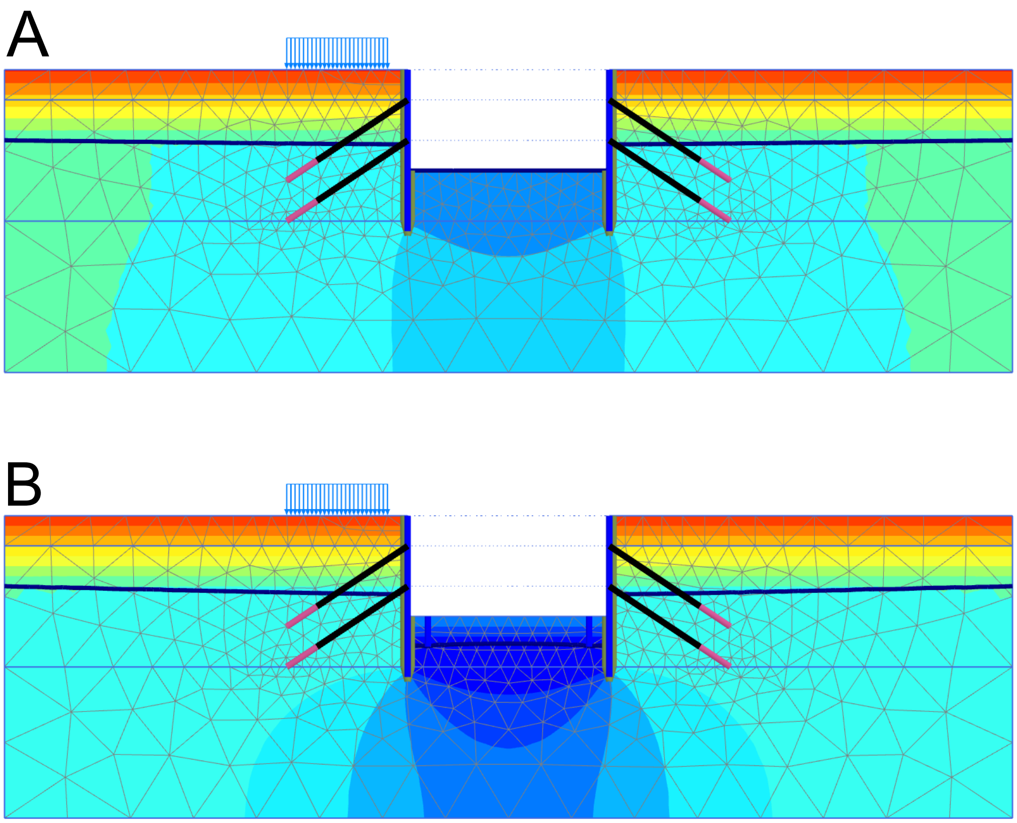



*Figure 133: Comparison of the resulting groundwater head*

A: Groundwater head (Phase 6 in the tutorial Dry excavation using a tie back wall [ADV] (on page 102))

B: Groundwater head (Phase 6 in the current project)

# 10

# Cyclic vertical capacity and stiffness of circular underwater footing [ADV]

This tutorial illustrates how to calculate the vertical bearing capacity and vertical stiffness of a circular stiff underwater footing (e.g. one of the footings of a jacket structure) exposed to cyclic loading during a storm. The storm is idealised by a distribution of load parcels with different magnitude. The cyclic accumulation tool is used to obtain soil parameters for the UDCAM-S model . The example considers a circular concrete footing with a radius of 11 m, placed on an over-consolidated clay layer.

The procedure for establishing non-linear stress-strain relationships and calculating load-displacement curves of a foundation under a cyclic vertical load component is presented. The analysis of the circular footing is performed with a 2D axisymmetric model. The soil profile consists of clay with an overconsolidation ratio, OCR, of 4, submerged unit weight of 10 kN/m$^3$ and an earth pressure coefficient, $K_0$ of 1. The (static) undrained shear strength from anisotropically consolidated triaxial compression tests has a constant value with a depth of $s_u{}^C$ = 130 kPa. The maximum shear modulus, $G_{max}$, of the clay is 67275 kPa. The cyclic behaviour of the soil is based on contour diagrams for Drammen clay (Andersen, Kleven & Heien, 1988 [2]) assuming that the behaviour is representative of the actual clay.

## 10.1 Objectives

- Obtain the UDCAM-S model input parameters by running the cyclic accumulation procedure, determining the stress-strain curves and optimising the material model parameters.
- Calculate the total cyclic vertical bearing capacity.
- Calculate the vertical stiffness accounting for cyclic loading for both the total and the cyclic component.

## 10.2 Geometry

The soil properties and footing geometry are shown in Figure 134 (on page 153) .

[2] Andersen, K.H., Kleven, A., Heien, .D. (1988). Cyclic soil data for design of gravity structures. Journal of Geotechnical Engineering, 517–539.

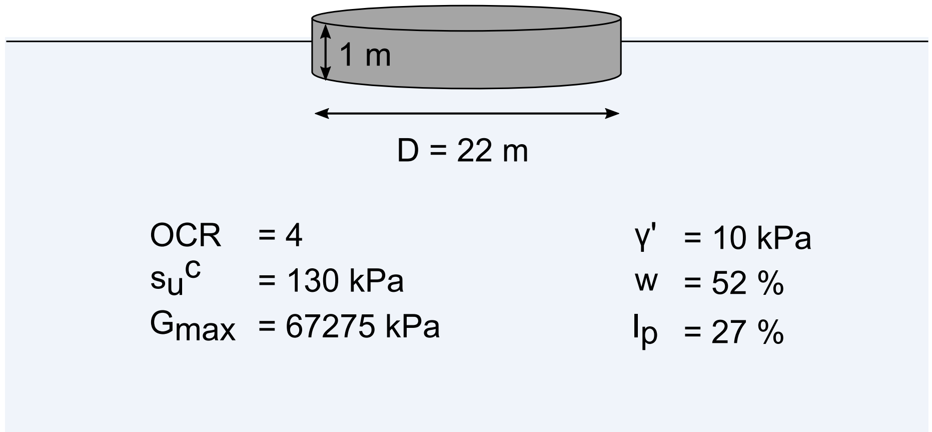



*Figure 134: Geometry of the project*

## 10.3 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 make sure that
   a. **Model** is set to **Axisymmetry** and
   b. **Elements** is set to **15-Noded**.
4. Define the limits for the model contour as
   a. $x_{min}$ = 0.0 m, $x_{max}$ = 40.0 m
   b. $y_{min}$ = -30.0 m and $y_{max}$ = 0.0 m

## 10.4 Define the soil stratigraphy

The sub-soil layers are defined using a borehole.

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Create a single soil layer with top level at 0.0 m and bottom level at -30.0m.
3. For simplicity, water is not taken into account in this example. The groundwater table is therefore set below the bottom of the model, and the soil weight is based on the effective (underwater) weight.
4. In the borehole column specify a value of -50.00 for **Head**.

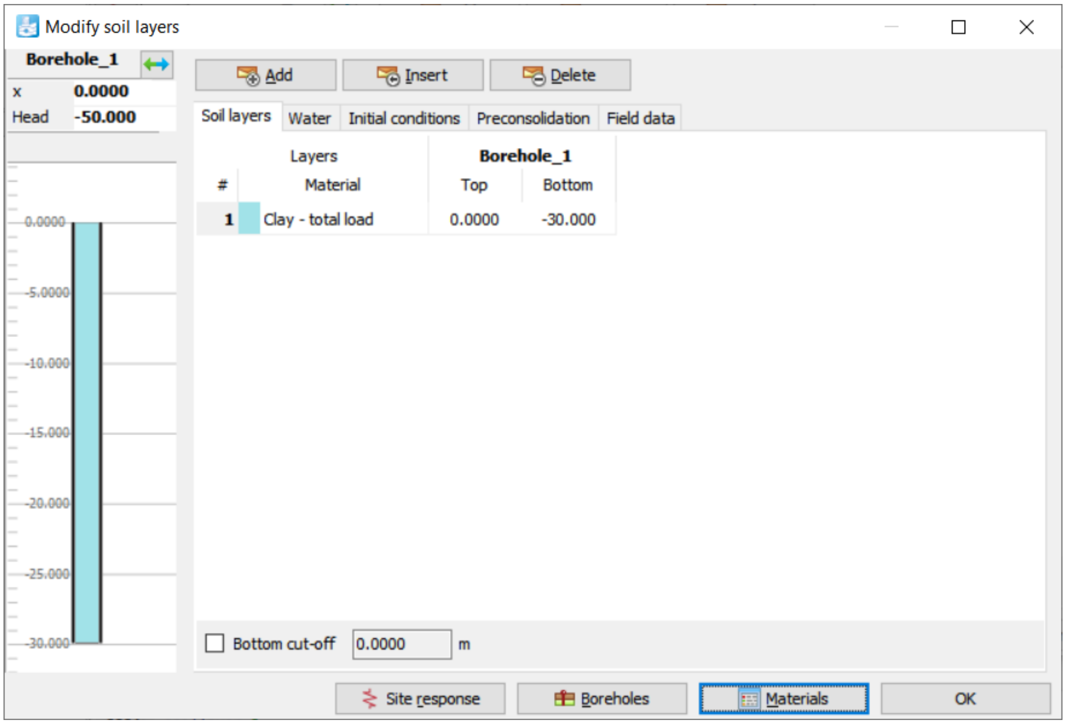



*Figure 135: Soil layer*

## 10.5 Create and assign material data sets

Three material data sets need to be created; two for the clay layer (Clay - total load and Clay - cyclic load) and one for the concrete foundation.

Open the **Material sets** window.

### 10.5.1 Material: Clay - total load

The model parameters for this material will be determined by the cyclic accumulation and optimisation tool.

The other properties are as shown in Table 21 (on page 154) :

**Table 21: Material properties**

| Parameter | Name | Clay - total load | Unit |
|---|---|---|---|
| **General** | | | |
| Identification | - | Clay - total load | - |

| Parameter | Name | Clay - total load | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | - | UDCAM-S model | - |
| Drainage type | - | Undrained (C) | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 10 | kN/m$^3$ |

To create the material set, follow these steps:

1. Choose **Soil and interfaces** as the **Set type** and click the **New** button.
2. On the **General** tab enter the values according to Table 21 (on page 154).
3. Proceed to the **Mechanical** tab.

Instead of entering the model parameters in this tab sheet, we will run the cyclic accumulation and optimisation tool. This procedure consists of three steps.

Click the **Mechanical** tab and click the **Cyclic accumulation and optimisation tool** option in the side window as shown in Figure 136 (on page 155).

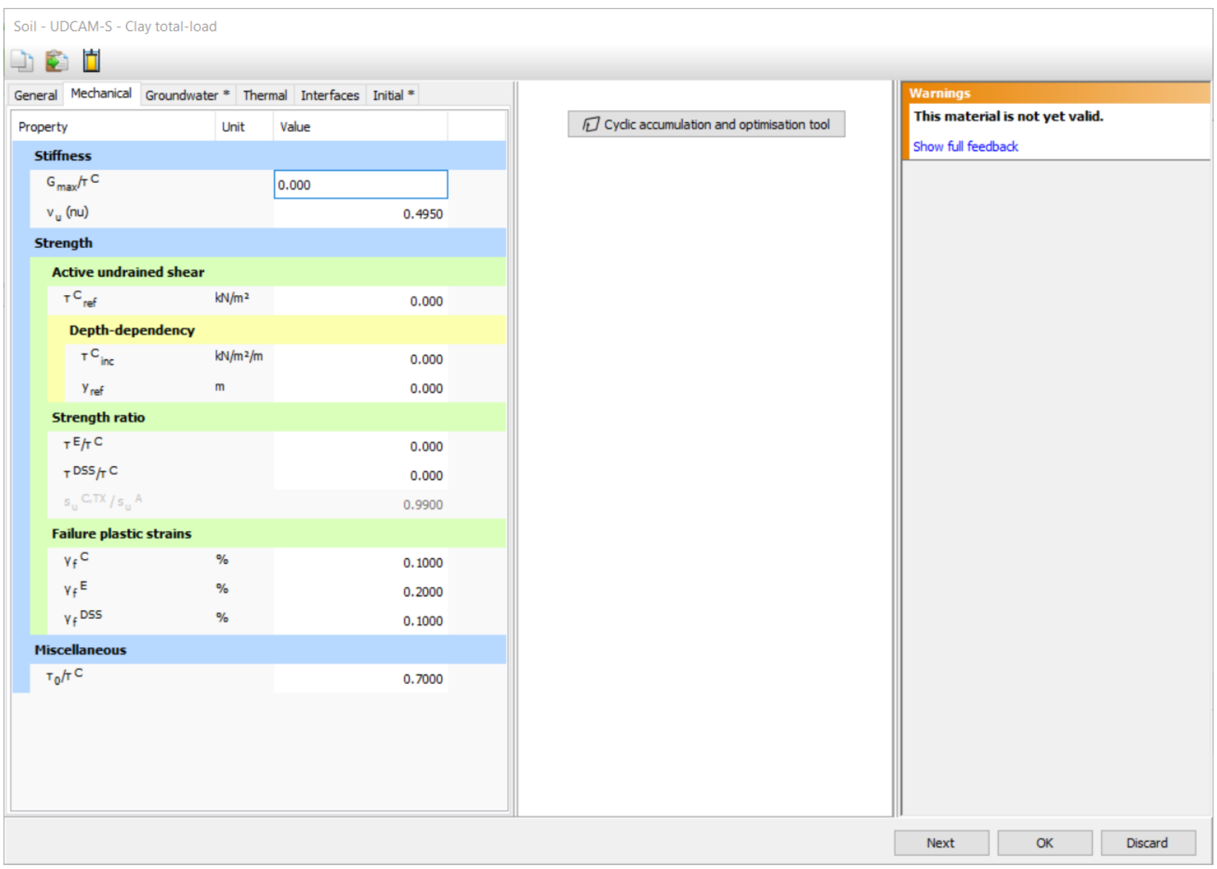



*Figure 136: Cyclic accumulation and optimisation tool*

A new window opens shown in Figure 137 (on page 156):

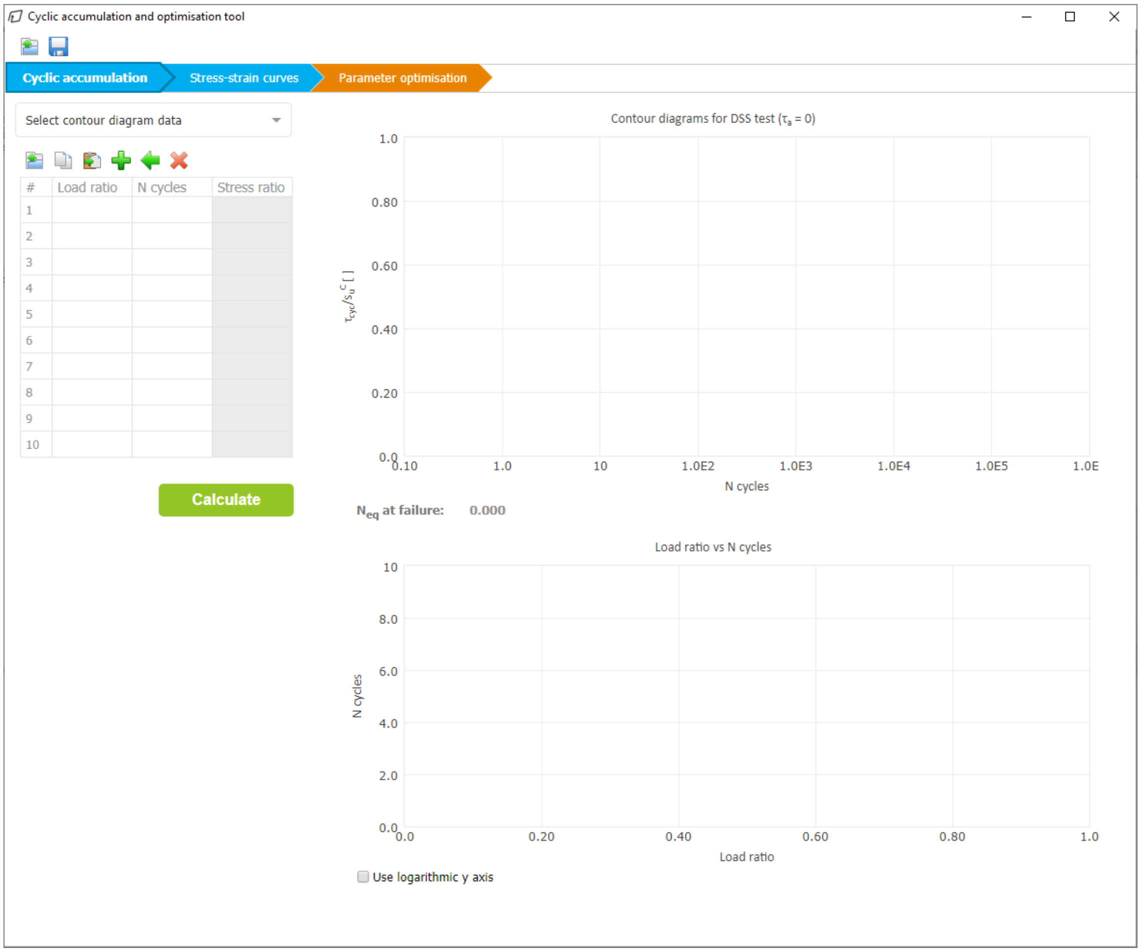



*Figure 137: Cyclic accumulation tool window*

The three steps of the cyclic accumulation and optimisation procedure are represented by the three modes (**Cyclic accumulation**, **Stress-strain curves** and **Parameter optimisation**) in the window.

## Cyclic accumulation

The purpose of this step is to determine the equivalent number of undrained cycles of the peak load, $N_{eq}$, for a given soil contour diagram and load distribution.

The following storm composition data will be used as shown in Table 22 (on page 156):

**Table 22: Composition of cyclic vertical load for a 6-hour design storm**

| # | Load ratio | N cycles |
|---|---|---|
| 1 | 0.02 | 2371 |

| # | Load ratio | N cycles |
|---|---|---|
| 2 | 0.11 | 2877 |
| 3 | 0.26 | 1079 |
| 4 | 0.40 | 163 |
| 5 | 0.51 | 64 |
| 6 | 0.62 | 25 |
| 7 | 0.75 | 10 |
| 8 | 0.89 | 3 |
| 9 | 1.0 | 1 |

1. Select an appropriate contour diagram from **Select contour diagram data** in the **Cyclic accumulation** tab. In this case, select **Drammen clay, OCR = 4**.

   **Note:** For more information about contour diagrams, see Andersen (2015) [3] and Reference Manual , Cyclic accumulation and optimisation tool.

2. The load ratios and number of cycles from the storm composition can be entered in the empty table. The storm composition is given in Table 22 (on page 156) (Jostad, Torgersrud, Engin & Hofstede, 2015) [4] as the cyclic vertical load normalized with respect to the maximum cyclic vertical load (Load ratio) and the number of cycles (N cycles). It is here assumed that the cyclic shear stress history in the soil is proportional to the maximum cyclic vertical load of the footing. The table should be entered such that the smallest load ratio is at the top and the highest load ratio is at the bottom.

   **Note:** The design storm is a load history that is transformed into parcels of constant cyclic load. Each parcel corresponds to a number of cycles at a constant amplitude determined from the time record of the load component. See Reference Manual, Cyclic accumulation and optimisation tool, for more information.

   When you've entered the load parcels in the table, the **Load ratio vs N cycles** graph will show a graphic representation of the data. For the data given here and the logarithmic scale turned on, the resulting graph is shown in Figure 138 (on page 158).

[3] Andersen, K.H. (2015). Cyclic soil parameters for offshore foundation design, volume The 3rd ISSMGE McClelland Lecture of Frontiers in Offshore Geotechnics III. Meyer (Ed). Taylor & Francis Group, London, ISFOG 2015. ISBN 978-1-138-02848-7.

[4] Jostad, H.P., Torgersrud, Ø., Engin, H.K., Hofstede, H. (2015). A fe procedure for calculation of fixity of jack-up foundations with skirts using cyclic strain contour diagrams. City University London, UK.

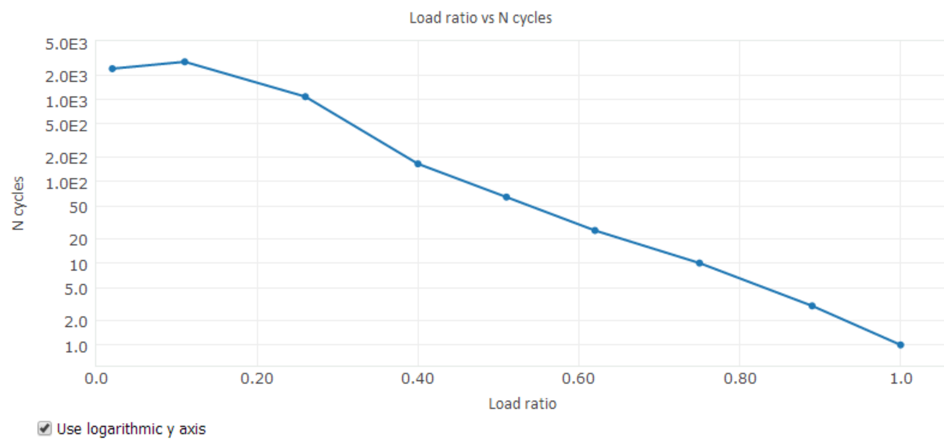



*Figure 138: Load ratio vs N cycles graph (logarithmic scale)*

**3.** Click **Calculate** to calculate the equivalent number of cycles $N_{eq}$.
The selected contour diagram is plotted together with the shear stress history for a scaling factor where the soil fails (here defined at 15% shear strain) at the last cycle (Figure 139 (on page 158)) and the loci of end-points of the stress history for different scaling factors. The calculated equivalent number of cycles corresponds to the value on the x-axis at the last point of the locus of end-points and is equal to 6.001.

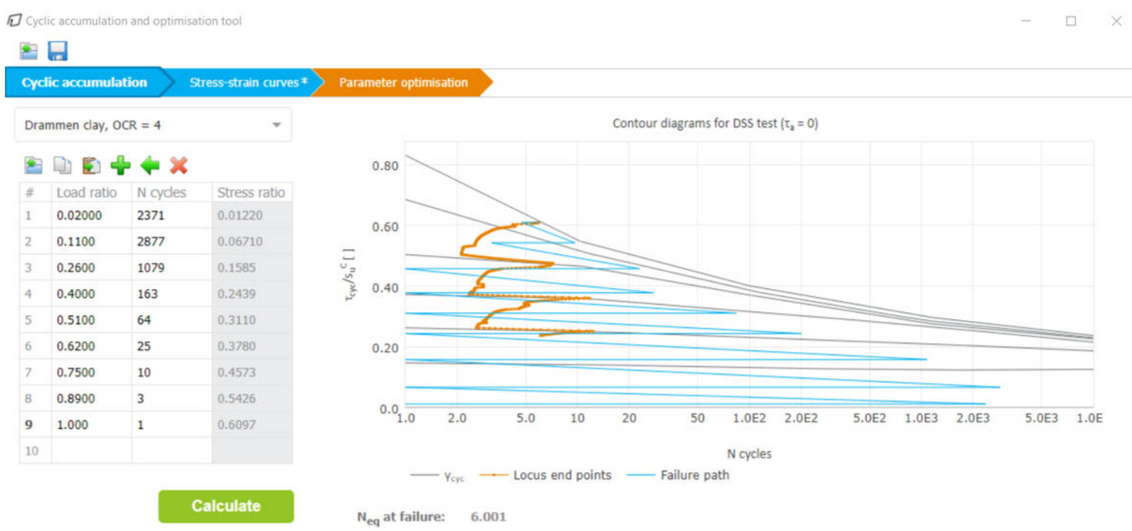



*Figure 139: Cyclic accumulation in PLAXIS 2D*

## Stress-strain curves

The purpose of this tab is to obtain non-linear stress-strain curves for a given (calculated) $N_{eq}$ and given cyclic over average shear stress ratio (here taken equal to the ratio between cyclic and average vertical load during the storm).

**1.** Go to the **Stress-strain curves** tab.

2. For the $N_{eq}$ determination, keep the default option **From cyclic accumulation**. The calculated equivalent number of cycles is adopted from the previous tab.
3. Keep the **Soil behaviour** as **Anisotropic**, and the **Scaling factor**, **DSS** and **Scaling factor**, **TX** as 1.

**Note:**

- Cyclic strength can be scaled based on available soil specific cyclic strength.
- If the plasticity index and/or water content of the soil is different from Drammen clay, the cyclic strength can be scaled by applying a scaling factor different from 1 (see Andersen, 2015 [5] for details).

4. Set the cyclic to average shear stress ratio for DSS, triaxial compression and triaxial extension, describing the inclination of the stress path, to appropriate values. In this example, the following input values are selected to obtain strain compatibility at failure, i.e. the same cyclic and average shear strain for the different stress paths at failure.
   a. cyclic to average ratio for DSS $(\Delta\tau_{cyc}/\Delta\tau_a)^{DSS}$ = 1.1,
   b. triaxial compression $(\Delta\tau_{cyc}/\Delta\tau_a)^{TXC}$ = 1.3 and
   c. extension $(\Delta\tau_{cyc}/\Delta\tau_a)^{TXE}$ = -6.3
5. Select the load type as, **Total load** for this first material.
   DSS and triaxial contour diagrams are plotted together with stress paths described by the cyclic to average ratios (Figure 140 (on page 160)). Notice that the shear stresses are normalised with respect to the static undrained shear strength in compression. The extracted stress-strain curves are plotted below the contour diagrams.
6. Click **Calculate** to produce the corresponding normalised stress-strain curves below the contour diagrams. See Figure 140 (on page 160) for the outcome.

[5] Andersen, K.H. (2015). Cyclic soil parameters for offshore foundation design, volume The 3rd ISSMGE McClelland Lecture of Frontiers in Offshore GeotechnicsIII. Meyer (Ed). Taylor & Francis Group, London, ISFOG 2015. ISBN978-1-138-02848-7.


*Figure 140: Stress-strain curves for total load*

## Parameter optimisation

The purpose of the optimisation is to obtain a set of parameters for the UDCAM-S model.

The parameter ranges and the results you will see after the optimisation are shown in Table 23 (on page 160):

**Table 23: Parameter ranges and results after optimization**

| Parameter | Name | Min value | Max value | Optimised value | Unit |
|---|---|---|---|---|---|
| Ratio of the initial shear modulus to the degraded shear strength at failure in triaxial compression | $G_{ur} / \tau^C$ | 400.0 | 480.0 | 420.4 | - |
| Shear strain at failure in triaxial compression | $\gamma_f^C$ | 6.0 | 8.0 | 6.431 | % |

| Parameter | Name | Min value | Max value | Optimised value | Unit |
|---|---|---|---|---|---|
| Shear strain at failure in triaxial extension | $\gamma_f^E$ | 5.0 | 8.0 | 7.873 | % |
| Shear strain at failure in direct simple shear | $\gamma_f^{DSS}$ | 8.0 | 12.0 | 11.97 | % |
| Ratio of the cyclic compression shear strength over the undrained static compression shear strength | $\tau^C / S_u{}^C$ | 1.14 | 1.16 | 1.152 | - |
| Ratio of the cyclic DSS shear strength over the undrained static compression shear strength | $\tau^{DSS} / S_u{}^C$ | 0.89 | 0.91 | 0.9051 | - |
| Ratio of the cyclic extension shear strength over the undrained static compression shear strength | $\tau^E / S_u{}^C$ | 0.62 | 0.64 | 0.6208 | - |
| Reference degraded shear strength at failure in the triaxial compression test | $\tau^C{}_{ref}$ | - | - | 149.7 | - |
| Reference depth | $y_{ref}$ | - | - | 0.000 | m |
| Increase of degraded shear strength at failure in the triaxial compression test with depth | $\tau^C{}_{inc}$ | - | - | 0.000 | $kN/m^2/m$ |
| Ratio of the degraded shear strength at failure in the triaxial extension test to the degraded shear strength in the triaxial compression test | $\tau^E / \tau^C$ | - | - | 0.5389 | - |
| Initial mobilisation of the shear strength with respect to the degraded TXC shear strength | $\tau^0 / \tau^C$ | - | - | 2.332E-3 | - |
| Ratio of the degraded shear strength at failure in the direct simple shear test to the degraded shear strength in the triaxial compression test | $\tau^{DSS} / \tau^C$ | - | - | 0.7858 | - |

Use the following steps to calculate the optimised values.

1. Click the **Parameter optimisation** tab.
2. Enter the parameters of the clay in the **Static properties.** Set $s_u{}^C{}_{ref}$ to 130.0 and $K_0$ determination to **Manual** and set $K_0$ to 1.0.
3. Propose minimum and maximum values for the parameters listed in Table 23 (on page 160).

> **Note:**
>
> In the optimisation, set minimum and maximum values of $\tau^C$ / $S_u{}^C$, $\tau^{DSS}$ / $S_u{}^C$, and $\tau^E$ / $S_u{}^C$ close to the results from the strain interpolation if one wants to keep these values.
>
> Calculate $G_{max}$ / $\tau^C$ by dividing $G_{max}$ from soil properties with results for ($\tau^C$ / $S_u{}^C$) · $S_u{}^C$.
>
> Set the minimum and maximum values close to this value.

4. Click **Calculate** to obtain optimised parameters (Figure 141 (on page 162) and column Optimised value of Table 23 (on page 160)).

   After a few seconds, the optimal values are shown in the corresponding column in the Parameter ranges table. Based on these values, the optimised parameters are calculated and listed in the right-hand side of the table as shown in Figure 141 (on page 162)

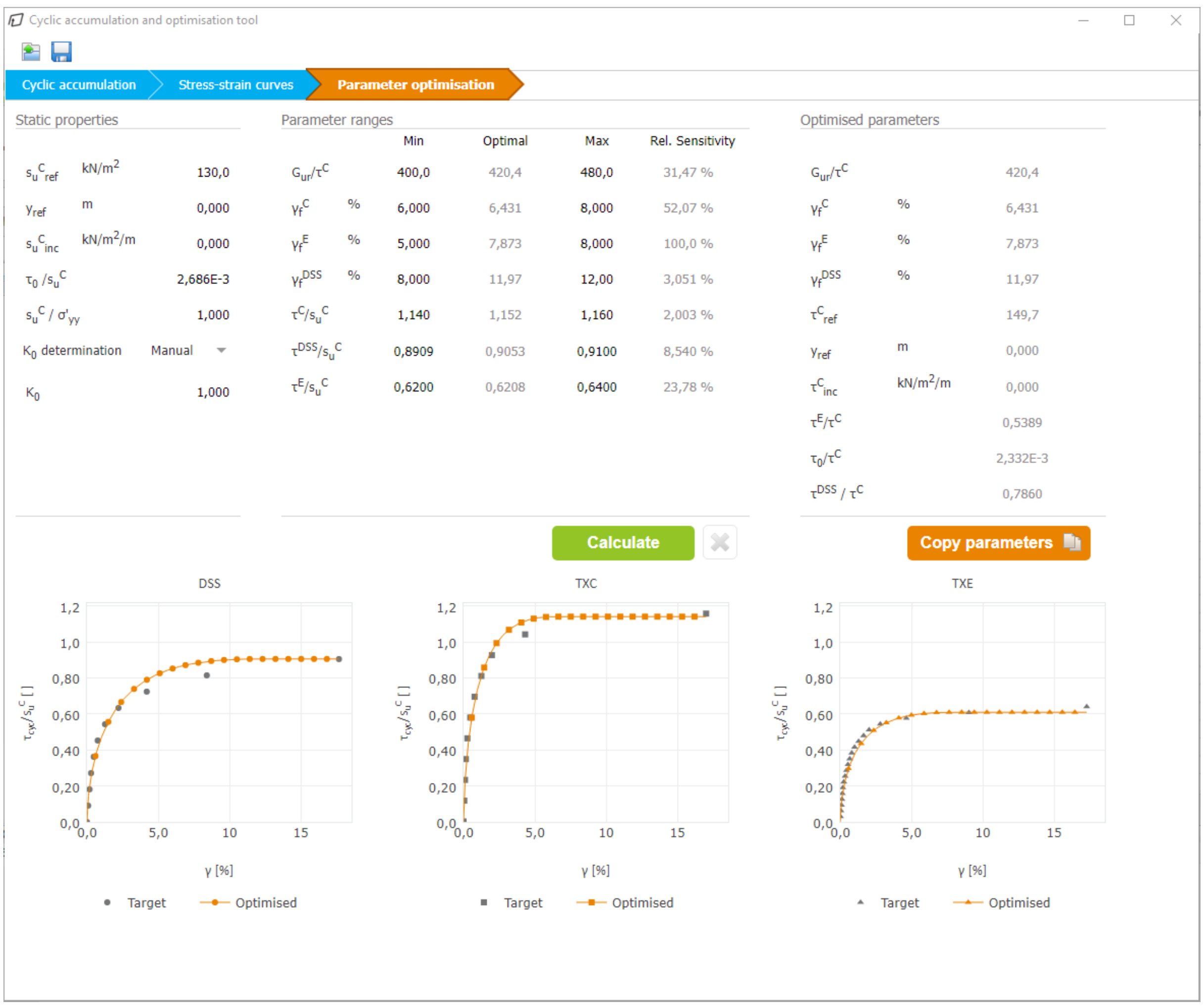



*Figure 141: Optimised parameters for total load*

The resulting stress-strain curves from test simulations with the UDCAM-S model using the optimised parameters are shown together with the target points from the contour diagrams.

5. When the calculation has finished, save the application state of the Cyclic accumulation and optimisation tool. The saved data will be used when creating another material. To save the application state, press the **Save** button at the top of the window. Save the state under the file name `optimised_total.json`.
6. Copy the optimised material parameters: Press the **Copy parameters** button and go back to the **Soil-UDCAM-S** window describing the material.
7. Click the **Paste material** button.
   The values in the **Mechanical** tab are replaced with the new values as shown in Figure 142 (on page 163) .

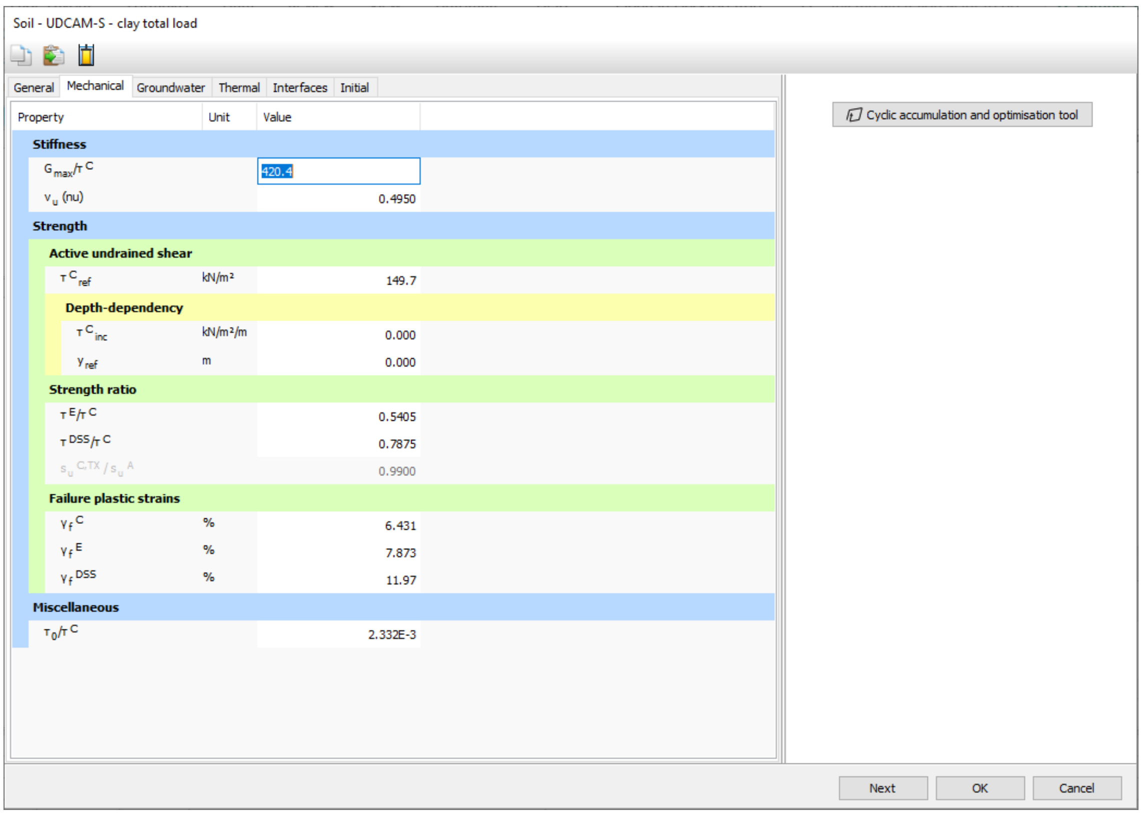



*Figure 142: Copy parameters into Clay total material*

8. Go to the **Initial** tab and set $K_0$ to 1 by setting $K_0$ determination to Manual, check $K_{0,x} = K_{0,z}$ (default) and set $K_{0,x}$ to 1.
9. Click **OK** to close the created material.
10. Assign the **Clay - total load** material set to the soil layer in the borehole.

### 10.5.2 Material: Clay - cyclic load

Create a material for the second clay. Some information from the *Clay - total load* material will be reused. The optimisation of the parameters has to be recalculated though, based on other conditions.

The parameter ranges and the results you will see after the optimisation are shown in Table 24 (on page 164):

**Table 24: Parameter ranges and results after optimisation**

| Parameter | Name | Min value | Max value | Optimised value | Unit |
|---|---|---|---|---|---|
| Ratio of the initial shear modulus to the degraded shear strength at failure in triaxial compression | $G_{max} / \tau^C$ | 700.0 | 800.0 | 703.2 | - |
| Shear strain at failure in triaxial compression | $\gamma_f^C$ | 1.0 | 3.0 | 2.966 | % |
| Shear strain at failure in triaxial extension | $\gamma_f^E$ | 1.0 | 3.0 | 2.699 | % |
| Shear strain at failure in direct simple shear | $\gamma_f^{DSS}$ | 1.0 | 3.0 | 2.946 | % |
| Ratio of the cyclic compression shear strength over the undrained static compression shear strength | $\tau^C / S_u{}^C$ | 0.66 | 0.67 | 0.6667 | - |
| Ratio of the cyclic DSS shear strength over the undrained static compression shear strength | $\tau^{DSS} / S_u{}^C$ | 0.47 | 0.49 | 0.4787 | - |
| Ratio of the cyclic extension shear strength over the undrained static compression shear strength | $\tau^E / S_u{}^C$ | 0.57 | 0.59 | 0.5790 | - |
| Reference degraded shear strength at failure in the triaxial compression test | $\tau^C{}_{ref}$ | - | - | 86.67 | - |
| Reference depth | $y_{ref}$ | - | - | 0.000 | m |
| Increase of degraded shear strength at failure in the triaxial compression test with depth | $\tau^C{}_{inc}$ | - | - | 0.000 | $kN/m^2/m$ |
| Ratio of the degraded shear strength at failure in the triaxial extension test to the degraded shear strength in the triaxial compression test | $\tau^E / \tau^C$ | - | - | 0.8684 | - |

| Parameter | Name | Min value | Max value | Optimised value | Unit |
|---|---|---|---|---|---|
| Initial mobilisation of the shear strength with respect to the degraded TXC shear strength | $\tau^0 / \tau^C$ | - | - | 0.000 | - |
| Ratio of the degraded shear strength at failure in the direct simple shear test to the degraded shear strength in the triaxial compression test | $\tau^{DSS} / \tau^C$ | - | - | 0.7181 | - |

Use the following steps to calculate the optimised values.

1. Copy the **Clay - total load** material.
2. Enter `Clay - cyclic load` for the identification.
3. Go to the **Mechanical** tab.

   Like for the first material, also here the parameters will be determined using the **Cyclic accumulation and optimisation tool**.
4. Click the **Cyclic accumulation and optimisation tool** button on the **Mechanical** tab to open the tool.
5. Click the **Open file** button  and choose the application state `optimised_total.json` that was saved after optimisation of the first material.
   All tabs will be filled with data.
6. Leave the **Cyclic accumulation** tab as it is.
7. Go to the **Stress-strain curves** tab, set load type to **Cyclic load**.
8. Press **Calculate** and let the calculation finish.
   The stress-strain curves are shown in Figure 143 (on page 166):

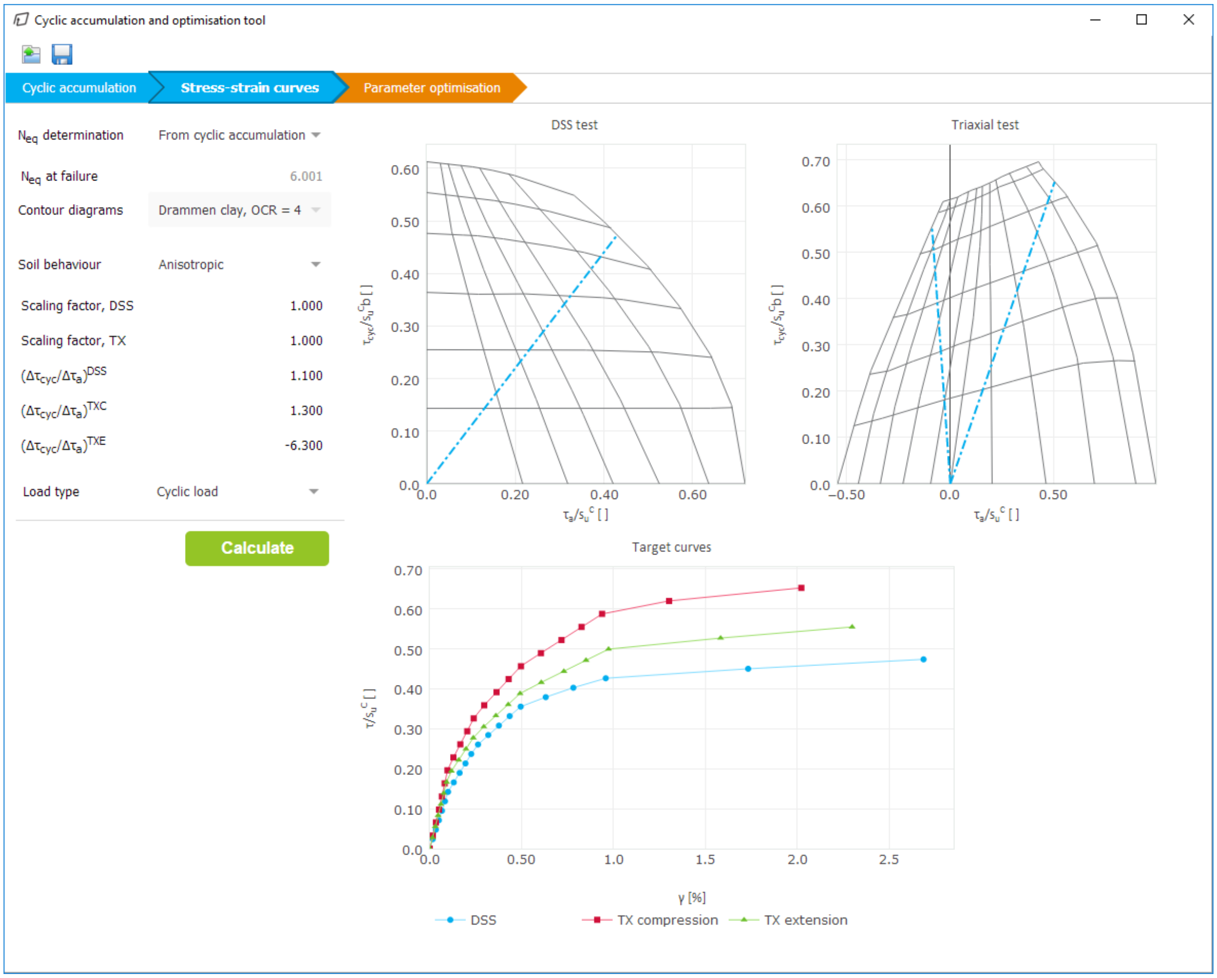



*Figure 143: Stress-strain curves for cyclic load*

9. Go to the **Parameter optimisation** tab. Accept the notification about resetting the optimisation tab to get updated values.
10. Make sure that $s_u{}^C{}_{ref}$ is set to 130.0 and set $K_0$ determination to Automatic.
11. Modify the minimum and maximum values for the **Parameter ranges**, see Table 24 (on page 164) for values.
12. Click **Calculate** to get the optimised parameters.
    The optimised parameters are shown in the Figure 144 (on page 167) and are also listed in the column 'Optimised value' in Table 24 (on page 164).

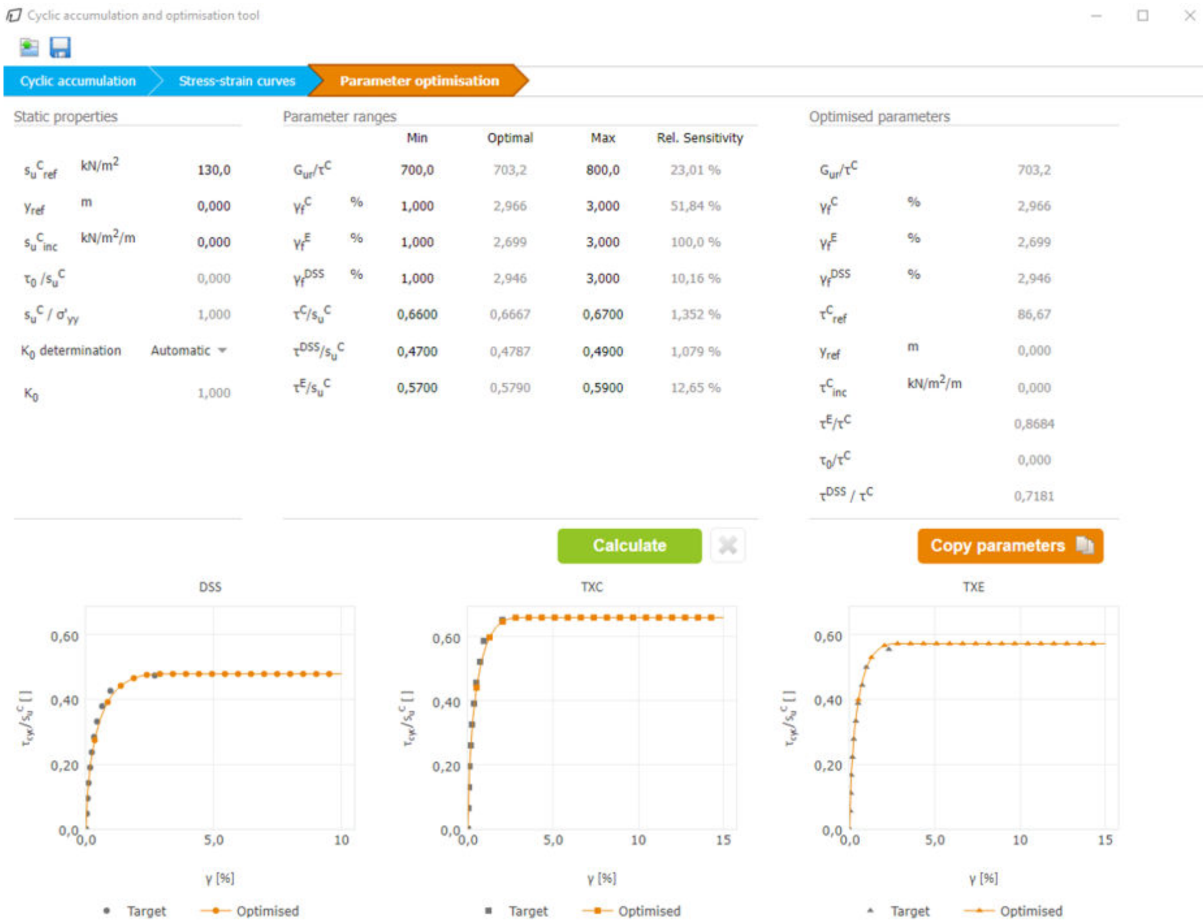



*Figure 144: Optimised parameters for cyclic load*

13. Save the application state under the file name `optimised_cyclic.json`.
14. Copy the optimised material parameters: Click the **Copy parameters** button and go back to the **Soil-UDCAM-S** window.
15. Click the **Paste material** button .
    The values in the **Mechanical** tab are replaced with the new values.
16. Click **OK** to close the created material.

## 10.5.3 Material: Concrete

Create a new material for the concrete foundation.

1. Choose **Soil and interfaces** as the **Set** type and click the **New** button.
2. Enter `Concrete footing` for the **Identification** and select **Linear elastic** as the **Soil model**.

3. Set the **Drainage type** to **Non-porous**.
4. Enter the properties of the material:
   a. a unit weight of 24 kN/m$^3$,
   b. Young's modulus of 30x10$^6$ kN/m$^2$ and
   c. a Poisson's ratio of 0.1.
5. Click **OK** to close the created material.
6. Click **OK** to close the **Material sets** window.

## 10.6 Define the structural elements

The concrete foundation and interfaces have to be defined.

### 10.6.1 Define the concrete foundation

1. Click the **Structures** tab to proceed with the input of structural elements in the Structures model.
2. Select the **Create soil polygon** feature in the side toolbar and click on (0.0, 0.0), (11.0, 0.0), (11.0, -1.0) and (0.0, -1.0).

**Note:** Do not yet assign the **Concrete footing** material to the polygon.

### 10.6.2 Define the interfaces

Create an interface to model the interaction of the foundation and the surrounding soil. Extend the interface half a meter into the soil. Make sure the interface is at the outer side of the footing (inside the soil). The interface is created in two parts.

1. Click **Create interface** to create the upper part from (11.0, -1.0) to (11.0, 0.0), Figure 146 (on page 169).
2. Click **Create interface** to create the lower part (between foundation and soil) from (11.0, -1.5) to (11.0, -1.0), Figure 146 (on page 169).
3. The upper part interface (between the foundation and the soil) is modeled with a reduced strength of 30%.
   a. Make a copy of the **Clay - total load** material and name it `Clay - total load - interface .`
   b. Reduce the interface strength by setting $R_{inter}$ to 0.3 as shown in Figure 145 (on page 168) and

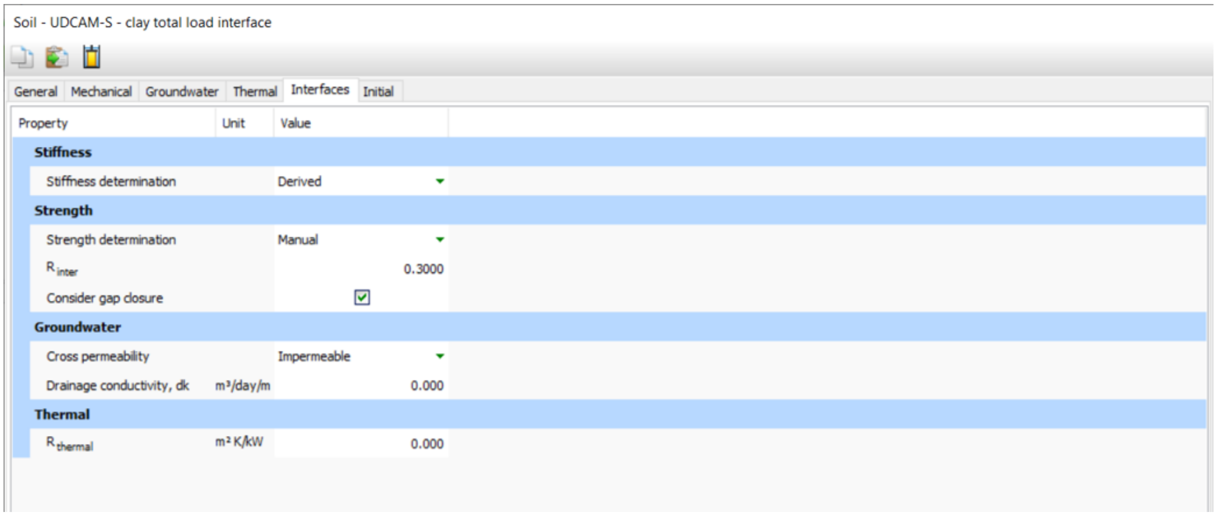



*Figure 145: Clay-Total load interface*

   **c.** Assign this to the upper part of the interface.

**4.** For Phase 3 (Calculate vertical cyclic stiffness), another material with reduced strength is needed.

   **a.** Make a copy of the **Clay - cyclic load material** and name it `Clay - cyclic load - interface`.

   **b.** Reduce the interface strength by setting $R_{inter}$ to 0.3.

   **c.** Do not assign this yet. It will be assigned when defining Phase 3.

**5.** For the interface material extended into the soil, full soil strength is applied ($R_{inter}$ = 1.0), as implicitly defined in the original clay material **Clay - total load.** Keep the default setting **Material mode**: From adjacent soil. The geometry of the model is shown in Figure 146 (on page 169):

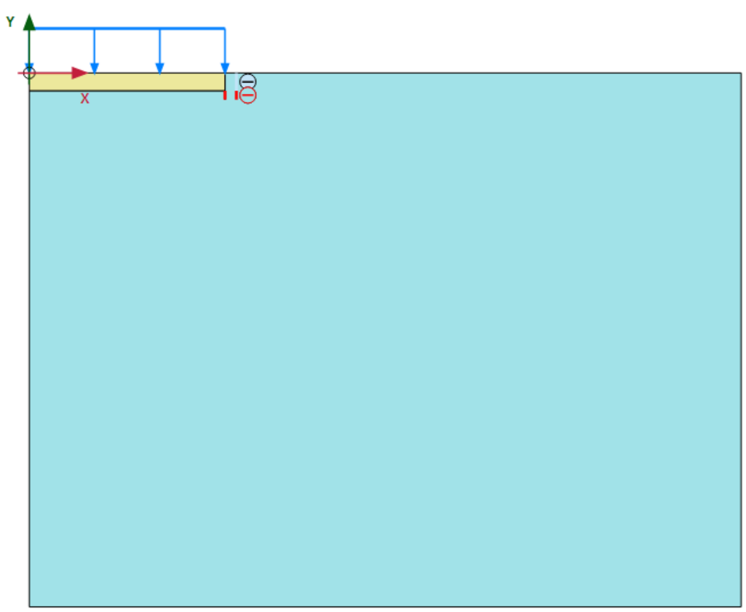



*Figure 146: Geometry of the model*

### 10.6.3 Define a vertical load

In order to calculate the cyclic vertical capacity and stiffness, a vertical load is applied at the top of the foundation.

1. Define a distributed load by selecting **Create line load** and click (0.0, 0.0) and (11.0, 0.0).
2. In the **Selection explorer** set the value of $q_{y,start,ref}$ to -1000 kN/m/m.

## 10.7 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 147 (on page 170).

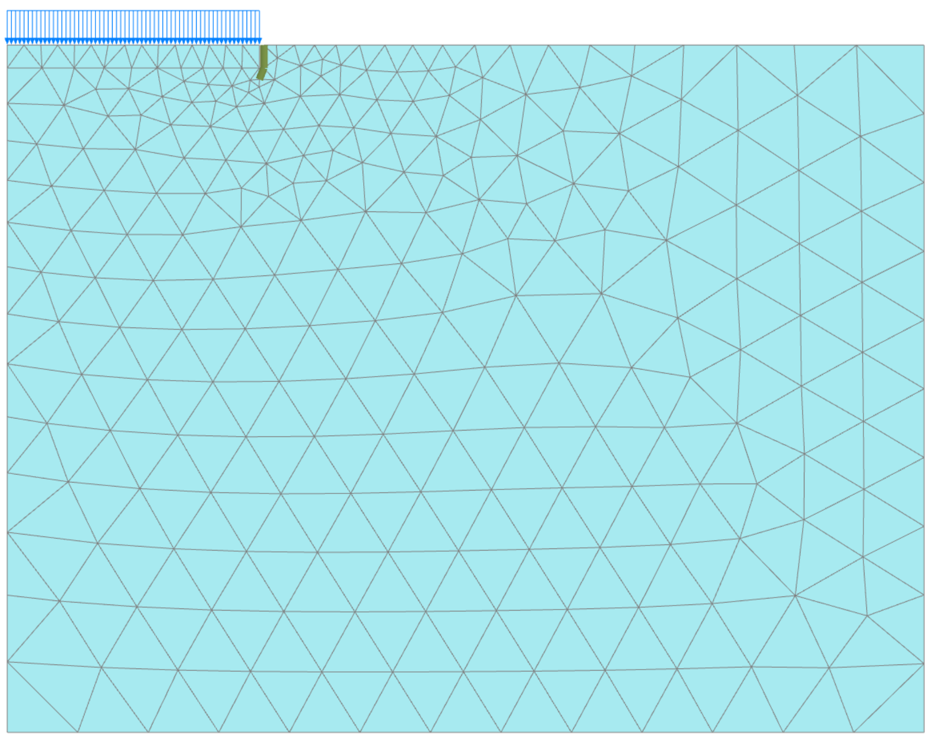


*Figure 147: The generated mesh*

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

# 10.8 Define and perform the calculation

The calculation consists of the following phases:

- In the Initial phase, the initial stress conditions are generated by the **K0 procedure**, using the default values.
- In Phase 1 the footing is activated by assigning the **Concrete** material to the corresponding polygon. The interfaces are also activated.
- In Phase 2 the total cyclic vertical bearing capacity and stiffness are calculated.
- In Phase 3 the cyclic vertical bearing capacity and stiffness are computed.

## 10.8.1 Initial phase

1. Proceed to **Staged construction mode** .
2. In the **Phases explorer** double-click the initial phase.
3. Make sure that **Calculation type** is set to K0 procedure.
4. Click OK to close the **Phases** window.

## 10.8.2 Phase 1: Footing and interface activation

1. Click the **Add phase** button to create a new phase.
2. Phase 1 starts from the Initial phase.
3. Activate the footing by assigning the **Concrete footing** material to the corresponding polygon.
4. Activate the interfaces as well.

## 10.8.3 Phase 2: Cyclic Vertical Bearing capacity and stiffness

In Phase 2 the total cyclic vertical bearing capacity and stiffness are calculated. The vertical bearing capacity is obtained by increasing the vertical load (stress) until failure. The stiffness is calculated as the force divided by the displacement.

1. Click the **Add phase** button to create a new phase.
2. Phase 2 starts from Phase 1.
3. In the **Phases** window go to the **Deformation control parameters** subtree and select the **Reset displacements to zero** option and **Reset small strain**.
4. In the **Phases** window go to the **Numerical control parameters** and in **Max number of steps stored** set **500** steps.

5. Activate the line load.

### 10.8.4 Phase 3: Calculate vertical cyclic stiffness

In Phase 3, which also starts from Phase 1, the vertical cyclic stiffness is calculated by activating the **Clay - cyclic load material.** The vertical bearing capacity is obtained by increasing the vertical load (stress) until failure.

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window set the **Start from phase** to Phase 1.
3. Go to the **Deformation control parameters** subtree and select the **Reset displacements to zero** option and **Reset small strain**.
4. In the **Phases** window goto the **Numerical control parameters** and in **Max number of steps stored** set **500** steps and close the **Phases** window.
5. Replace the soil material with the **Clay - cyclic load**.
6. Assign the material **Clay - cyclic load - interface** material to the upper part of the interface. The material mode of the lower part of the interface remains **From adjacent soil**.
7. Make sure the line is activated.

The calculation definition is now complete.

### 10.8.5 Execute the calculation

Before starting the calculation it is recommended to select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams.

To do this, follow these steps:

1. Click the **Select points for curves** button in the side toolbar.
   The connectivity plot is displayed in the Output program and the **Select points** window is activated.
2. Select a *pre-calc node* on the footing (0.0, 0.0). Close the **Select points** window.
3. Click on the **Update** tab to close the Output program and go back to the Input program.
4. Click the **Calculate** button to calculate the project.
5. Once the calculation is finished, go to the Output program and click on the **Select points for curves** button and chose a *post-calc node* on the footing (0.0, 0.0).

## 10.9 Results

## 10.9.1 Total load cyclic vertical bearing capacity

Applied vertical stress (load): $q_y$ = -1000kN/m$^2$

Failure at: $q_y$ = 720kN/m$^2$(Figure 148 (on page 173))

Total vertical bearing capacity: $V_{cap} = q_y \cdot \text{Area} = 720\ \text{kN/m}^2 \cdot \pi \cdot (11\text{m})^2 = 273.7\text{MN}$

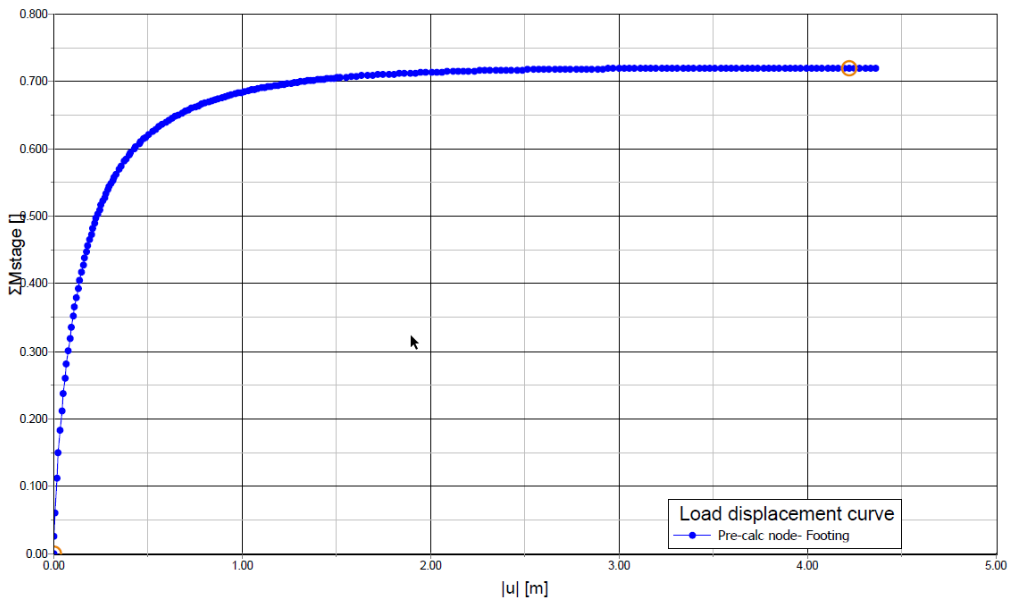



*Figure 148: Total displacement vs ΣMstage*

**Note: Generating a load displacement curve using pre-calc points**

The load-displacement curve as function of $q_y$ can be generated on a spreadsheet using the calculation results on selected *precalculation nodes* of ΣMstage versus their corresponding displacement (also called load-displacement curve).

Take into account that for PLAXIS a distributed load $q_y$ in any calculation step can be defined as:

$q_{active}=q_{phase.start}+\Sigma Mstage(q_{phase.end}-q_{phase.start})$

where:

- $q_{phase.start}$ is the load value at the start of the phase (or actually the load value at the end of the previous phase).
- $q_{phase.end}$ is the desired load value at the end of the current phase, i.e. the defined value in the Staged Construction settings.

Since for this tutorial $q_{phase.start}$ =0 then $q_{active}=\Sigma Mstage \times q_{phase.end}$

**Procedure:**

1. From the Output program go to **Curves manager** and obtain the ΣMstage-displacement curve.

2. Click on the icon. Copy the |u| vs ΣMstage values for all steps and paste them on the spreadsheet. Ensure that the values and their units pasted are consistent with PLAXIS output.
3. Multiply each value of the ΣMstage column with the value of $q_{phase.end}$. For this example, $q_{phase.end}$ is equal to the defined vertical load of 1000KN.
4. Graph |u| vs $q_y$.

For comparison, the static vertical bearing capacity (using the static undrained shear strength) is found to be 228.1MN. The reason for the larger vertical bearing capacity is that the shear strengths increase due to the higher strain rate during wave loading, compared to the value obtained from standard monotonic laboratory tests, and this effect is larger than the cyclic degradation during the storm.

## 10.9.2 Cyclic load cyclic vertical bearing capacity

Applied vertical stress (load): $q_y$ = -1000kN/m$^2$

Failure at: $q_y$ = 458.1kN/m$^2$(Figure 149 (on page 174))

Total vertical bearing capacity:

$V_{cap} = q_y \cdot \text{Area} = 458.1\text{kN/m}^2 \cdot \pi \cdot (11\text{ m})^2 = 174.14\text{MN}$

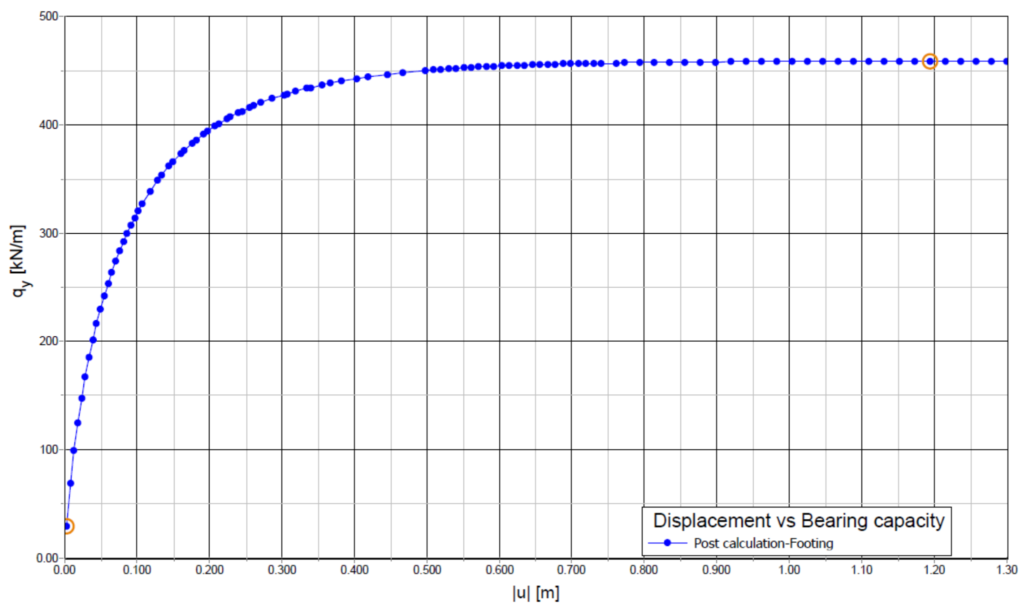



*Figure 149: Total displacement vs vertical bearing capacity curve for cyclic load*

**Note: Generating a load displacement curve using post-calc points**

Load displacement curves as a function of $q_y$ vs |u| (see Figure 149 (on page 174)) can be generated directly in the **Curves manager** using the data on post-cal points. For this it is necessary to ensure that before running the calculation a **Max number of steps stored** (inside the **Numerical control parameters**) is defined for each specific phase (for this example 500 steps were utilized). Be aware that storing several calculation steps to obtain results on post-cal nodes might produce heavier files with can be unsuitable depending on the project. If this is case it is advised to obtain load displacement curves with pre-calc nodes as indicated in previous sections.

In the **Curves manager** window, select the Invert sign option to obtain the positive values of $q_y$ before generating the chart.

## 10.9.3 Vertical stiffness

The vertical stiffness (accounting for cyclic loading) is calculated as $k_y = F_y / u_y$ for both the total and the cyclic component. The total vertical displacement includes accumulated vertical displacements during the storm. Load versus stiffness is shown in the following Figure 150 (on page 175):

**Note:** To construct the vertical load vs vertical stiffness graph, use the values of the load displacement curve (|u| vs $q_y$) and operate $F_y$ for each step, take into account:

- $F_y = q_y$ x $Area_{footing}$.
- For this example, $u_y$ is equal to |u|.

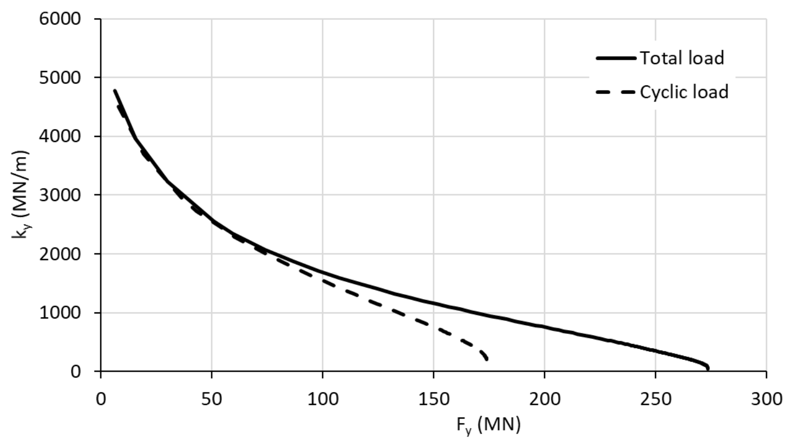



*Figure 150: Vertical load versus stiffness for total and cyclic load components*

# 11 Flow through an embankment [ULT]

In this chapter the flow through an embankment will be considered. Flow takes place from the left side (river) to the right side (polder). As a result seepage will take place at the right side of the embankment. The position of the phreatic level depends on the river water level, which varies in time.

**Objectives**

- Performing **Flow only** analysis
- Using cross section curves

**Geometry**

Figure 151 (on page 176) shows the layout of the embankment problem where free surface groundwater flow occurs. The crest of the embankment has a width of 2.0 m. Initially the water in the river is 1.5 m deep. The difference in water level between the river and the polder is 3.5 m.

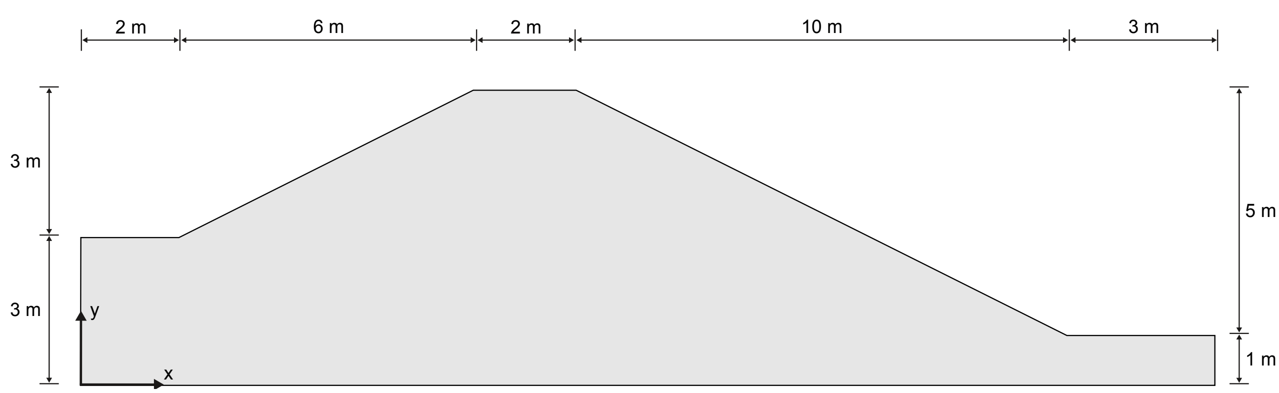



*Figure 151: Geometry of the project*

## 11.1 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, $x_{max}$ = 23 m, $y_{min}$ = 0 m and $y_{max}$ = 6 m
5. Keep the default values for units, constants and the general parameters and click **OK**.

The **Project properties** window closes.

## 11.2 Define the soil stratigraphy

A number of boreholes has to be defined according to the information in Table 25 (on page 177).

**Table 25: Information on the boreholes in the model**

| Borehole number | Location (x) | Head | Top | Bottom |
|---|---|---|---|---|
| 1 | 2 | 4.5 | 3 | 0 |
| 2 | 8 | 4.5 | 6 | 0 |
| 3 | 10 | 4.0 | 6 | 0 |
| 4 | 20 | 1.0 | 1 | 0 |

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 2.
   The **Modify soil layers** window pops up.
2. Specify the head value as 4.5.
3. Add a soil layer in the borehole. Set the top level to 3. No change is required for the bottom boundary of the layer.
4. Create the rest of the required boreholes according to the information given in Table 25 (on page 177).

## 11.3 Create and assign material data set

A material data set needs to be created for the soil layer.

The sand layer has the following properties as shown in Table 26 (on page 177) .

**Table 26: Material properties of the embankment material (sand)**

| Parameter | Name | Sand | Unit |
|---|---|---|---|
| **General** | | | |
| Material model | - | Linear elastic | - |
| Drainage type | - | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 20 | kN/m$^3$ |

| Parameter | Name | Sand | Unit |
|---|---|---|---|
| **General** | | | |
| Saturated unit weight | $\gamma_{sat}$ | 20 | $kN/m^3$ |
| **Mechanical** | | | |
| Young's modulus | $E'_{ref}$ | $10 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 0.3 | - |
| **Groundwater** | | | |
| Classification type | - | Standard | - |
| Soil class | - | Medium fine | - |
| Flow parameters - Use defaults | - | From data set | - |
| Horizontal permeability | $k_x$ | 0.02272 | m/day |
| Vertical permeability | $k_y$ | 0.02272 | m/day |

To create the material set, follow these steps:

1.  Define the soil material according to the table above and assign the material dataset to the cluster. Skip the **Interfaces** and **Initial** tabsheets as these parameters are not relevant.
2. After assigning the material to the soil cluster close the **Modify soil layers** window.

## 11.4 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Select the two lines that form the left hand side slope and river bed as shown in Figure 152 (on page 179) . In the **Selection Explorer** specify a **Coarseness factor** of 0.5.

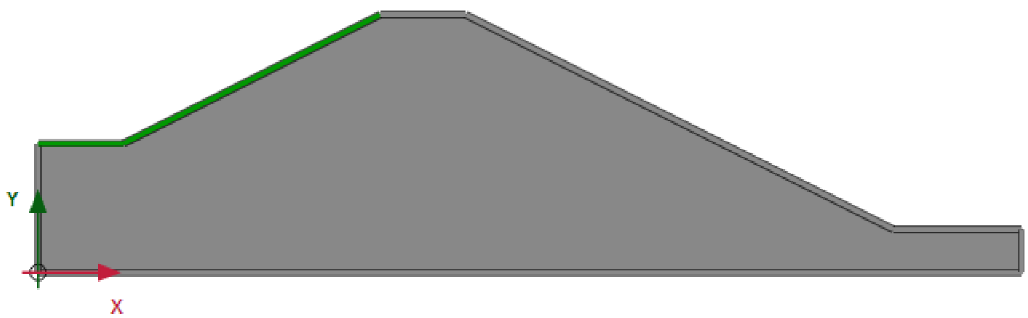



*Figure 152: Selection of left hand side of slope*

3. Click the **Generate mesh** button  to generate the mesh.
   The **Mesh options** window appears.
4. Select the **Fine** option in the **Element distribution** drop-down menu and generate the mesh.
5. Click the **View mesh** button  to view the mesh as shown in Figure 153 (on page 179).

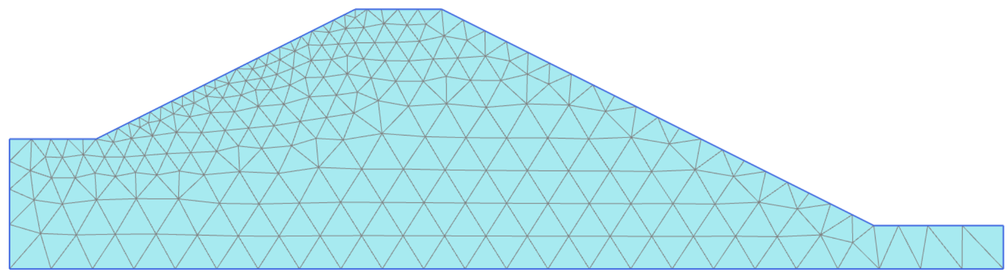


*Figure 153: The generated mesh*

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

## 11.5 Define and perform the calculation

In this project only the flow related behaviour will be analysed. The calculation process consists of three phases that will be defined in the **Staged construction** mode. In the initial phase, the groundwater flow in steady state is calculated for an average river level. In Phase 1, the transient groundwater flow is calculated for a harmonic variation of the water level. In Phase 2, the calculation is similar as in Phase 1, but the period is longer.

Click the **Staged construction** tab to proceed to the corresponding mode. A global level is automatically created according to the head values specified for each borehole (Table 25 (on page 177)). The model in the **Staged construction** mode is shown in Figure 154 (on page 180) .

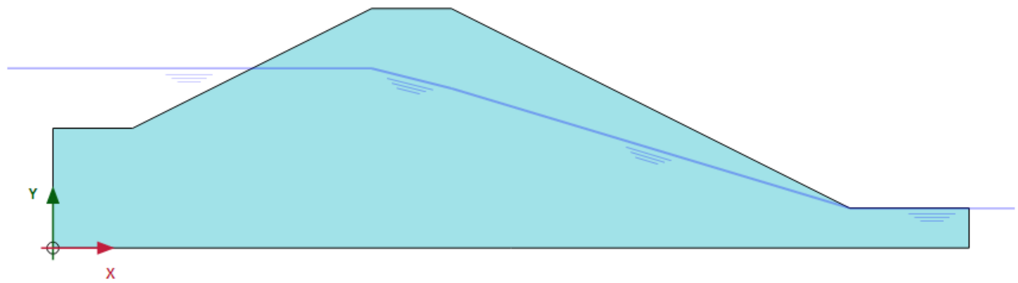



*Figure 154: The model in the **Staged construction mode***

**Note:** Note that the 'internal' part of the global water level will be replaced by the result of the groundwater flow calculation.

## 11.5.1 Initial phase

1. Double-click the initial phase in the **Phases** explorer.
2. In the **General** subtree select the **Flow only** option as the **Calculation type**.
3. The default values of the remaining parameters are valid for this phase. Click **OK** to close the **Phases** window.
4. In the **Model explorer** expand the **Model conditions** subtree.
5. In the **Model conditions** expand the **GroundwaterFlow** subtree. The default boundary conditions are relevant for the initial phase. Confirm that only the bottom boundary is closed. The expanded groundwater flow boundary conditions is shown in Figure 155 (on page 180).

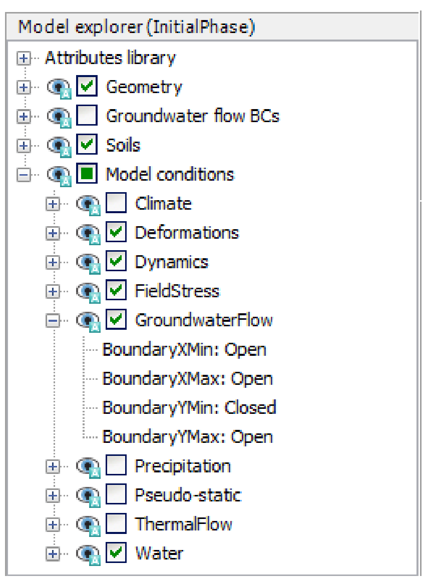



*Figure 155: The groundwater flow boundary conditions for the initial phase*

6. In the **Model explorer** expand the **Groundwater flow BCs** subtree. The boundary conditions at the extremities of the model are automatically created by the program and listed under the **GWFlowBaseBC**.

> **Note:** Note that when the boundary conditions under the **Groundwater flow BCs** subtree are active, the model conditions specified in the **GroundwaterFlow** are ignored.

## 11.5.2 Phase 1-Transient ground water flow analysis

1. Click the **Add phase** button to create a new phase.
2. In the **Phases explorer** double-click the current phase.
3. In the **General** subtree select the **Transient groundwater flow** option as pore pressure calculation type.
4. Set the **Time interval** to 1 day.
5. In the **Numerical control parameters** subtree set the **Max number of steps stored** parameter to 50. The default values of the remaining parameters will be used.
6. Click **OK** to close the **Phases** window.
7. Click the **Select multiple objects** button in the side toolbar.
8. Click **Select lines** > **Select water boundaries.**
9. Select the hydraulic boundaries as shown in Figure 156 (on page 181).

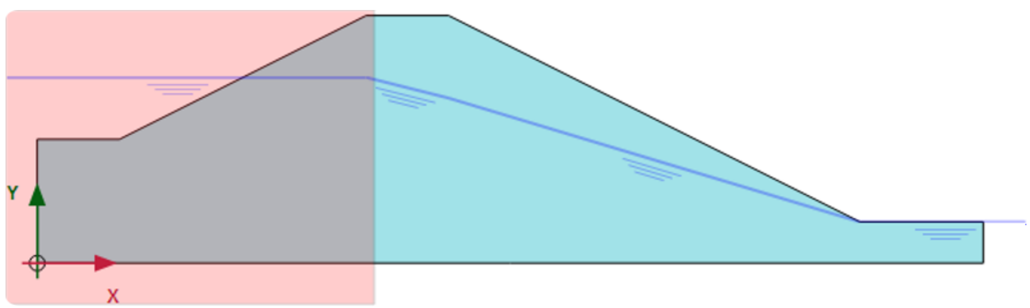



*Figure 156: Selected hydraulic boundaries*

10. Right-click and click **Activate**.
11. In the **Selection explorer** set the **Behaviour** parameter to **Head**.
12. Set $h_{ref}$ to 4.5 m.
13. Select the **Time dependent** option in the **Time dependency** drop-down menu.
14. Click on the **Head function** parameter.
15. Click the **Add** button to add a new head function.
16. In the **Flow functions** window select the **Harmonic** option in the **Signal** drop-down menu. Set the amplitude to 1 m, the phase angle to 0° and the period to 1 day.

    The flow function for the rapid phase is shown in Figure 157 (on page 182).

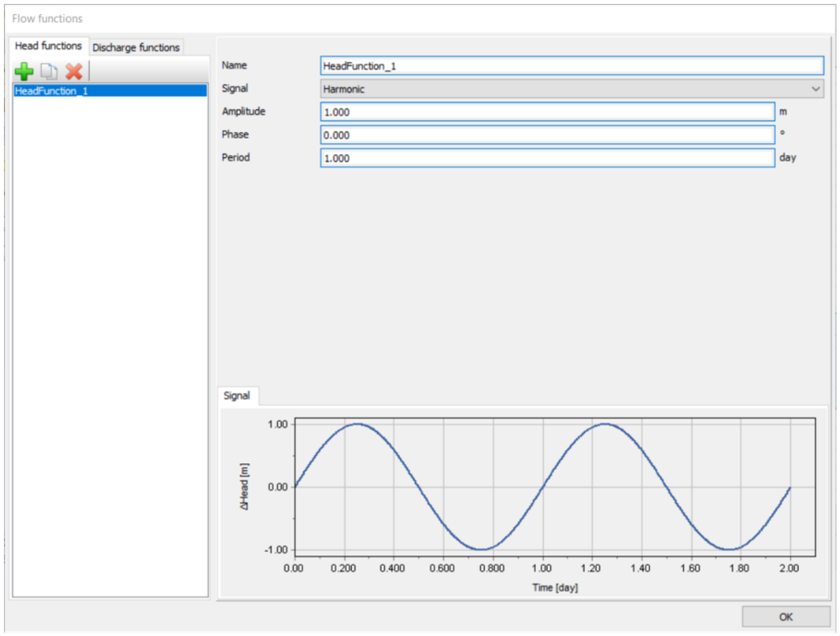



*Figure 157: The flow function for the rapid case*

17. Click **OK** to close the **Flow functions** window.

### 11.5.3 Phase 2-Long term groundwater flow analysis

1. Click the **Add phase** button to create a new phase.
2. In the **Phases explorer** double-click the current phase.
3. In the **General** subtree select the **Initial phase** in the **Start from phase** drop-down menu.
4. Select the **Transient groundwater flow** option as **Pore pressure calculation type**.
5. Set the **Time interval** to 10.0 day.
6. In the **Numerical control parameters** subtree set the **Max number of steps stored** parameter to 50. The default values of the remaining parameters will be used.
7. Click **OK** to close the **Phases** window.
8. Make sure that the same boundaries are selected as in Phase 1.
9. In the **Selection explorer** click on the **Head function** parameter.
10. Click the **Add** button to add a new head function.
11. In the **Flow functions** window select the **Harmonic** option in the **Signal** drop-down menu. Set the amplitude to 1 m, the phase angle to 0° and the period to 10 day as shown in Figure 158 (on page 183).

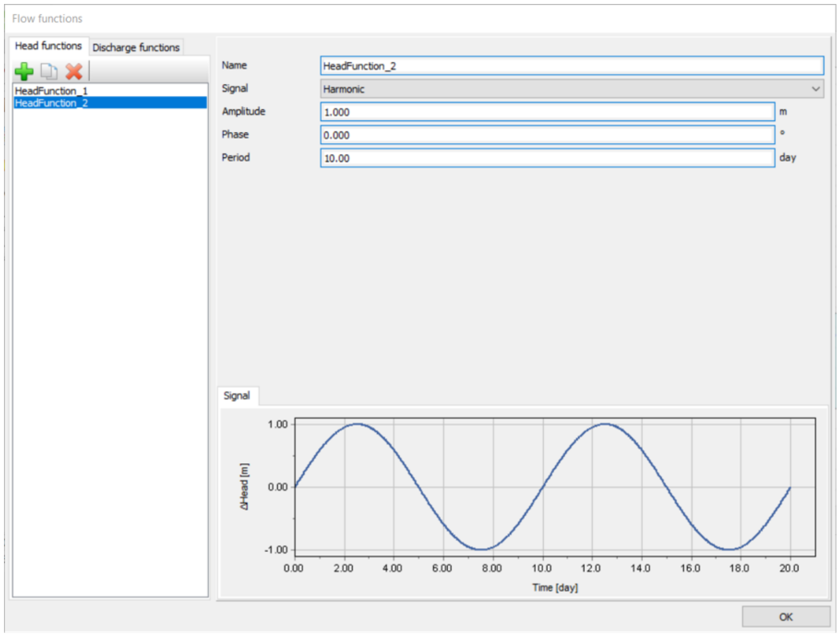



*Figure 158: The flow function for the slow case*

**12.** Click **OK** to close the **Flow functions** window.

## 11.5.4 Execute the calculation

To select points to be considered in curves:

1. In the **Staged construction** mode click the **Select point for curves** button in the side toolbar. The **Connectivity plot** is displayed in the Output program.
2. In the **Select points** window select nodes located nearest to (0 3) and (8 2.5) to be considered in curves.
3. Click **Update** to close the output program.
4. Click the **Calculate** button to calculate the project.
5. Save the project after the calculation has finished.

# 11.6 Results

In the Output program the **Create animation** tool can be used to animate the results displayed in the Output program. To create the animation follow these steps:

1. Click the menu **Stresses** > **Pore pressures** > **Groundwater head**.
2. Select the menu **File** > **Create animation**. The corresponding window pops up.

3. Define the name of the animation file and the location where it will be stored. By default the program names it according to the project and stores it in the project folder. In the same way animations can be created to compare the development of pore pressures or flow field.
4. Deselect the Initial phase and Phase 2, such that only Phase 1 is included in the animations and rename the animation accordingly. The **Create animation** window is shown in Figure 159 (on page 184):

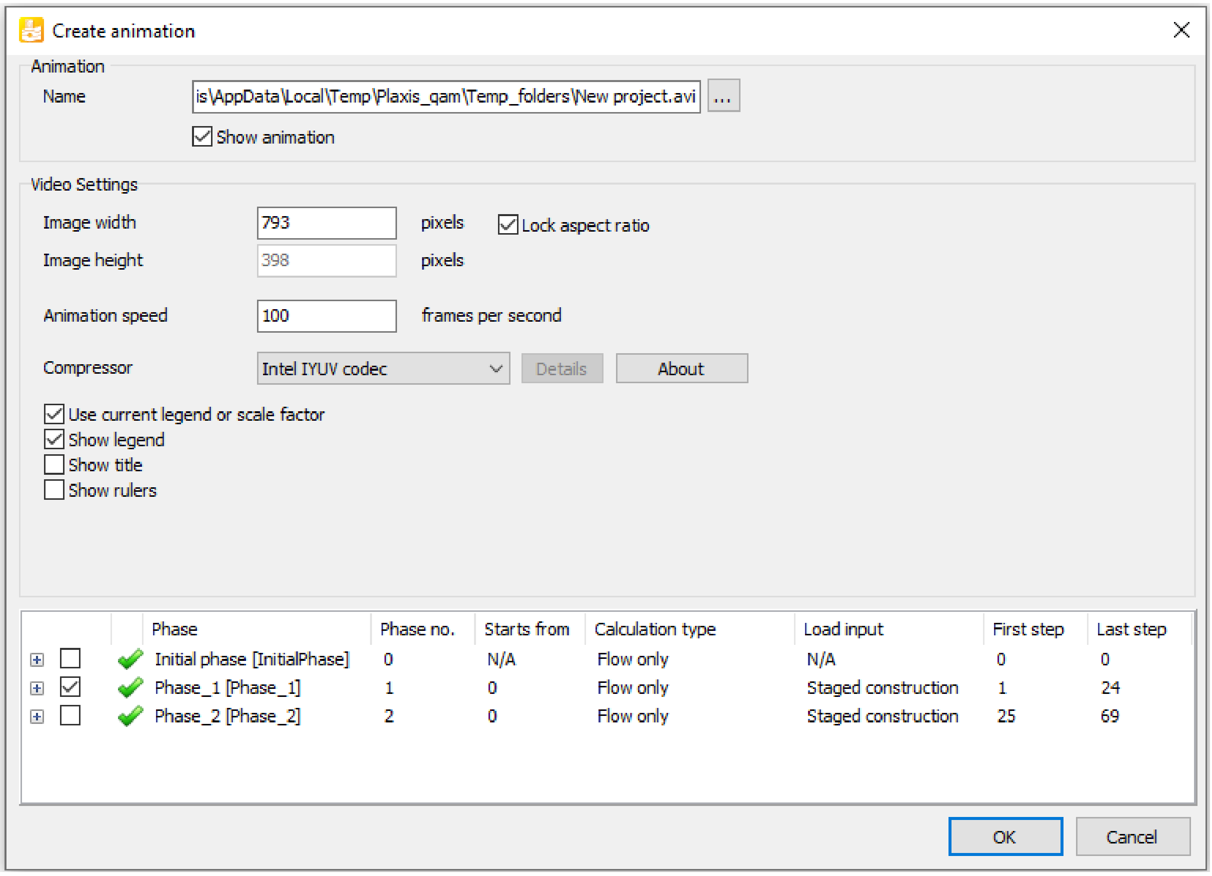



*Figure 159: **Create animation** window*

To view the results in a cross section:

1. Click the **Cross section** button in the side toolbar. The **Cross section points** window pops up and the start and the end points of the cross section can be defined. Draw a cross section through the points (2.0 3.0) and (20.0 1.0). The results in the cross section are displayed in a new window.
2. In the **Cross section** view select the menu **Stresses** > **Pore pressures** > **p active**.
3. Click the **Tools** menu and select the **Cross section curves** option . After the curves window pops up, select the menu **Selection style** > **Individual steps**.
4. Select Phase 1. The variation of the results in the cross section is displayed in a new window as shown in Figure 160 (on page 185).
5. Do the same for Phase 2. This may take about 30 seconds which is shown in Figure 161 (on page 185).
6. The variation of the results due to different time intervals in harmonic variation at a specific cross section can be compared, see the figures below.

It can be seen that the slower variation of the external water level has a more significant influence on the pore pressures in the embankment and over a larger distance.

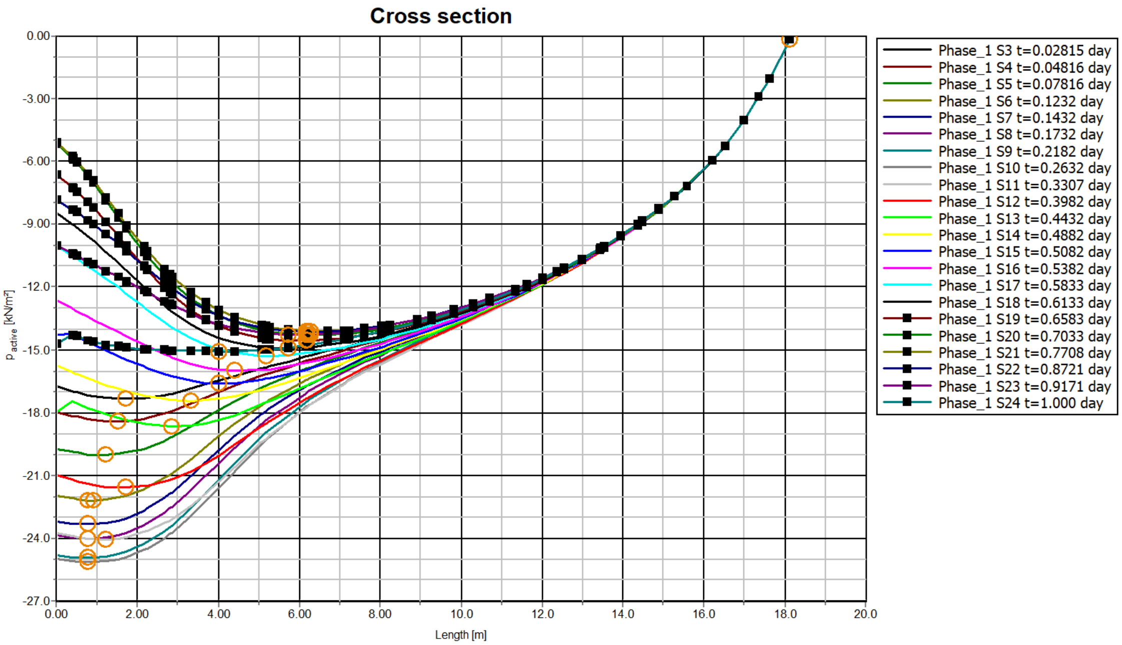



*Figure 160: Active pore pressure variation in the cross section in Phase 1*

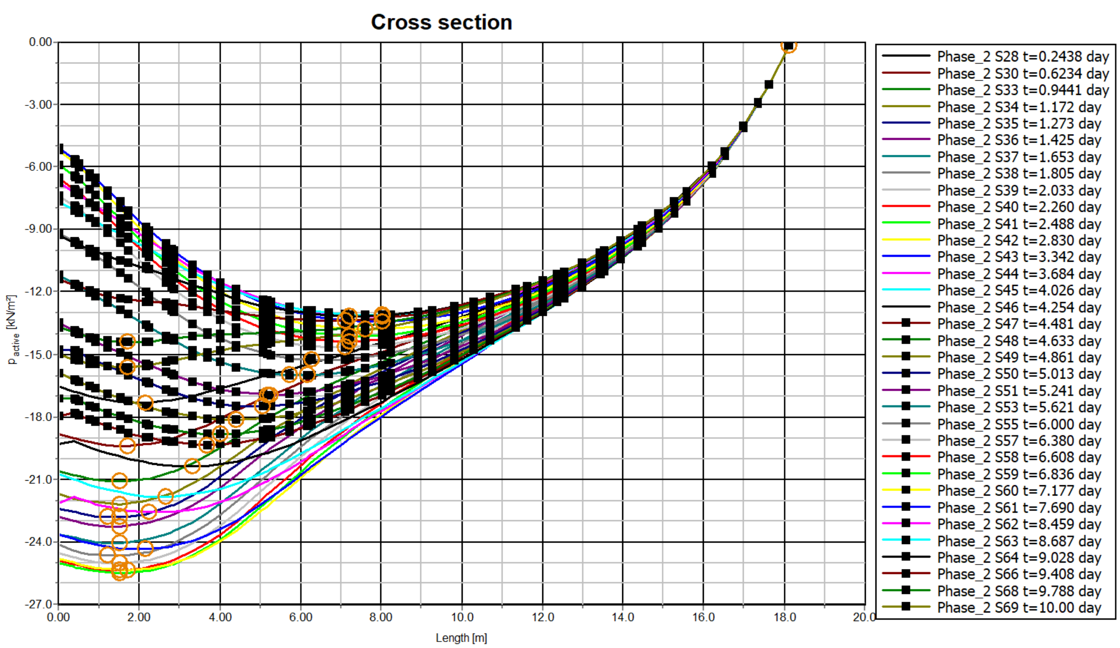



*Figure 161: Active pore pressure variation in the cross section in Phase 2*

# 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.

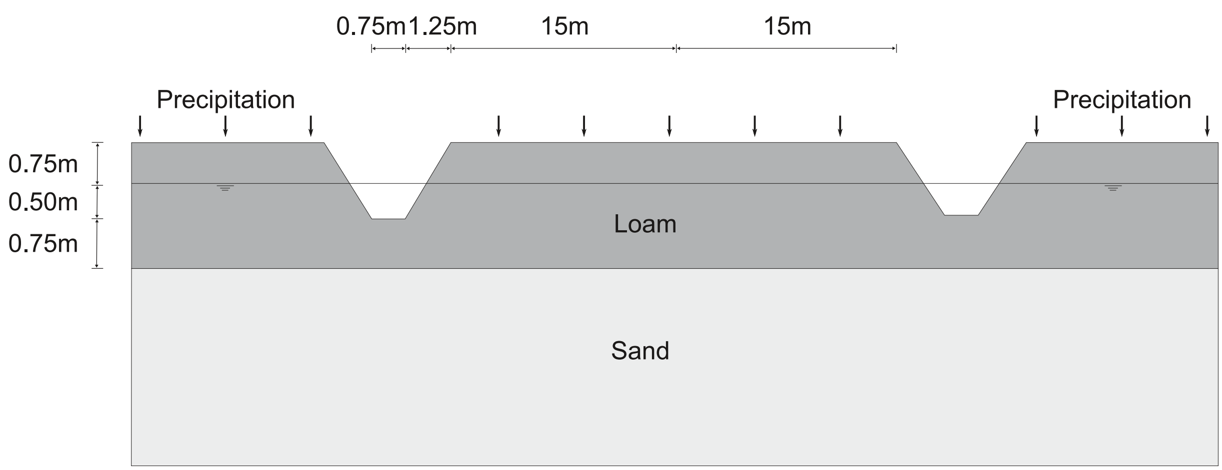



*Figure 162: Potato field geometry*

## 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).

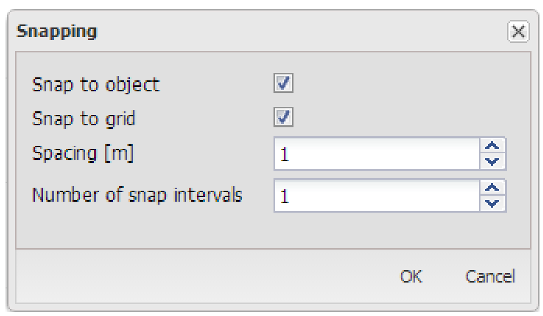



*Figure 163: Modification of the **Number of snap intervals***

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.

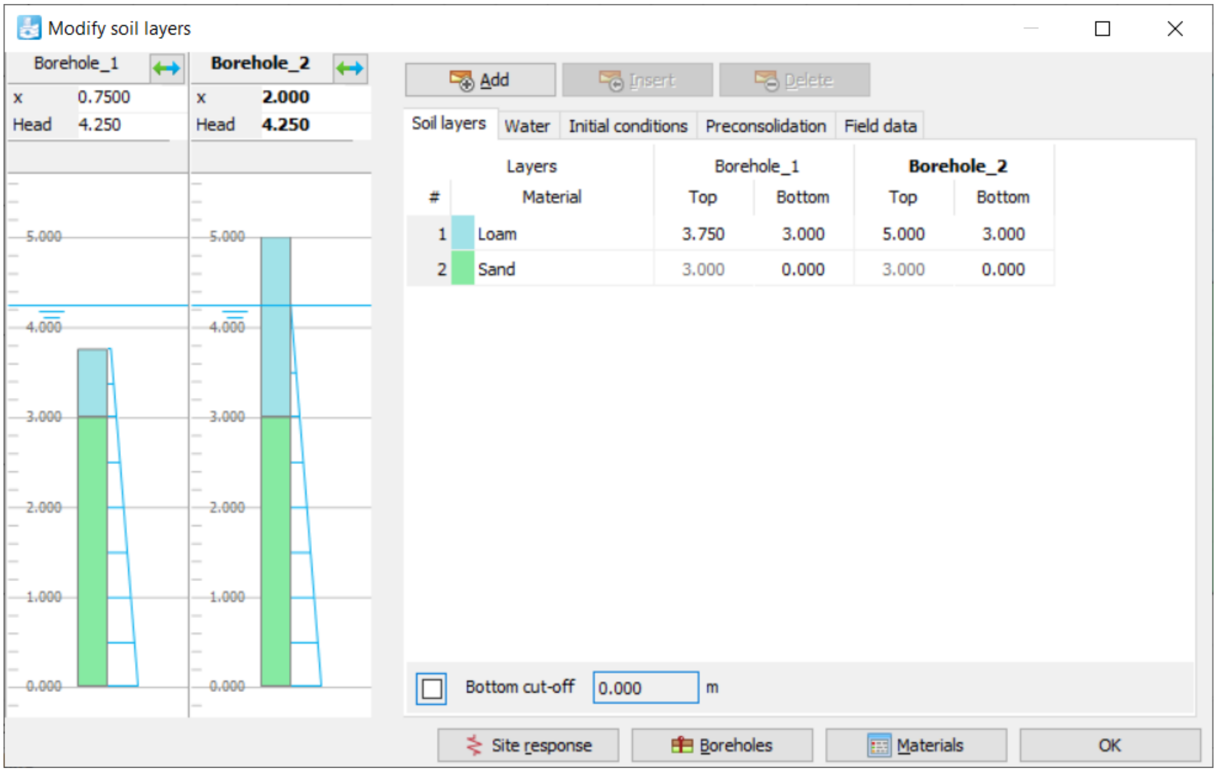



*Figure 164: Soil stratigraphy in the **Modify soil layers window***

## 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 | $\gamma_{unsat}$ | 19 | 20 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 19 | 20 | $kN/m^3$ |

| **Mechanical** | | | | |
|---|---|---|---|---|
| Stiffness | $E'_{ref}$ | $1 \cdot 10^3$ | $10 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 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).

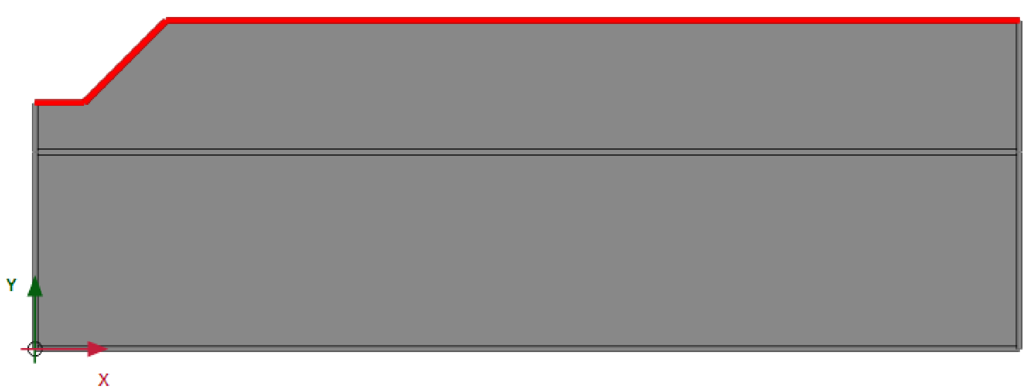



*Figure 165: The upper boundary of the model*

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).

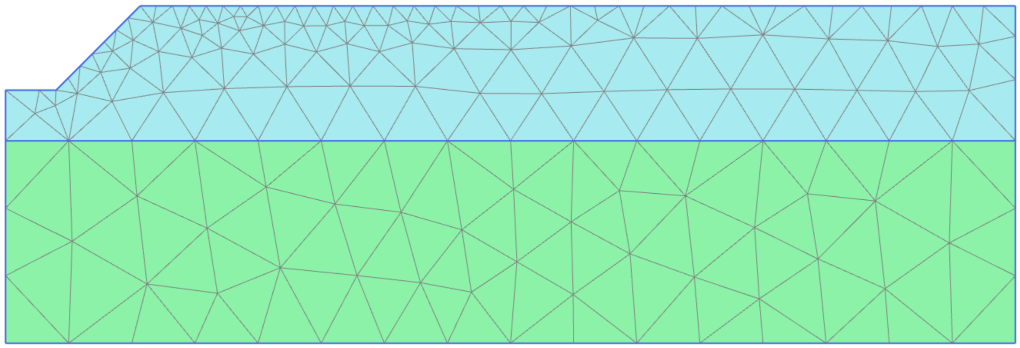


*Figure 166: The generated mesh*

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).

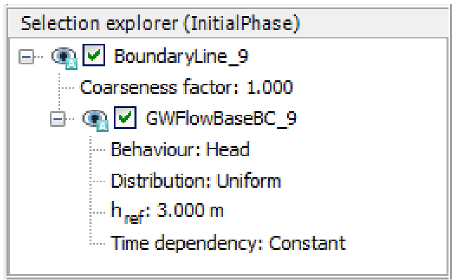



*Figure 167: Initial phase with ground water flow base*

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 [m³/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] | Δ 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.

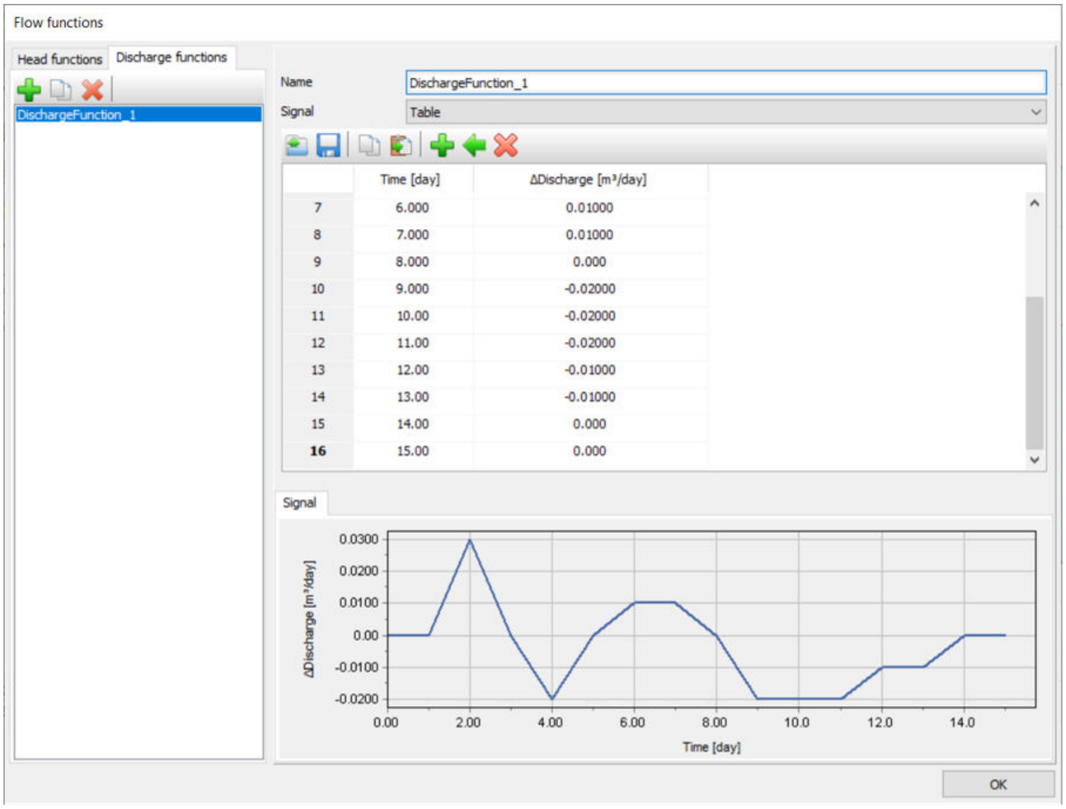



*Figure 168: The **Flow function** window displaying the precipitation data and plot*

**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 ($\psi_{min}$ = -1.0 m and $\psi_{max}$ = 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).

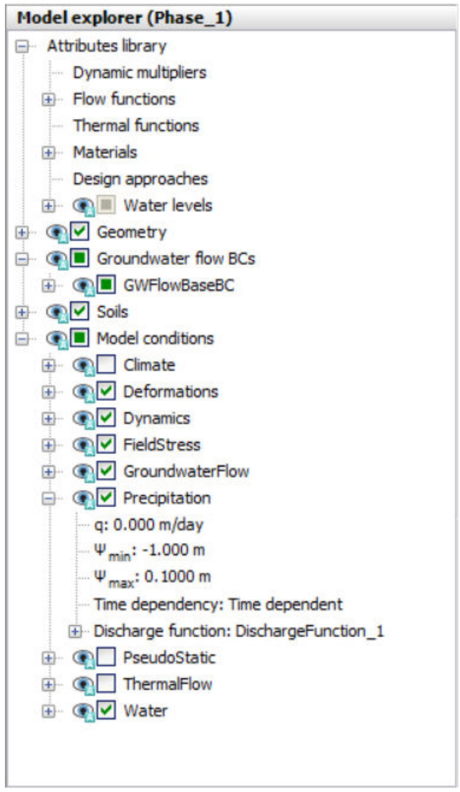



*Figure 169: Precipitation in the Model explorer*

**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).

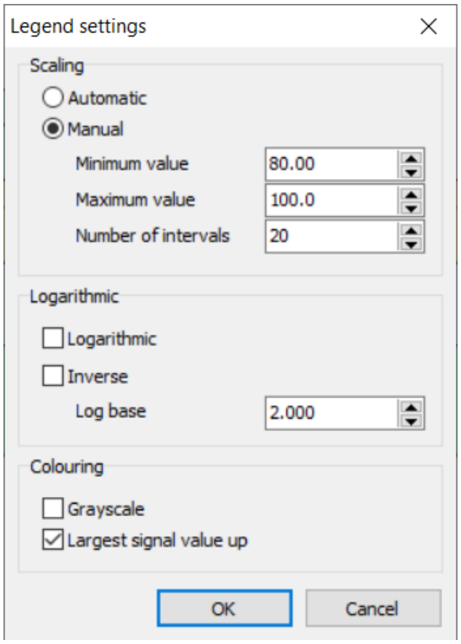



*Figure 170: Value for settings*

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

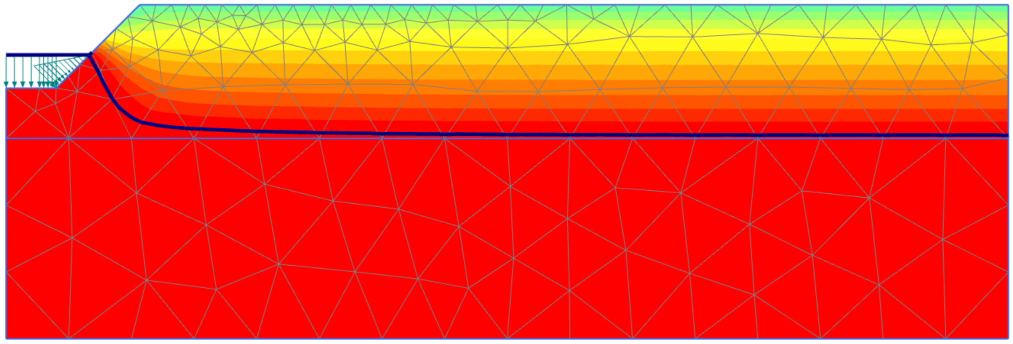


*Figure 171: Saturation field at day 15*

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.

# 13 Stability of dam under rapid drawdown [ULT]

This example concerns the stability of a reservoir dam under conditions of drawdown. Fast reduction of the reservoir level may lead to instability of the dam due to high pore water pressures that remain inside the dam. To analyse such a situation using the finite element method, a fully coupled flow-deformation analysis is required. Time-dependent pore pressure is coupled with deformations development and used in a stability analysis. This example demonstrates how coupled analysis and stability analysis can interactively be performed in PLAXIS 2D.

**Objectives**

- Defining time-dependent hydraulic conditions (Flow functions).
- Defining transient flow conditions using water levels.

**Geometry**

The dam to be considered is 30m high and the width is 172.5m at the base and 5m at the top. The dam consists of a clay core with a well graded fill at both sides. The normal water level behind the dam is 25m high. A situation is considered where the water level drops 20m. The normal phreatic level at the right hand side of the dam is 10m below ground surface. The geometry of the dam is shown in Figure 172 (on page 196).

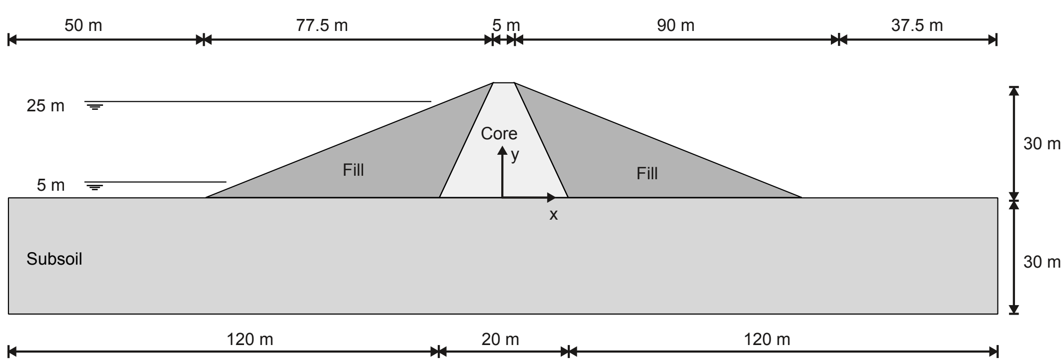



*Figure 172: Geometry of the project*

## 13.1 Create new project

To create the new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.

3. Keep the default units and constants and set the model **Contour** to $x_{min}$ = -130 m, $x_{max}$ = 130 m, $y_{min}$ = -30 m and $y_{max}$ = 30 m.

## 13.2 Define the soil stratigraphy

In order to define the underlying foundation soil, a borehole needs to be added and material properties must be assigned. A layer of 30 m overconsolidated silty sand is considered as sub-soil in the model.

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Add a soil layer extending from ground surface (y = 0) to a depth of 30 m (y = -30).

## 13.3 Create and assign material data sets

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

The layers have the following properties as shown in Table 29 (on page 197):

**Table 29: Material properties of the dam and subsoil**

| Parameter | Name | Core | Fill | Subsoil | Unit |
|---|---|---|---|---|---|
| **General** | | | | | |
| Soil model | Model | Mohr-Coulomb | Mohr-Coulomb | Mohr-Coulomb | - |
| Drainage type | Type | Undrained (B) | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 16 | 17 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 18 | 20 | 21 | kN/m$^3$ |
| **Mechanical** | | | | | |
| Young's modulus | $E'_{ref}$ | $1.5 \cdot 10^3$ | $20 \cdot 10^3$ | $50 \cdot 10^3$ | kN/m$^2$ |
| Poisson's ratio | $\nu$ | 0.35 | 0.33 | 0.3 | - |
| Cohesion | $c'_{ref}$ | - | 5 | 1 | kN/m$^2$ |
| Young's modulus increment | $E'_{inc}$ | 300 | - | - | kN/m$^2$/m |
| Undrained shear strength | $s_{u,ref}$ | 5 | - | - | kN/m$^2$ |

| | | | | | |
|---|---|---|---|---|---|
| **Mechanical** | | | | | |
| Friction angle | $\varphi'$ | - | 31 | 35 | ° |
| Dilatancy angle | $\psi$ | - | 1 | 5 | ° |
| Undrained shear strength increment | $s_{u,inc}$ | 3.0 | - | - | $kN/m^3$ |
| Reference level | $y_{ref}$ | 30 | - | - | m |
| **Groundwater** | | | | | |
| Classification type | - | Hypres | Hypres | Hypres | - |
| SWCC fitting method | - | Van Genuchten | Van Genuchten | Van Genuchten | - |
| Subsoil /Topsoil | - | Subsoil | Subsoil | Subsoil | - |
| Soil class(standard) | - | Very fine | Coarse | Coarse | - |
| Flow parameters - Use defaults | | None | None | None | - |
| Horizontal permeability | $k_x$ | $0.1 \cdot 10^{-3}$ | 1.00 | 0.01 | m/day |
| Vertical permeability | $k_y$ | $0.1 \cdot 10^{-3}$ | 1.00 | 0.01 | m/day |

To create the material sets, follow these steps:

1. Open the **Material sets** window.
2. Create data sets under the **Soil and interfaces** set type according to the information given in Table 29 (on page 197). Note that the **Thermal**, **Interfaces** and **Initial** tabsheets are not relevant (no thermal properties, no interfaces or **K0 procedure** are used).
3. Assign the **Subsoil** material dataset to the soil layer in the borehole.

## 13.4 Define the dam

The dam will be defined in the **Structures mode**.

In order to draw the dam with the mouse it is necessary to decrease the snap-to-grid distance. By default this distance is 1 m, but in this tutorial it should be 0.5m. In order to change the snap-to-grid distance, select the **Snapping options** button below the drawing area. The **Spacing** defines the distance between 2 grid points and the *Intervals* defines the amount of snap-to-grid intervals between 2 grid points. In order to have a snap-to-grid distance of 0.5m we can set either the **Spacing** to 0.5m and leave the **Intervals** to 1, or we can leave the **Spacing** at 1 m and set the amount of **Intervals** to 2.

1. Click on the **Polygon** button to define a polygon through the points located at (-80 0), (92.5 0), (2.5 30) and (-2.5 30).
2. Click again on the **Polygon** button, and this time select the **Cut polygon** button to create the sub-clusters in the dam. Define two cutting lines from (-10 0) to (-2.5 30) and from (10 0) to (2.5 30).
3. Assign the corresponding material datasets to the soil clusters.

## 13.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Fine**.
3. Click the **View mesh** button to view the mesh which is shown in Figure 173 (on page 199).

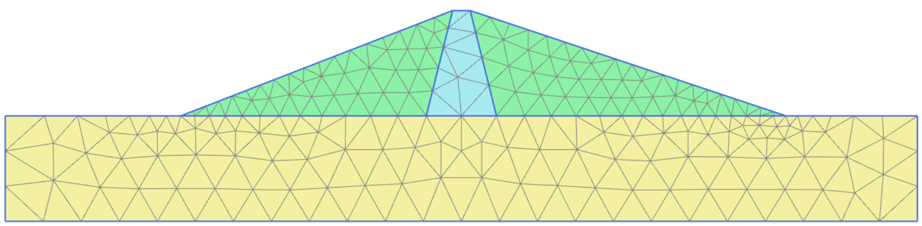


*Figure 173: The generated mesh*

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

## 13.6 Define and perform the calculation

The following cases will be considered:

- A long term situation with water level at 25m.
- A quick drop of the water level from 25 to 5m.
- A slow drop of the water from 25 to 5m.
- A long term situation with water level at 5m.

In addition to **Initial phase**, the calculation consists of eight phases. In the initial phase, initial stresses and initial pore water pressures of the dam under normal working conditions are calculated using **Gravity loading**. For this situation the water pressure distribution is calculated using a steady-state groundwater flow calculation. The first and second phases both start from the initial phase (i.e. a dam with a reservoir level at 25m) and the water level is lowered to 5 m. A distinction is made in the time interval at which this is done (i.e. different speeds of water level reduction; rapid drawdown and slow drawdown). In both cases the water pressure distribution is calculated using a fully coupled flow-deformation analysis. The third calculation phase also starts from the initial phase and considers the long-term behaviour of the dam at the low reservoir level of 5 m, which involves a steady-state groundwater flow calculation to calculate the water pressure distribution.

Finally, for all the water pressure situations the safety factor of the dam is calculated by means of phi-c reduction.

Note that only the water conditions will be defined for different calculation phases. The model requires no changes in the geometry. Water levels can be defined in the **Flow conditions mode**.

### 13.6.1 Initial phase: Dam construction & high reservoir

By default the initial phase is added in the **Phases explorer**.

1. Proceed to the **Flow conditions mode** by clicking the corresponding tab.
2. Activate **Fill** and **Core**.
3. In the **Phases explorer** double-click **Initial phase**.
   The **Phases** window is displayed.
4. In the **General** subtree specify the name of the phase (e.g. High reservoir).
5. Select the **Calculation type** > **Gravity loading** option .
6. Select the **Pore pressure calculation type** > **Steady state groundwater flow** option
7. In the **Deformation control parameters** subtree, uncheck the **Ignore suction** option.

> **Note:** Note that in this exercise we will do a fully coupled flow-deformation analysis and this type of analysis always takes into account suction in the unsaturated zone. Therefore, it is advised to also take into account suction in calculation phases prior to the fully coupled flow deformation analysis (like the initial phase in this tutorial) to avoid the unbalance in soil stresses that would occur when changing between phases with and without suctions.

The phases window after putting all the parameters is shown in Figure 174 (on page 201):

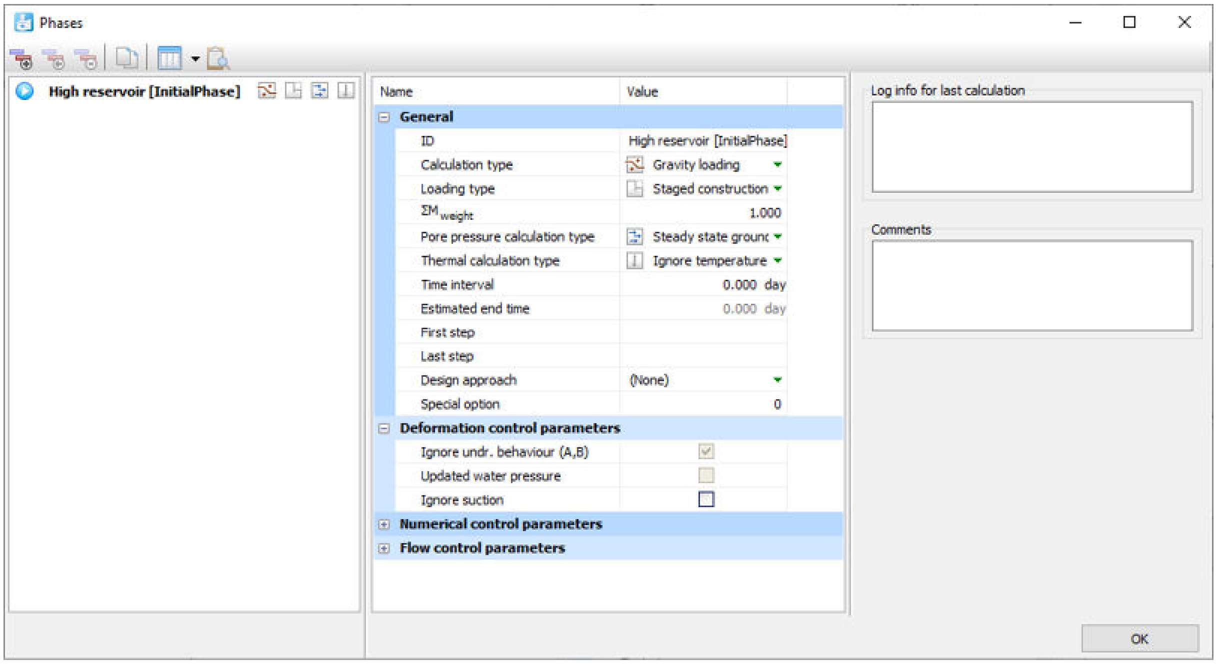



*Figure 174: The **Phases** window*

8. Click **OK** to close the **Phases** window.

**Note:** Note that by default **Undrained behaviour (A)** and **(B)** are ignored for a **Gravity loading** calculation type. The corresponding option is available in the **Deformation control parameters** subtree in the **Phases** window

9. Define the water level corresponding to the level of water in the reservoir prior to the drawdown. The water level consists of four points:

- Starting at the very left side at a level of 25 m above the ground surface (-132 25);
- Second point just inside the dam at a level of 25 m (-10 25);
- Third point near the dam toe (93 -10);
- Forth point just outside the right boundary at a level of 10 m below the ground surface (132 -10).

**Note:** Straight lines can be defined by keeping the **Shift** key pressed while defining the geometry.

The defined water level is shown in Figure 175 (on page 201)

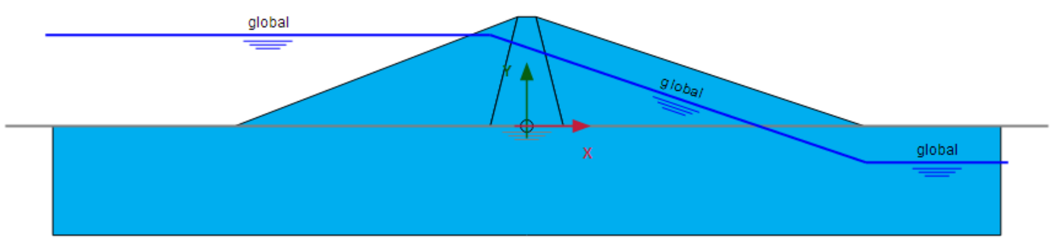



*Figure 175: High water level in the reservoir*

10. Right-click the created water level and select the **Make global** option in the appearing menu.

Note that the global water level can also be specified in the **Model Explorer** by selecting the corresponding option in the menu **Model conditions** > **Water** > **GlobalWaterLevel** > **UserWaterLevel_1**.

11. In the **Model explorer** expand the **Attributes library**.
12. Expand the **Water levels** subtree. The levels created in the **Flow conditions** mode are grouped under **User water levels**.
13. Expand the **User water levels** subtree.
    The created water level can be seen named as UserWaterLevel_1. The location of the water levels in **Model explorer** is shown in Figure 176 (on page 202)

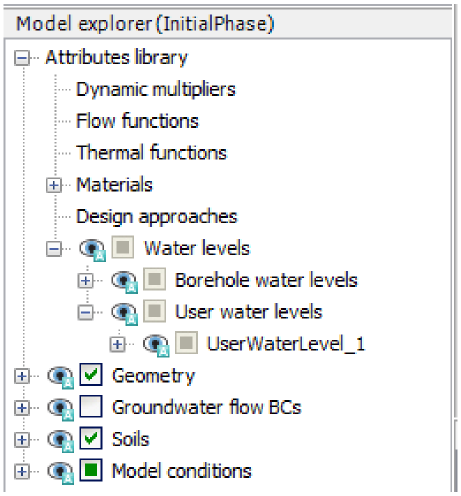



*Figure 176: Water levels in **Model explorer***

14. Double-click on the created water level and rename it as *FullReservoir_steady*. This is a distinctive name that satisfies the naming requirements (no invalid characters).
15. Expand the **Model conditions** subtree.
16. Expand the **GroundWaterFlow** subtree.

    Note that by default the boundary at the bottom of the model is set to **Closed** shown in Figure 177 (on page 203). This is relevant for this example.

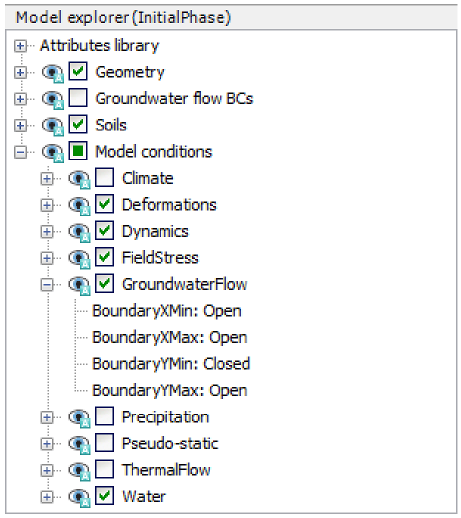



*Figure 177: **GroundwaterFlow** boundary conditions in **Model explorer***

### 13.6.2 Phase 1: Rapid drawdown

In this phase rapid drawdown of the reservoir level is considered.

1. Click the **Add phase** button to create a new phase.
2. In the **Phases explorer** double-click the newly added phase.
   The **Phases** window is displayed.
3. In the **General** subtree specify the name of the phase (e.g. Rapid drawdown). Note that the **High reservoir** phase is automatically selected in the **Start from phase** drop-down menu.
4. As **Calculation type**, select the **Fully coupled flow-deformation** option .
5. To the **Time interval** parameter assign a value of 5 days .
6. In the **Deformation control parameters** subtree, make sure that the **Reset displacements to zero** and **Reset small strain** options are selected.
7. Click **OK** to close the **Phases** window.
8. Due to the global nature of the water levels, if an attribute is assigned to a water level in the model it will affect it in all phases. The water level in this phase has the same geometry with the one previously defined, however it is time dependent and a function needs to be assigned to it. As a result, it is required to create a new water level with the same geometry and different attributes. In **Model explorer** go to the **Attributes library** > **Water levels** > **User water levels** and right-click on FullReservoir_Steady as shown in Figure 178 (on page 203). Then select the **Duplicate** option in the appearing menu.

   A copy of the water level is created.

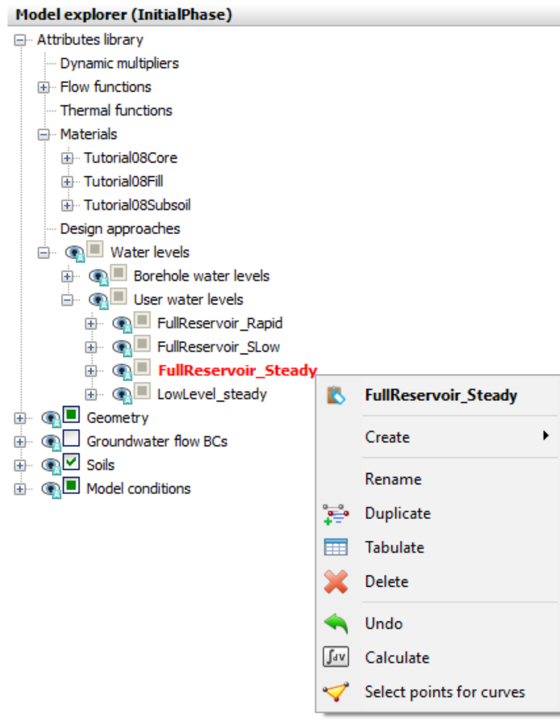



*Figure 178: Duplication of water level*

**9.** Rename the newly created water level as *FullReservoir_Rapid*.

The behaviour of the water levels can be described by specifying **Flow functions**. Note that **Flow functions** are global entities and are available in the **Model explorer** under the **Attributes library**.

To define the flow functions:

**a.** In the **Model explorer** go to the **Attributes library** and right-click the **Flow functions** option. Select the **Edit** option in the appearing menu. The **Flow functions** window is displayed.

**b.** In the **Head functions** tabsheet add a new function by clicking the corresponding button. The new function is highlighted in the list and options to define the function are displayed.

**c.** Specify a proper **Name** to the function for the rapid drawdown (e.g. Rapid).

**d.** From the **Signal** drop-down menu, select the **Linear** option .

**e.** Specify a time interval of 5 days.

**f.** As **ΔHead** assign a value of -20 m representing the amount of the head decrease.

A graph is displayed showing the defined function in Figure 179 (on page 205).

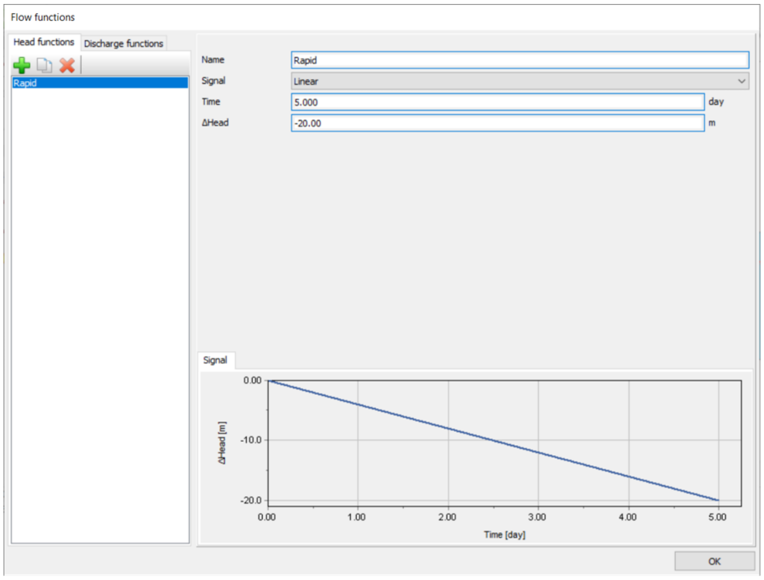



*Figure 179: The flow function for the rapid drawdown case*

**g.** Click **OK** to close the **Flow functions** window.

**10.** In the **Model explorer** right-click on **FullReservoir_Rapid** and select the **Use as global water level** option in the appearing menu.

**11.** Expand the **FullReservoir_Rapid** subtree. Note that the water level is composed of 3 water segments. Select the water segment in the upstream shoulder (left from the dam, at the reservoir side).

**12.** Expand the subtree of the selected segment and from the **TimeDependency** parameter select the **Time dependent** option.

**13.** For the **HeadFunction** parameter select the **Rapid** option.
Figure 180 (on page 206) shows the selected water segment in **Model explorer**.

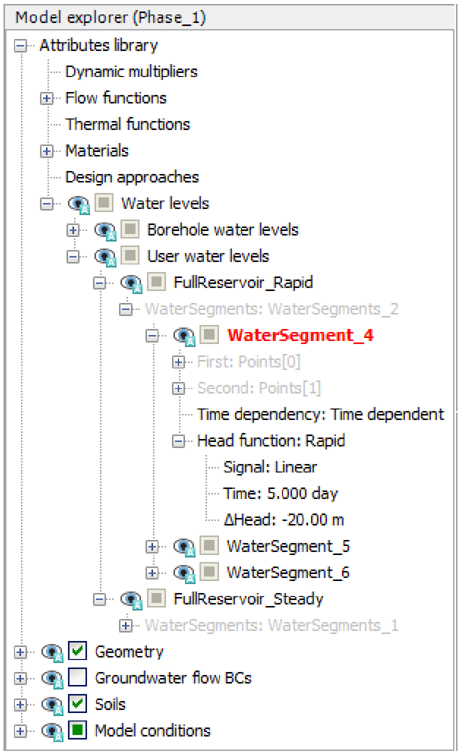



*Figure 180: Properties of the lowering water segment*

**14.** Note that in the **Model explorer** under the **Model conditions** in the **Water** subtree the **GlobalWaterLevel** gets assigned the new water level (FullReservoir_Rapid).

The configuration of the phase is shown in Figure 181 (on page 206).

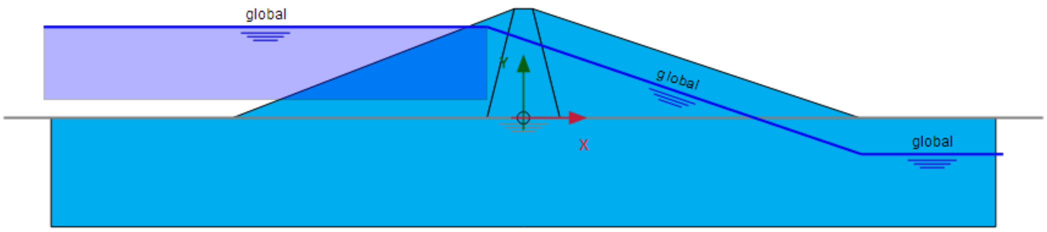



*Figure 181: Configuration of the rapid drawdown phase*

Note that the shadow under the water level segment in the upstream shoulder indicates the variation of the water level during the phase.

### 13.6.3 Phase 2: Slow drawdown

In this phase the drawdown of the reservoir level is performed at a lower rate.

1. Select the **High reservoir** phase in the **Phases explorer**.
2. Click the **Add phase** button to create a new phase.
3. In **Phases explorer** double-click the newly added phase. The **Phases** window is displayed.
4. In the **General** subtree specify the name of the phase (e.g. Slow drawdown). The **High reservoir** phase is automatically selected for the **Start from phase** parameter.
5. Select the **Fully coupled flow deformation** option as calculation type.
6. Assign a value of 50 days to the **Time interval** parameter.
7. Make sure that the **Reset displacements to zero** and **Reset small strain** options are selected in the **Deformation control parameters** subtree. The **Ignore suction** option is unchecked by default.
8. Click **OK** to close the **Phases** window.
9. Create a new duplicate of the high water level. The newly created water level will be used as **Global water level** in the slow drawdown phase. Even though the water level in this phase has the same geometry as the previously defined ones, the flow function for the time dependency is different.
10. Rename the newly created water level as 'FullReservoir_Slow'.
11. Add a new flow function following the steps described for the previous phase.
    a. Specify a proper name to the function for the slow drawdown (e.g. Slow).
    b. From the **Signal** drop-down menu select the **Linear** option.
    c. Specify a time interval of 50 days.
    d. Assign a value of -20 m to **ΔHead**, representing the amount of the head decrease. A graph is displayed showing the defined function.
    e. Click **OK** to close the **Flow functions** window as shown in Figure 182 (on page 207).

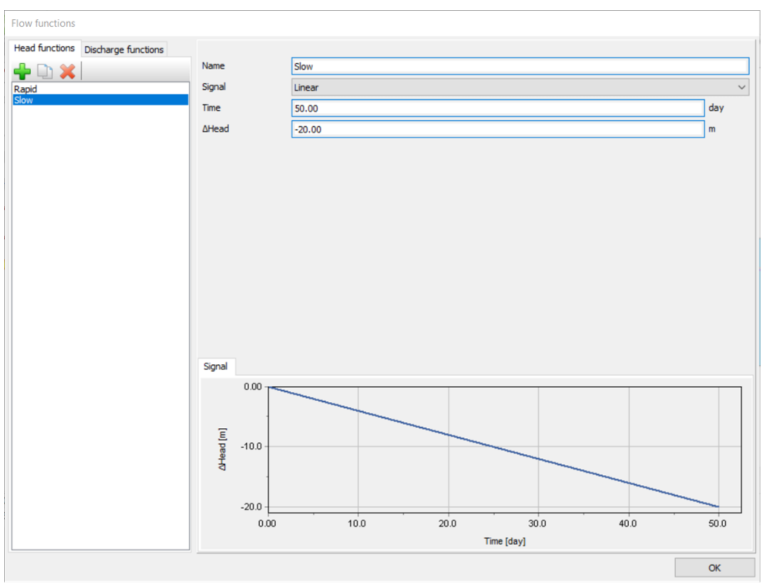



*Figure 182: The flow function for the slow drawdown case*

12. In the **Model Explorer** > **Attributes library** > **Water levels** > **User water levels** right-click on **FullReservoir_Slow** and select the **Use as global water level** option in the appearing menu.
13. Expand the **FullReservoir_Slow** subtree. Select the water segment in the upstream shoulder (left from the dam, at the reservoir side).
    The segment selected in **Model explorer** is indicated by a red colour in the model.

14. Expand the subtree of the selected segment and for the **TimeDependency** parameter select the **Time dependent** option.
15. Select the **Slow** option for the **HeadFunction** parameter.
Note that in the **Model explorer** under the **Model conditions** in the **Water** subtree the **GlobalWaterLevel** gets assigned the new water level (FullReservoir_Slow).

## 13.6.4 Phase 3: Low level

This phase considers the steady-state situation of a low reservoir level.

1. Go to the **Phases explorer** and select the **High reservoir** phase.
2. Click the **Add phase** button to create a new phase.
3. In **Phases explorer** double-click the newly added phase.
The **Phases** window is displayed.
4. In the **General** subtree specify the name of the phase (e.g. Low level). The **High reservoir** phase is automatically selected for the **Start from phase** parameter.
5. In the **Calculation type** make sure that the **Plastic** option is selected.
6. As **Pore pressure calculation type**select the **Steady state groundwater flow** option .
7. In the **Deformation control parameters** subtree select **Ignore und. behaviour (A,B).** Then make sure that the **Reset displacements to zero** and **Reset small strain** options are selected. Finally, uncheck the **Ignore suction** option.
8. Click **OK** to close the **Phases** window.
9. Define the water level corresponding to the level of water in the reservoir after the drawdown. The water level consists of four points:
   - Starting at the very left side at a level of 5 m above the ground the surface (-132 5);
   - Second point inside the dam at a level of 5 m (-60 5);
   - Third point at (93 -10);
   - Fourth point just outside the right boundary at a level of 10 m below the ground surface (132 -10).
10. Rename the newly created water level as 'LowLevel_Steady'.
11. In the **Model explorer** > **Model conditions** > **Water** assign to the **GlobalWaterLevel** the newly created water level (LowLevel_Steady).

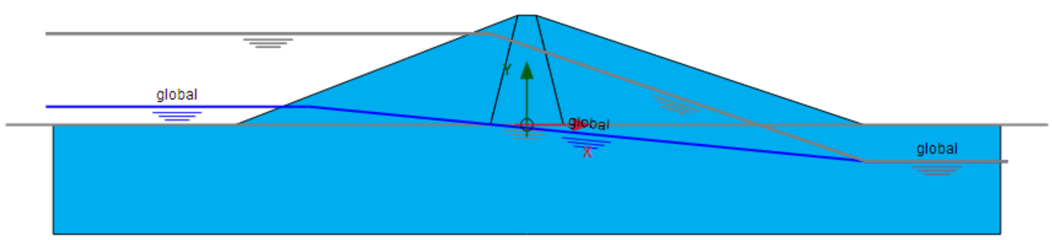



*Figure 183: Model for the low level case in the **Flow conditions** mode*

All the defined water levels are shown in Figure 183 (on page 208).

## 13.6.5 Phase 4 to 7: Safety analysis

In Phases 4 to 7 stability calculations are defined for the previous phases.

1. Select the parent phase in the **Phases explorer**.
2. Click the **Add phase** button to create a new phase. Proceed to the **Phases** window.
3. Set **Calculation type** to **Safety** .
4. In the **Deformation control** subtree, select **Reset displacements to zero**.
5. In the **Deformation control parameters** subtree, check the **Ignore suction** option for all the safety analyses.

> **Note:** Taking suction into account in a **Safety** phase gives a higher factor of safety, hence ignoring suction in a **Safety** phase is more conservative. In the **Safety** analysis of PLAXIS 2D, any unbalance due to changing from suction to no suction is first solved before the factor of safety is determined. As a result, $\Sigma M_{sf}$ can decrease in the first part of the calculation.

6. The final view of the phases explorer window is shown in Figure 184 (on page 209).

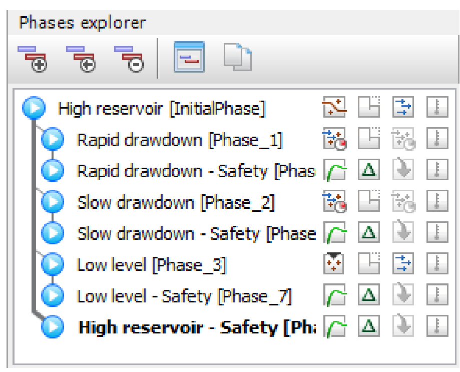



*Figure 184: The final view of Phases explorer*

## 13.6.6 Execute the calculation

1. Proceed to the **Staged construction** mode.
2. Select nodes located at the crest (-2.5 30) and at the toe of the dam (-80 0).
3. Click the **Calculate** button to calculate the project and ignore the warnings regarding the influence of suction in the **Safety** analysis..
4. Save the project after the calculation has finished.

# 13.7 Results

The results of the four groundwater flow calculations in terms of pore pressure distribution are shown in the figures below. Four different situations were considered:

- The steady-state situation with a high (standard) reservoir level is shown in Figure 185 (on page 210).

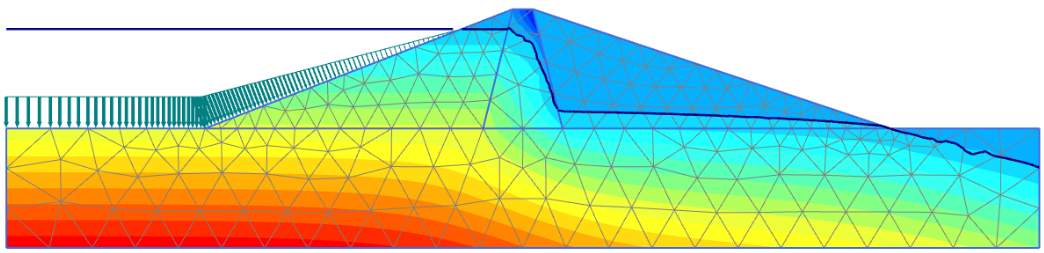


*Figure 185: Pore pressure distribution, ($p_{active}$), for high reservoir level*

- The pore pressure distribution after rapid drawdown of the reservoir level is shown in Figure 186 (on page 210).

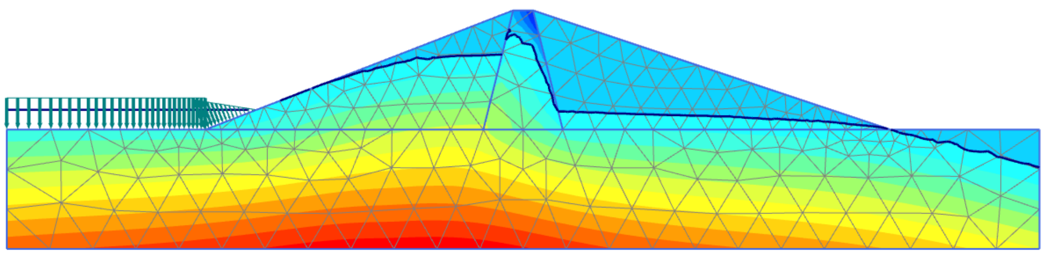


*Figure 186: Pore pressure distribution, ($p_{active}$), after rapid drawdown*

- The pore pressure distribution after slow drawdown of the reservoir level is shown in Figure 187 (on page 210).

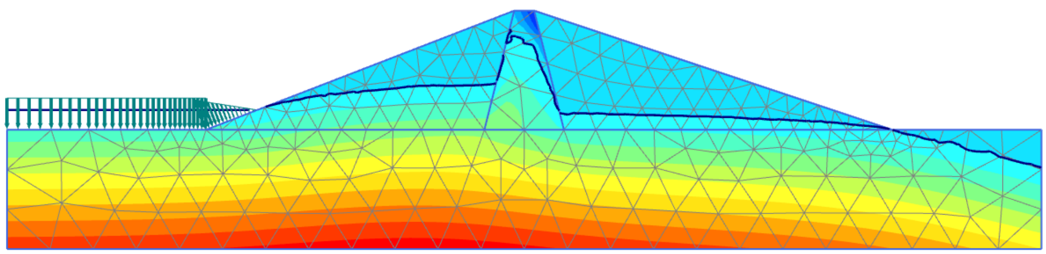


*Figure 187: Pore pressure distribution, ($p_{active}$), after slow drawdown*

- The steady-state situation with a low reservoir level is shown in Figure 188 (on page 211).

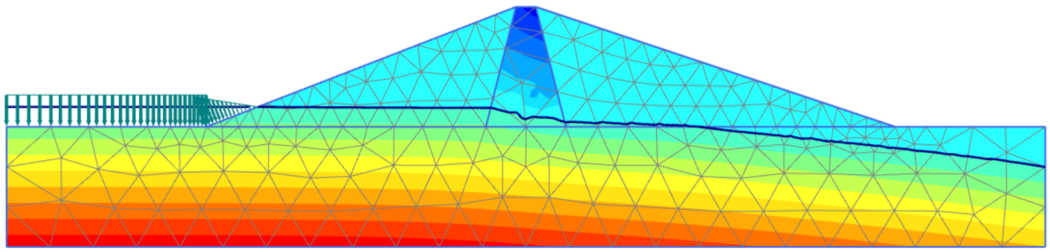


*Figure 188: Pore pressure distribution, ($p_{active}$), for low reservoir level*

**Note:**

The phreatic level can be smoother with a high refinement of the mesh in the core.

When the change of pore pressure is taken into account in a deformation analysis, some additional deformation of the dam will occur. These deformations and the effective stress distribution can be viewed on the basis of the results of the first four calculation phases. Here, attention is focused on the variation of the safety factor of the dam for the different situations. Therefore, the development of **ΣMsf** is plotted for the phases 4 to 7 as a function of the displacement of the dam crest point (-2.5 30.0), see Figure 189 (on page 211).

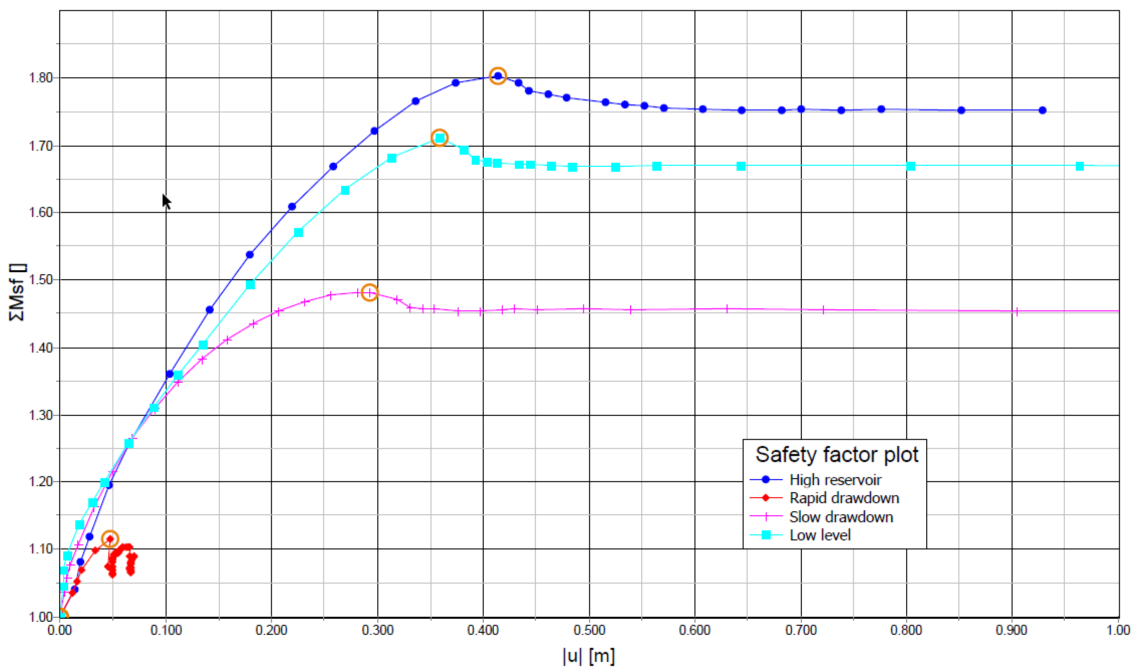



*Figure 189: Safety factors for different situations*

Rapid drawdown of a reservoir level can reduce the stability of a dam significantly. Fully coupled flow-deformation and stability analysis can be performed with PLAXIS 2D to effectively analyse such situations.

**Note:**

In the strength reduction method the factor of safety is considered the value of the strength reduction factor **ΣMsf** for which progressive failure occurs. Hence, a **Safety** analysis should be performed with sufficient steps to make sure that failure really occurs. This can be checked from the graph of displacement vs **ΣMsf** where the curve flattens: the strength can no longer be reduced while the displacement rapidly increase.

**Note:**

By removing the suction in the safety analysis, an out-of-balance force is introduced at the beginning of the calculation. Without the contribution of the suction, the **ΣMsf** can decrease in the first part of the calculation.

# 14 Dynamics analysis of a generator on an elastic foundation [ULT]

Using PLAXIS 2D, it is possible to simulate dynamic soil-structure interaction. Here the influence of a vibrating source on its surrounding soil is studied. Oscillations caused by the generator are transmitted through the footing into the subsoil.

The physical damping due to the viscous effects is taken into consideration via the Rayleigh damping. Also, due to axisymmetry 'geometric damping' can be significant in attenuating the vibration.

The modelling of the boundaries is one of the key points. In order to avoid spurious wave reflections at the model boundaries (which do not exist in reality), special conditions have to be applied in order to absorb waves reaching the boundaries.

**Objectives**

- Defining a **Dynamic** calculation
- Defining dynamic loads
- Defining dynamic boundary conditions (viscous)
- Defining material damping by means of Rayleigh damping

**Geometry**

The vibrating source is a generator founded on a 0.2 m thick concrete footing of 1 m in diameter as shown in Figure 190 (on page 213).

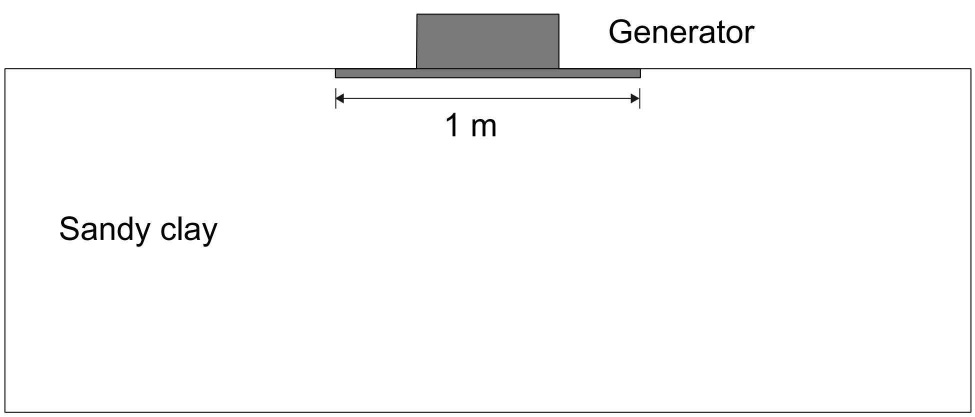



*Figure 190: Generator founded on elastic subsoil*

# 14.1 Create new project

To create the 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. Due to the three dimensional nature of the problem, an axisymmetric model is used. In the **Model** tabsheet select the **Axisymmetric** option for **Model** and keep the default option for **Elements** (**15-Noded**).
4. Keep the default values for units and constants and set the model contour to $x_{min}$ = 0 m, $x_{max}$ = 20 m, $y_{min}$ = -10 m and $y_{max}$ = 0 m.

> **Note:** The model boundaries should be sufficiently far from the region of interest, to avoid disturbances due to possible reflections. Although special measures are adopted in order to avoid spurious reflections (viscous boundaries), there is always a small influence and it is still a good habit to put boundaries far away. In a dynamics analysis, model boundaries are generally taken further away than in a static analysis.

# 14.2 Define the soil stratigraphy

The subsoil consists of one layer with a depth of 10 m. The ground level is defined at y = 0. Note that water conditions are not considered in this example. To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
2. Create a soil layer extending from ground surface (y = 0) to a depth of 10 m (y = -10).
3. Keep the **Head** in the borehole at 0 m. This means that the sub-soil is fully saturated.

# 14.3 Create and assign material data sets

The soil layer consists of sandy clay, which is assumed to be elastic. Create a data set under the **Soil and interfaces** set type according to the information given in Table 30 (on page 215). The specified Young's modulus seems relatively high. This is because the dynamic stiffness of the ground is generally considerably larger than the static stiffness, since dynamic loadings are usually fast and cause very small strains.

**Table 30: Material properties of the material**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | Model | Linear elastic | - |
| Drainage type | Type | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 20 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | kN/m$^3$ |
| **Mechanical** | | | |
| Stiffness | $E'_{ref}$ | $50 \cdot 10^3$ | kN/m$^2$ |
| Poisson's ratio | $\nu'$ | 0.3 | - |
| **Initial** | | | |
| $K_0$ determination | - | Automatic | - |
| Lateral earth pressure coefficient | $K_{0,x}$ | 0.50 | - |

**Note:** When using Mohr-Coulomb or linear elastic models the wave velocities $V_p$ and $V_s$ are calculated from the elastic parameters and the soil weight. $V_p$ and $V_s$can also be entered as input; the elastic parameters are then calculated automatically. See also Elastic parameters and the Wave Velocity relationships in the *Mechanical Tabsheet* of the Reference Manual.

## 14.4 Define the structural elements

The generator is defined in the **Structures mode**.

**Table 31: Material properties of the footing**

| Parameter | Name | Value | Unit |
|---|---|---|---|
| Material type | - | Elastic | - |
| Weight | $w$ | 5 | kN/m/m |
| Isotropic | - | Yes | - |
| Axial stiffness | $EA_1$ | $7.6 \cdot 10^6$ | kN/m |

| Parameter | Name | Value | Unit |
|---|---|---|---|
| Flexural rigidity | *EI* | $24 \cdot 10^3$ | $kNm^2/m$ |
| Poisson's ration | $\nu$ | 0 | - |

1. Create a plate extending from (0 0) to (0.5 0) to represent the footing.
2. Define a material data set for the footing according to the information given in Table 31 (on page 215). The footing is assumed to be elastic and has a weight of 5 $kN/m^2$.
3. Apply a distributed load on the footing to model the weight of the generator as well as the vibrations that it produces. The actual value of the load will be defined later.
   The model is shown in Figure 191 (on page 216):

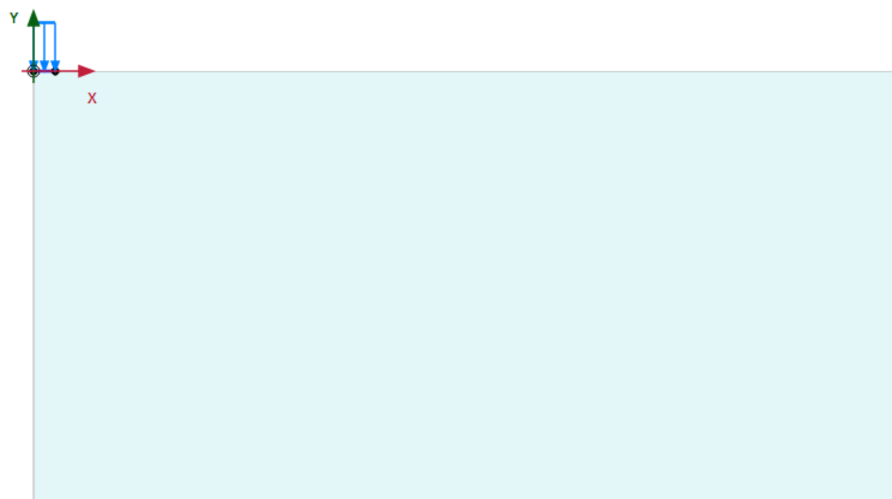



*Figure 191: Model layout*

## 14.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button to generate the mesh. Use the default option for the **Element distribution** parameter (**Medium**).
3. Click the **View mesh** button to view the mesh.

   The resulting mesh is shown in Figure 192 (on page 217) . Note that the mesh is automatically refined under the footing.

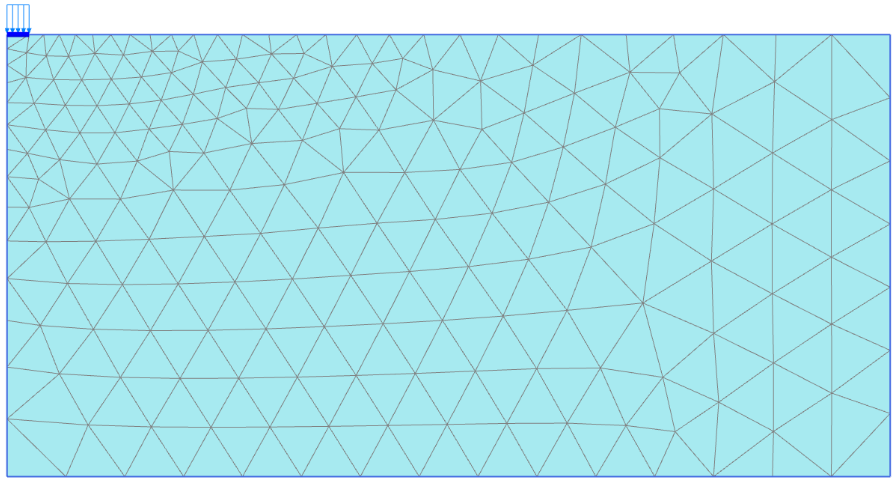


*Figure 192: The generated mesh*

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

# 14.6 Define and perform the calculation

The calculation consists of 4 phases and it will be defined in the **Staged construction mode** .

## 14.6.1 Initial phase

1. Click the **Staged construction** tab to proceed with the definition of the calculation phases.
2. The initial phase has already been introduced. The default settings of the initial phase will be used in this tutorial.

## 14.6.2 Phase 1: Footing

1. Click the **Add phase** button  to create a new phase. The default settings of the added phase will be used for this calculation phase.
2. Activate the footing.
3. Activate the static component of the distributed load. In the **Selection explorer** set $q_{y,start,ref}$ value to -8 kN/m/m. Do not activate the dynamic component of the load as shown in Figure 193 (on page 218).

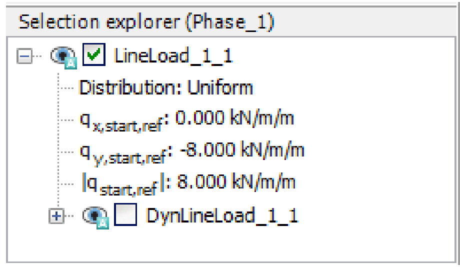



*Figure 193: Specification of the static load component in the* ***Selection explorer***

## 14.6.3 Phase 2: Start generator

In this phase, a vertical harmonic load, with a frequency of 10 Hz and amplitude of 10 kN/m$^2$, is applied to simulate the vibrations transmitted by the generator. Five cycles with a total time interval of 0.5 sec are considered.

1. Click the **Add phase** button to create a new phase.
2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as calculation type.
3. Set the **Dynamic time interval** parameter to 0.5 s.
4. In the **Deformation control parameters** subtree in the **Phases** window select the **Reset displacements to zero** parameter. The default values of the remaining parameters will be used for this calculation phase.
5. In the **Model explorer** expand the **Attributes library** subtree.
6. Right-click the **Dynamic multipliers** subtree and select the **Edit** option from the appearing menu. The **Multipliers** window pops up.
7. Click the **Load multipliers** tab.
8. Click the **Add** button to introduce a multiplier for the loads.
9. Define a **Harmonic** signal with an **Amplitude** of 10, a **Phase** of 0° and a **Frequency** of 10 Hz and as shown in Figure 194 (on page 219):

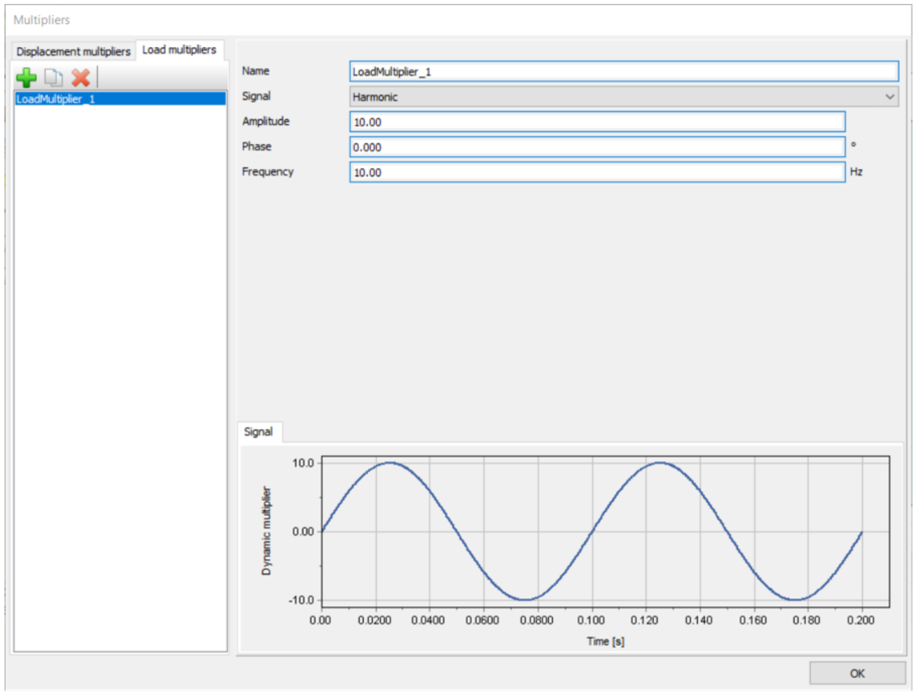
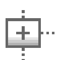



*Figure 194: Definition of a **Harmonic** multiplier*

10.  In the **Selection explorer**, activate the dynamic component of the load (DynLineLoad_1).
11. Specify the components of the load as ($q_{x, start, ref}$, $q_{y, start, ref}$) = (0.0, -1.0). Click Multiplier_y in the dynamic load subtree and select the LoadMultiplier_1 option from the drop-down menu as shown in Figure 195 (on page 220).

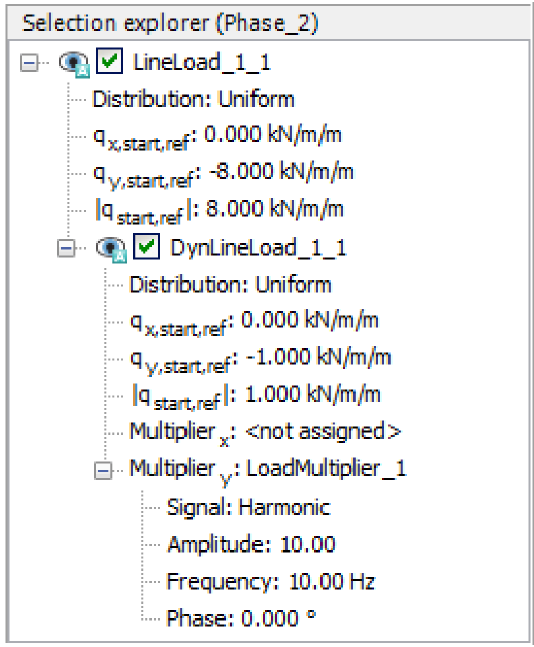



*Figure 195: Specification of the dynamic load component in the **Selection explorer***

> **Note:** The dynamic multipliers can be defined in the **Geometry** modes as well as in the **Calculation** modes.

**12.** Special boundary conditions have to be defined to account for the fact that in reality the soil is a semi-infinite medium. Without these special boundary conditions the waves would be reflected on the model boundaries, causing perturbations. To avoid these spurious reflections, viscous boundaries are specified at Xmax and Ymin. The dynamic boundaries can be specified in the **Model explorer** > **Model conditions** > **Dynamics** subtree as shown in Figure 196 (on page 221).

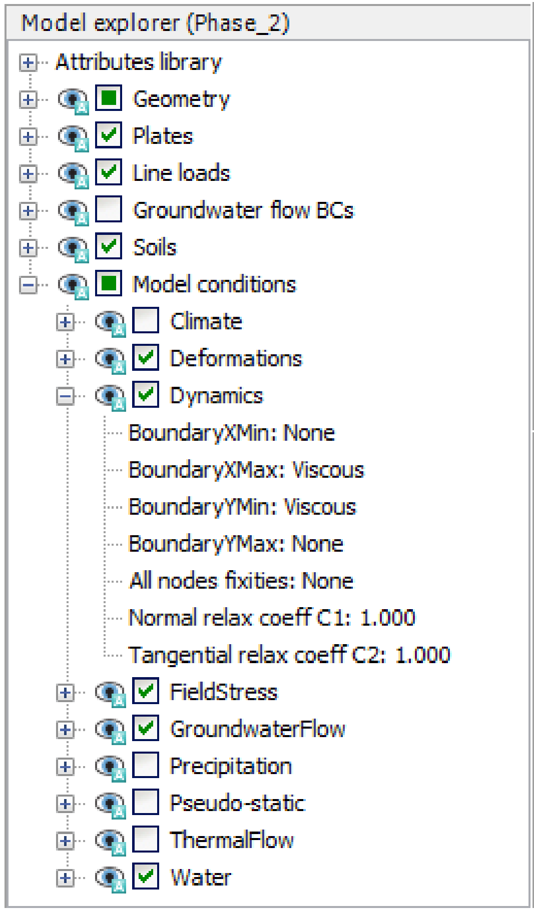



*Figure 196: Boundary conditions for* ***dynamics calculations***

## 14.6.4 Phase 3: Stop generator

1. Click the **Add phase** button to create a new phase.
2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as calculation type.
3. Set the **Dynamic time interval** parameter to 0.5 s.
4. In the **Staged construction** mode deactivate the dynamic component of the surface load. Note that the static load is still active. The dynamic boundary conditions of this phase should be the same as in the previous phase.

## 14.6.5 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar and select nodes located at the ground surface, for instance at (1.4 0), (1.9 0) and (3.6 0), to consider in curves.
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the Save button .

## 14.6.6 Additional calculation with damping

In a second calculation, material damping is introduced by means of Rayleigh damping. Rayleigh damping can be entered in the material data set. The following steps are necessary:

1. Save the project under another name.
2. Open the material data set of the soil.
3. In the **General** tabsheet **Rayleigh damping** parameter is displayed .Set the **Input method** to **SDOF Equivalent.**
4. In order to introduce 5% of material damping, set the value of the **ξ** parameter to **5%** for both targets as $\xi_1$ and $\xi_2$ and set the frequency values to **1** and **10** for $f_1$ and $f_2$ respectively.
5. The values of **α** and **β** are automatically calculated by the program as shown in Figure 197 (on page 223)..

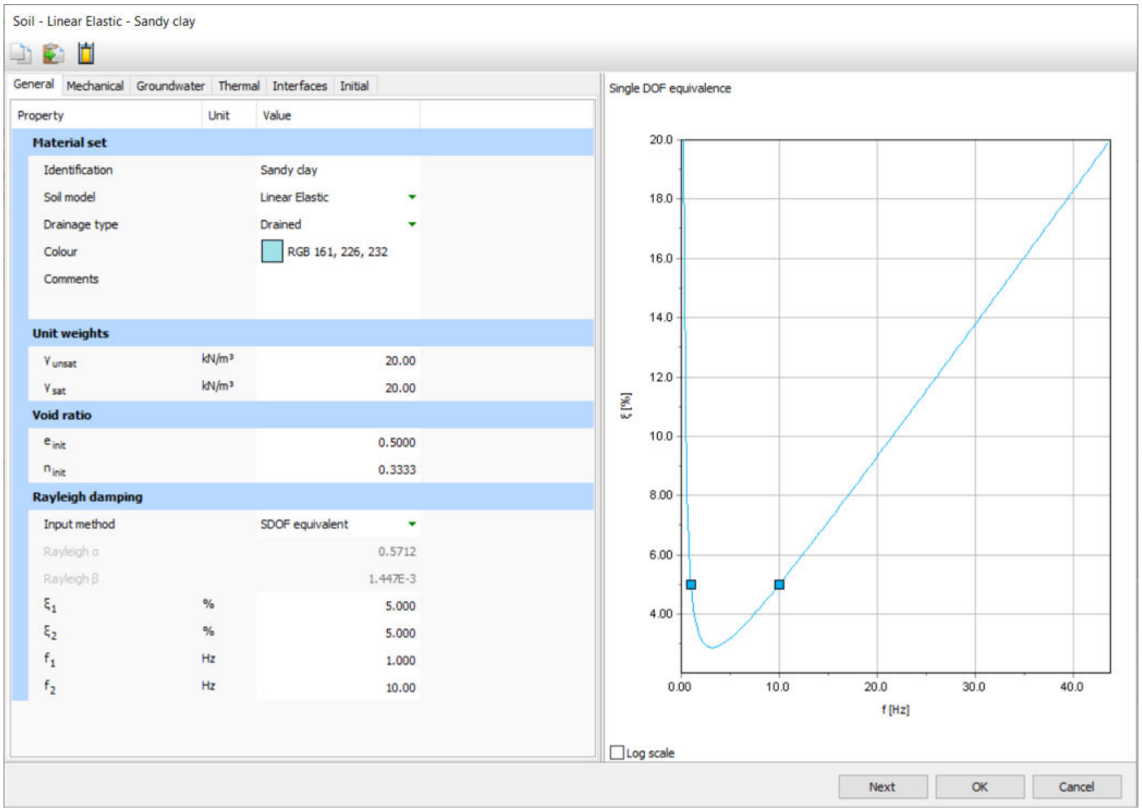



*Figure 197: Input of Rayleigh damping*

6. Click **OK** to close the data base.
7. Check whether the phases are properly defined (according to the information given before) and start the calculation.

## 14.7 Results

The **Curve generator** feature is particularly useful for dynamics analysis. You can easily display the actual loading versus time (input) and also displacements, velocities and accelerations of the pre-selected points versus time. The evolution of the defined multipliers with time can be plotted by assigning **Dynamic time** to the x-axis and $u_y$ to the y-axis.

Figure 198 (on page 224) shows the response of the pre-selected points at the surface of the structure. It can be seen that even with no damping, the waves are dissipated which can be attributed to the geometric damping.

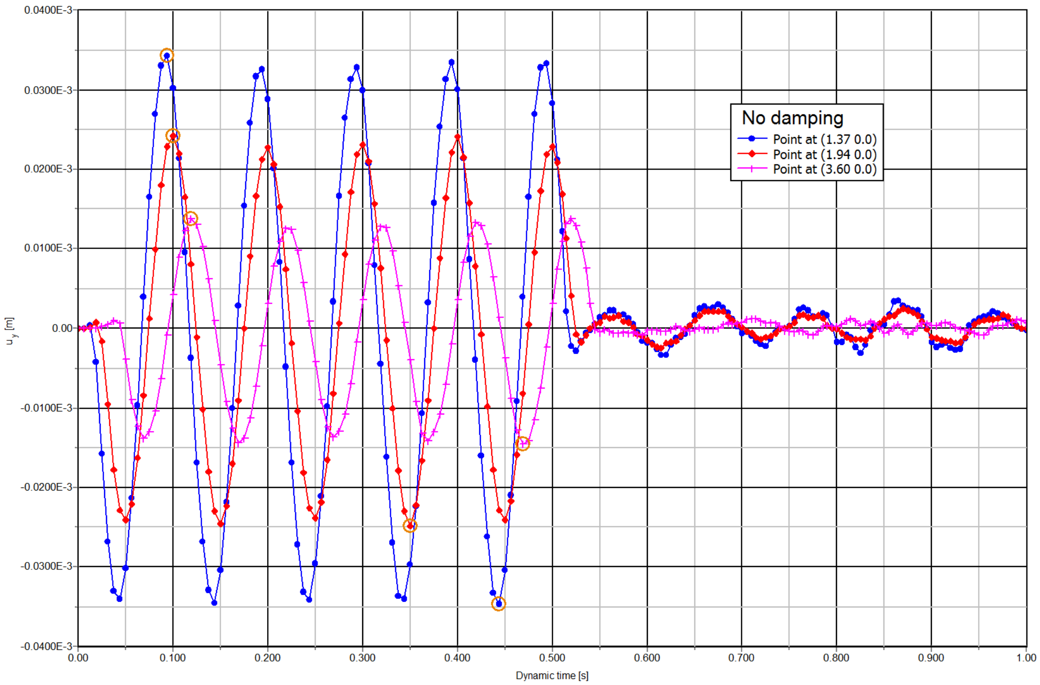



*Figure 198: Vertical displacement vs time on the surface at different distances to the vibrating source (without damping)*

The presence of damping is clear in Figure 199 (on page 225).

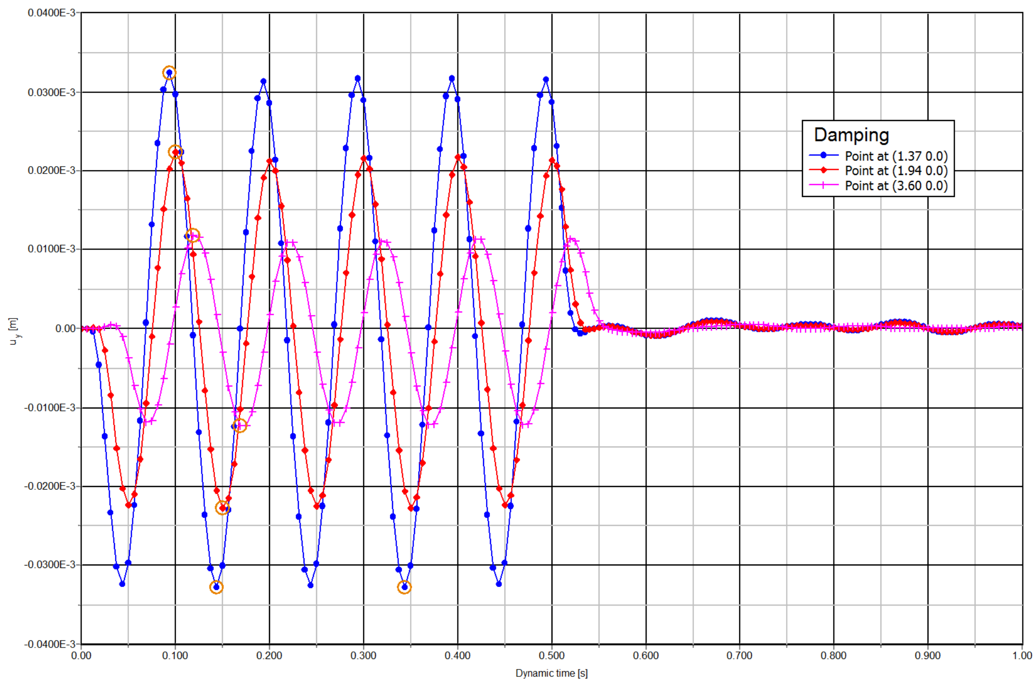



*Figure 199: Vertical displacement vstime on the surface at different distances to the vibrating source (with damping)*

It can be seen that the vibration is totally seized when some time is elapsed after the removal of the force (at t = 0.5 s). Also, the displacement amplitudes are lower. Compare the curves without and with damping.

It is possible in the Output program to display displacements, velocities and accelerations at a particular time, by choosing the appropriate option in the **Deformations** menu. Figure 200 (on page 225) shows the total accelerations in the soil at the end of phase 2 (t = 0.5 s).

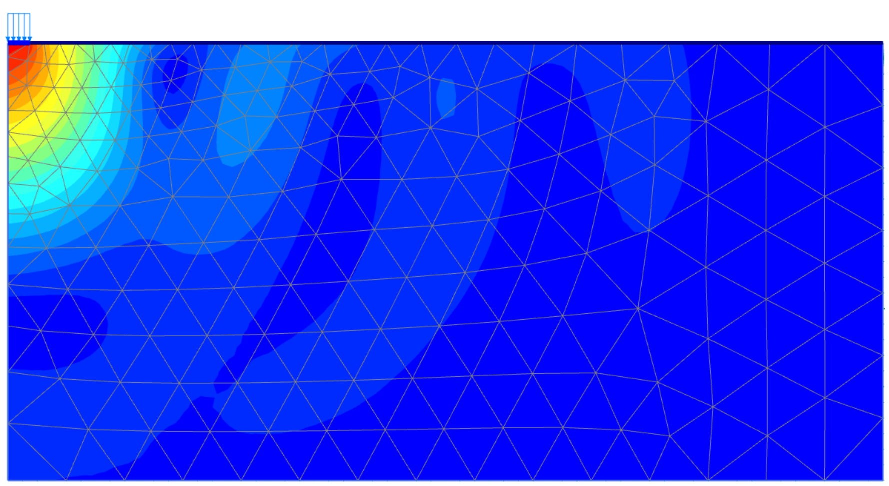


*Figure 200: Acceleration (**|a|**) in the soil at the end of phase 2 (with damping)*

# 15
# Pile driving [ULT]

Pile driving is a dynamic process that causes vibrations in the surrounding soil. Moreover, excess pore pressures are generated due to the quick stress increase around the pile.

In this example focus is put on the irreversible deformations below the pile. In order to simulate this process most realistically, the behaviour of the sand layer is modelled by means of the Hardening Soil model with small-strain stiffness.

**Geometry**

This example involves driving a concrete pile through an 11 m thick clay layer into a sand layer. The pile has a diameter of 0.4 m which is displayed in Figure 201 (on page 226).

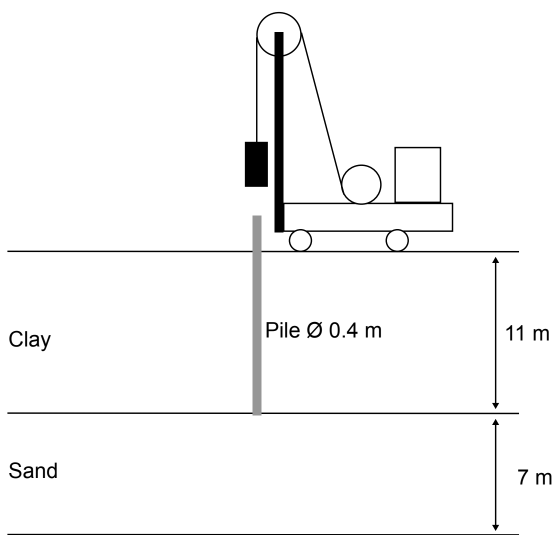



*Figure 201: Pile driving situation*

## 15.1 Create new project

To create the new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.

3. In the **Model** tabsheet select the **Axisymmetry** option for **Model** and keep the default option for **Elements** (**15-Noded**).
4. Keep the default values for units and constants and set the model **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 30 m, $y_{min}$ = 0 m and $y_{max}$ = 18 m.

## 15.2 Define the soil stratigraphy

The subsoil is divided into an 11 m thick clay layer and a 7 m thick sand layer. The phreatic level is assumed to be at the ground surface. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line. To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
2. Create two soil layers extending from y = 18 to y = 7 and from y = 7 to y = 0.
3. Set the **Head** in the borehole at 18 m.

## 15.3 Create and assign material data sets

The clay layer is modelled with the Mohr-Coulomb model. The behaviour is considered to be **Undrained (B)**. An interface strength reduction factor is used to simulate the reduced friction along the pile shaft.

In order to model the non-linear deformations below the tip of the pile in a right way, the sand layer is modelled by means of the Hardening Soil model with small-strain stiffness. Because of the fast loading process, the sand layer is also considered to behave undrained. The short interface in the sand layer does not represent soil-structure interaction. As a result, the interface strength reduction factor should be taken equal to unity (rigid).

The layers have the following properties:

**Table 32: Material properties of the clay layer and the pile**

| Parameter | Name | Clay | Pile | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | Model | Mohr-Coulomb | Linear elastic | - |
| Drainage type | Type | Undrained (B) | Non-porous | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 24 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 18 | - | kN/m$^3$ |

| | | | | |
|---|---|---|---|---|
| **Mechanical** | | | | |
| Young's modulus (constant) | $E'_{ref}$ | $5.0 \cdot 10^3$ | $30 \cdot 10^6$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 0.3 | 0.1 | - |
| Young's modulus increment | $E'_{inc}$ | $1.0 \cdot 10^3$ | - | $kN/m^2$ |
| Reference level | $y_{ref}$ | 18 | - | m |
| Undrained shear strength | $s_{u,ref}$ | 5.0 | - | $kN/m^2$ |
| Undrained shear strength increment | $s_{u,inc}$ | 3 | - | $kN/m^3$ |
| Reference level | $y_{ref}$ | 18 | - | m |
| Cohesion | $c'_{ref}$- | - | - | $kN/m^2$ |
| **Interface** | | | | |
| Strength determination | Type | Manual | Rigid | - |
| Interface reduction factor | $R_{inter}$ | 0.5 | 1.0 | - |
| **Initial** | | | | |
| $K_0$ determination | - | Automatic | Automatic | - |
| Lateral earth pressure coefficient | $K_{0,x}$ | 0.5000 | 0.5000 | - |

**Table 33: Material properties of the sand layer**

| Parameter | Name | Sand | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | Model | HS small | - |
| Drainage type | Type | Undrained (A) | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 17 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | $kN/m^3$ |

| Mechanical | | | |
|---|---|---|---|
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $50 \cdot 10^3$ | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $50 \cdot 10^3$ | $kN/m^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $150 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu_{ur}$ | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | - |
| Shear modulus at very small strains | $G_0{}^{ref}$ | $120 \cdot 10^3$ | $kN/m^2$ |
| Shear strain at which $G_s$ = 0.722 $G_0$ | $\gamma_{0.7}$ | $0.1 \cdot 10^{-3}$ | - |
| Cohesion | $c'_{ref}$ | 0 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 31 | ° |
| Dilatancy angle | $\psi$ | 0 | ° |
| **Interface** | | | |
| Strength determination | - | Rigid | - |
| **Initial** | | | |
| $K_0$ determination | - | Automatic | - |

1. Create the material data sets according to Table 32 (on page 227) and Table 33 (on page 228)

## 15.4 Define the structural elements

The pile is defined as a column of 0.2 m width. The **Interface** elements are placed along the pile to model the interaction between the pile and the soil. The interface should be extended to about half a meter into the sand layer. Note that the interface should be defined only at the side of the soil. A proper modelling of the pile-soil interaction is important to include the material damping caused by the sliding of the soil along the pile during penetration and to allow for sufficient flexibility around the pile tip.

**Note:** Use the **Zoom in** feature to create the pile and the interface.

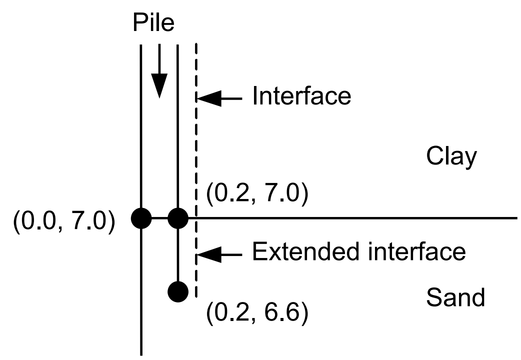



*Figure 202: Extended interface*

## 15.4.1 Define the pile

To define the concrete pile:

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .
2. Select the **Create line** feature in the side toolbar and draw a line from (0.2 6.6) to (0.2 18).
3. Assign a negative interface to the line to model the interaction of the pile with the surrounding soil.

   The pile is made of concrete, which is modelled by means of the linear elastic model considering non-porous behaviour. In the beginning, the pile is not present, so initially the clay properties are present in the area the represents the pile.

## 15.4.2 Define a load

In order to model the driving force, a distributed unit load is created on top of the pile. To create a dynamic load:

1. Define a distributed load by clicking on **Create load** > **Create line load** from the tool bar and then on (0 18.0) and (0.2 18.0).
2. The load components will be defined in the **Selection explorer**. Note that the static component of the load will not be used in this project. The program will neglect the static components of the load if it (static load) is not activated.
3. Expand the **Dynamic load** subtree and specify a unit load in the gravity direction.
4. Click the Multiplier_y drop down menu and click on the appearing plus button .
   The **Multipliers** window pops up and a new load multiplier is automatically added.

5. Define a **Harmonic** signal with an **Amplitude** of 5000, a **Phase** of 0° and a **Frequency** of 50 Hz and as shown in Figure 203 (on page 231) . During the pile driving phase, we will only consider half a cycle (0.01 s) of this signal.

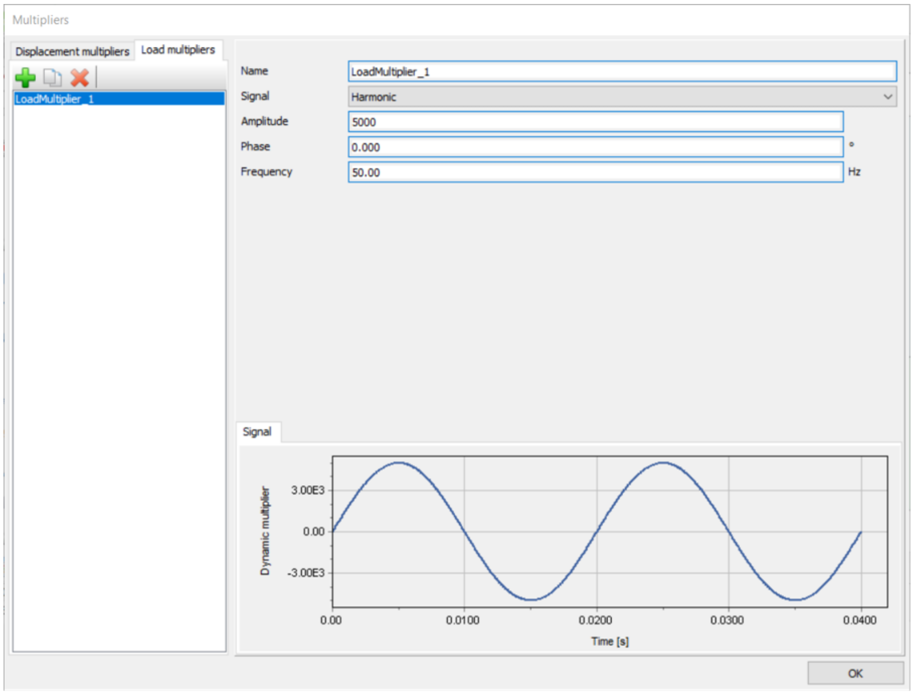



*Figure 203: Definition of an Harmonic multiplier*

**Note:**

Note that dynamic multipliers can be defined by right-clicking the **Dynamic multipliers** subtree under **Attributes library** in the **Model explorer**.

Note that dynamic multipliers are attributes and as such it is possible to define them in all the program's modes.

The final geometry model is shown in Figure 204 (on page 232):


*Figure 204: The geometry model*

## 15.5 Generate the mesh

1. Proceed to the **Mesh- mode** .
2. Click the **Generate mesh** button to generate the mesh. Use the default option for the **Element distribution** parameter (**Medium**).
3. Click the **View mesh** button to view the mesh shown in Figure 205 (on page 233).

   The resulting mesh is shown. Note that the mesh is automatically refined under the footing.

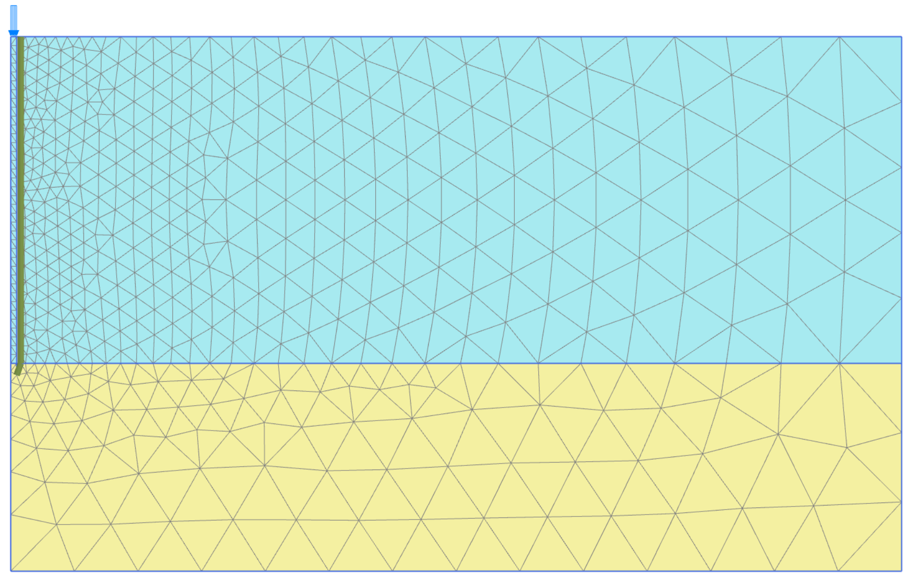


*Figure 205: The generated mesh*

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

## 15.6 Define and perform the calculation

The calculation consists of 3 phases. In the Initial phase, the initial stress conditions are generated. In the Phase 1 the pile is created. In the Phase 2 the pile is subjected to a single stroke, which is simulated by activating half a harmonic cycle of load. In the Phase 3 the load is kept zero and the dynamic response of the pile and soil is analysed in time. The last two phases involve dynamics calculations.

### 15.6.1 Initial phase

Initial effective stresses are generated by the **K0 procedure**, using the default values. Note that in the initial situation the pile does not exist and that the clay properties should be assigned to the corresponding cluster. The phreatic level is assumed to be at the ground surface. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

### 15.6.2 Phase 1: Pile activation

1. Click the **Add phase** button  to create a new phase.
2. In the **General** subtree in the **Phases** window, the **Plastic** option is selected as **Calculation type**.

3. The **Staged construction** option is by default selected as **Loading type**.
4. In the **Staged construction** mode assign the pile properties to the pile cluster.
5. Activate the interface in the **Clay** layer only. The model for the Phase 1 in the **Staged construction** mode is displayed in Figure 206 (on page 234).


*Figure 206: Configuration of Phase 1 in the **Staged construction** mode*

## 15.6.3 Phase 2: Pile driving

1. Click the **Add phase** button to create a new phase.
2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as **Calculation type**.
3. Set the **Dynamic time interval** to 0.01 s.
4. In the **Deformation control parameters** subtree select **Reset displacements to zero**.
5. In the **Staged construction mode** activate the dynamic component of the distributed load.

   The activated dynamic component of the load in **Selection explorer** is shown in Figure 207 (on page 235) .

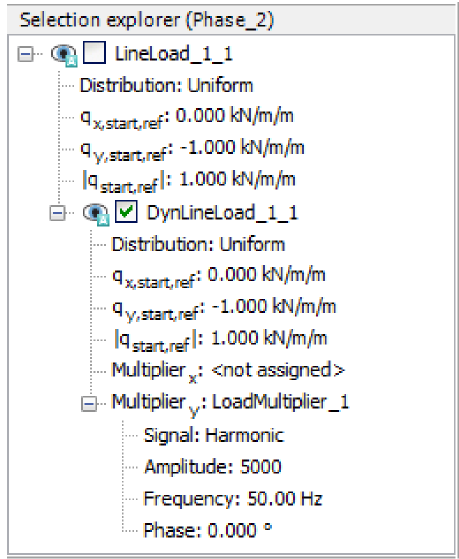



*Figure 207: The dynamic load component in the **Selection explorer***

6. Expand the subtree **Model explorer** > **Model conditions** > **Dynamics** is shown in Figure 208 (on page 235) .

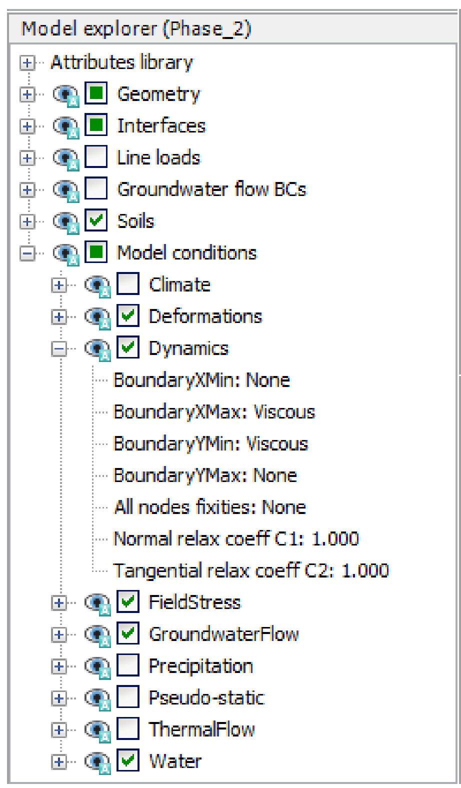



*Figure 208: Boundary conditions for **dynamics calculations***

7. Specify viscous boundaries at $x_{max}$ and $y_{min}$.

The result of this phase is half a harmonic cycle of the external load. At the end of this phase, the load is back to zero.

### 15.6.4 Phase 3: Fading

1. Click the **Add phase** button to create a new phase.
2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as **Calculation type**.
3. Set the **Dynamic time interval** to 0.19 s.
4. In the **Staged construction mode** de-activate the dynamic load.

### 15.6.5 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar and select a node at the top of the pile for load displacement curves.
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button.

## 15.7 Results

Figure 209 (on page 237) shows the settlement of the pile (top point) versus time. From this figure the following observations can be made:

- The maximum vertical settlement of the pile top due to this single stroke is almost 14 mm. However, the final settlement is about 9.5 mm.
- Most of the settlement occurs in phase 3 after the stroke has ended. This is due to the fact that the compression wave is still propagating downwards in the pile, causing additional settlements.
- Despite the absence of Rayleigh damping, the vibration of the pile is damped due to soil plasticity and the fact that wave energy is absorbed at the model boundaries.

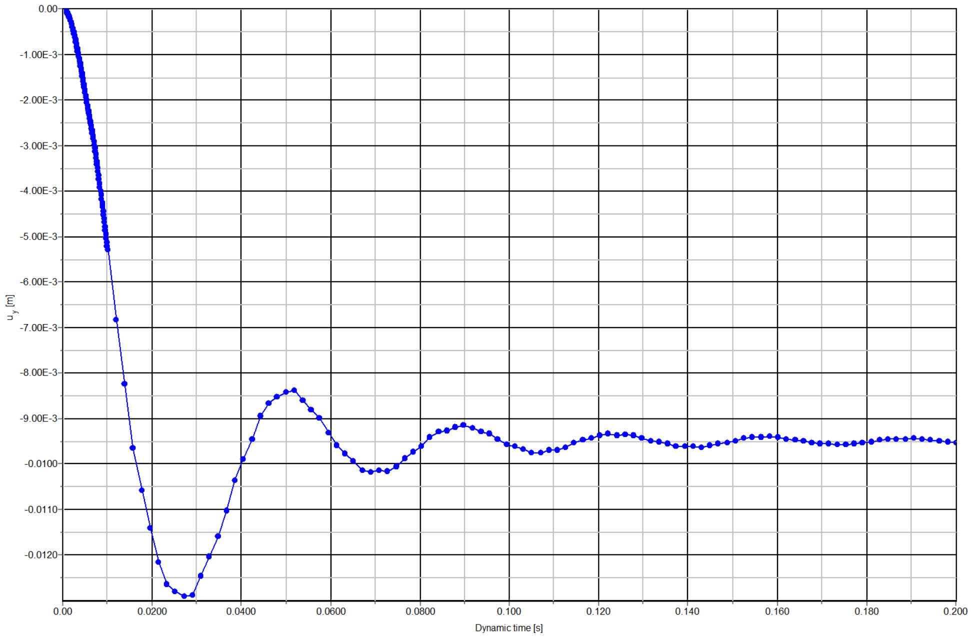



*Figure 209: Pile settlement vs. time*

When looking at the output of the second calculation phase (t = 0.01 s, i.e. just after the stroke), it can be seen that large excess pore pressures occur very locally around the pile tip. This reduces the shear strength of the soil and contributes to the penetration of the pile into the sand layer. The excess pore pressures remain also in the third phase since consolidation is not considered.

Figure 210 (on page 237) shows the shear stresses in the interface elements at t = 0.01 s. The plot shows that the maximum shear stress is reached all along the pile, which indicates that the soil is sliding along the pile.

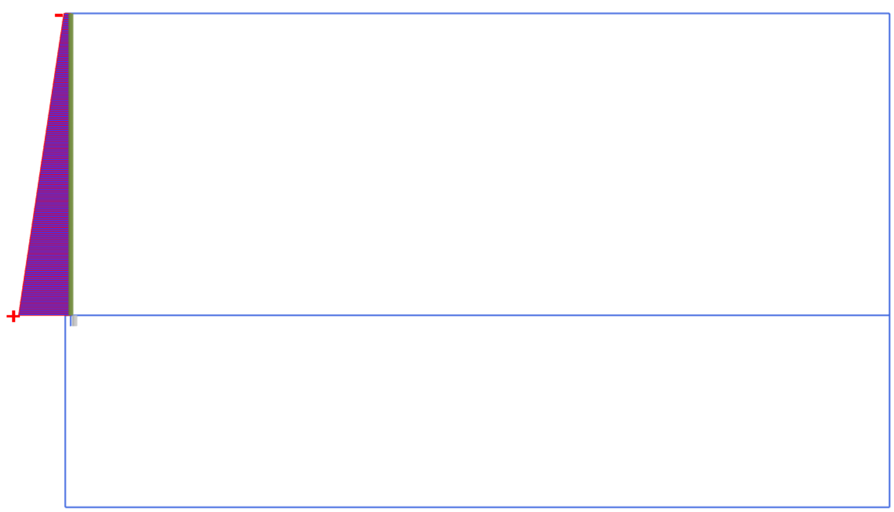


*Figure 210: Maximum shear stresses in the interface at t = 0.01 s*

When looking at the deformed mesh of the last calculation phase (t = 0.2 s), it can also be seen that the final settlement of the pile is about 9.5 mm. In order to see the whole dynamic process it is suggested to use the option **Create Animation** to view a 'movie' of the deformed mesh in time. You may notice that the first part of the animation is slower than the second part.

# 16 Free vibration and earthquake analysis of a building [ULT]

This example demonstrates the natural frequency of a five-storey building when subjected to free vibration and earthquake loading. The two calculations employ different dynamic boundary conditions:

- In the free vibration, the **Viscous** boundary conditions are considered. This option is suitable for problems where the dynamic source is inside the mesh.
- For the earthquake loading, the **Free-field** and **Compliant base** boundary conditions are considered. This option is preferred for earthquake analysis, where the dynamic input is applied at the bottom of the model.

**Objectives**

- Performing a **Dynamic** calculation
- Defining dynamic boundary conditions (free-field, compliant base and viscous)
- Defining earthquakes by means of dynamic multipliers
- Modelling free vibration of structures
- Modelling hysteretic behaviour by means of Hardening Soil model with small-strain stiffness
- Evaluating the natural frequency from Fourier spectrum

**Geometry**

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including basement. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep. A value of 5 $kN/m^2$ is taken as the weight of the floors and the walls. The building is constructed on a rather loose sand of 15 m depth underlain by a deep denser sand layer. In the model, 25 m of the deep sand layer will be considered as shown in Figure 211 (on page 240).

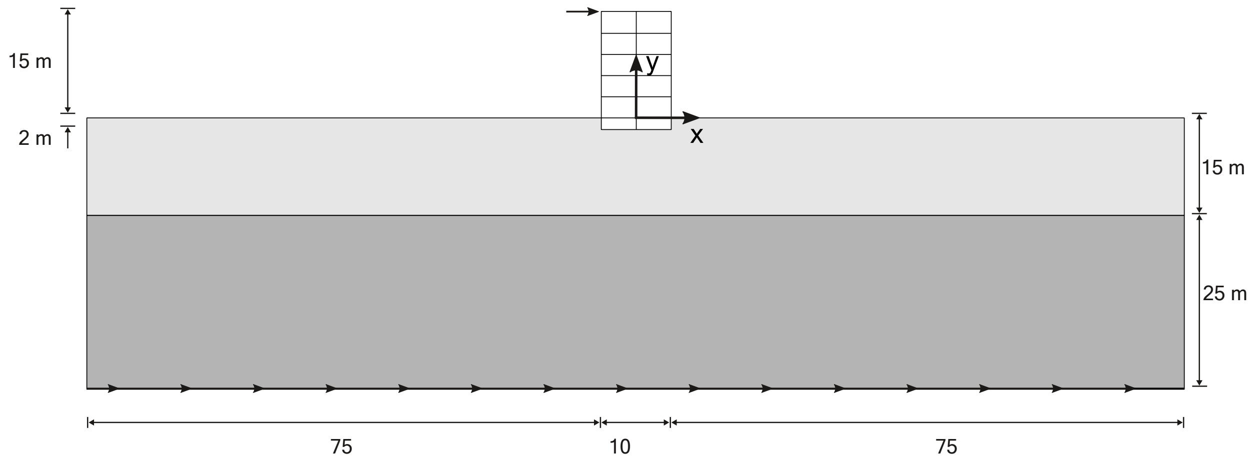



*Figure 211: Geometry of the project*

## 16.1 Create new project

To create the new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** 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-Noded**).
4. Keep the default units and constants and set the model **Contour** to $x_{min}$ = -80 m, $x_{max}$ = 80 m, $y_{min}$ = -40 m and $y_{max}$ = 15 m.

## 16.2 Define the soil stratigraphy

The subsoil is divided into a 15 m thick rather loose sand layer and a 25 m thick medium dense sand layer. The phreatic level is assumed to be at y = -15 m. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Add two soil layers extending from y = 0 to y = -15 and from y = -15 to y = -40.
3. Set the **Head** in the borehole at -15 m.

## 16.3 Create and assign material data sets

The upper layer consists of rather loose sand and the lower one is a medium dense sand. Both have Hardening Soil model with small-strain stiffness properties. The presence of the groundwater is neglected. The soil layers with Hardening Soil model with small-strain stiffness properties have inherent hysteretic damping.

- Open the **Material sets** window.
- Create data sets under **Soil and interfaces** set type according to the information given in Table 34 (on page 241).
- Assign the material datasets to the corresponding soil layers in the borehole.

**Table 34: Material properties**

| Parameter | Name | Upper sand layer | Lower sand layer | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | HS small | HS small | - |
| Drainage type | - | Drained | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 16 | 20 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | 20 | kN/m$^3$ |
| **Mechanical** | | | | |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $20 \cdot 10^3$ | $30 \cdot 10^3$ | kN/m$^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $26 \cdot 10^3$ | $36 \cdot 10^3$ | kN/m$^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $95 \cdot 10^3$ | $110 \cdot 10^3$ | kN/m$^2$ |
| Poisson's ratio | $\nu'_{ur}$ | 0.2 | 0.2 | - |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | 0.5 | - |
| Shear modulus at very small strains | $G_0{}^{ref}$ | $270 \cdot 10^3$ | $100 \cdot 10^3$ | kN/m$^2$ |
| Shear strain at which $G_s$ = 0.722 $G_0$ | $\gamma_{0.7}$ | $0.15 \cdot 10^{-3}$ | $0.1 \cdot 10^{-3}$ | - |
| Cohesion | $c'_{ref}$ | 10 | 5 | kN/m$^2$ |

| **Mechanical** | | | | |
|---|---|---|---|---|
| Friction angle | $\varphi'$ | 31 | 28 | ° |
| Dilatancy angle | $\psi$ | 0 | 0 | ° |

When subjected to cyclic shear loading, the Hardening Soil model with small-strain stiffness will show typical hysteretic behaviour. Starting from the small-strain shear stiffness, $G_0^{ref}$, the actual stiffness will decrease with increasing shear. The figures below display the Modulus reduction curves, i.e. the decay of the shear modulus with strain. Figure 212 (on page 242) shows the secant shear modulus and Figure 213 (on page 243) shows the tangent shear modulus.

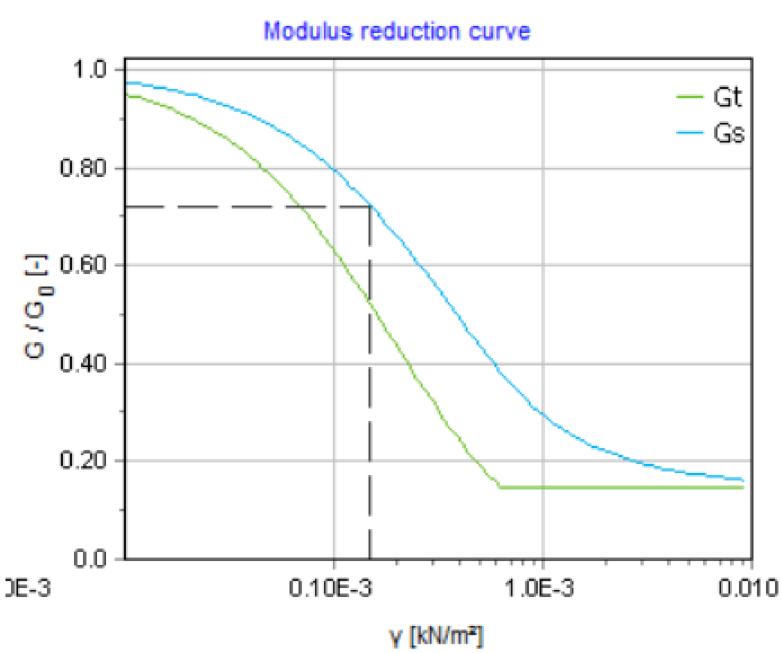



*Figure 212: Modulus reduction curves for the upper sand layer*

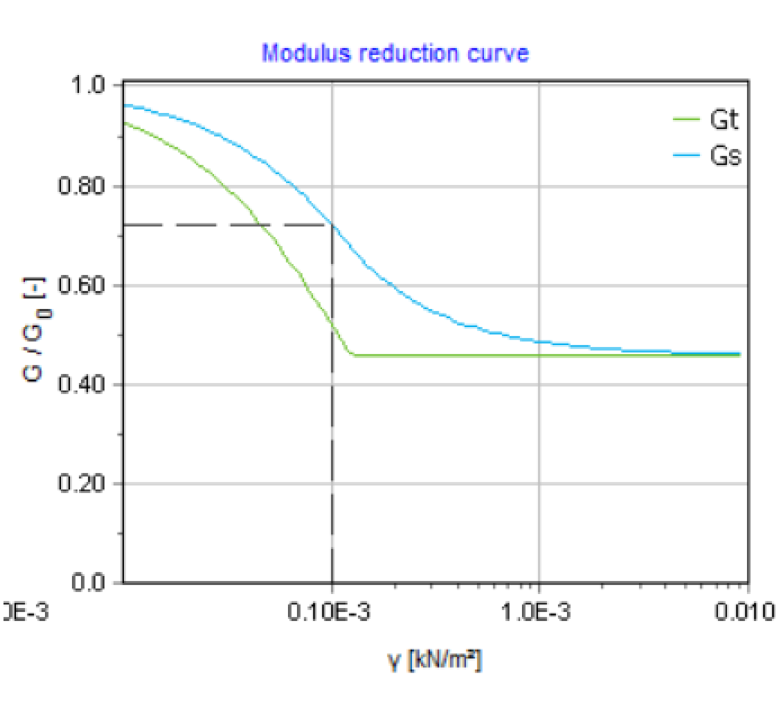



*Figure 213: Modulus reduction curve for the lower sand layer*

In the Hardening Soil model with small-strain stiffness, the tangent shear modulus is bound by a lower limit, $G_{ur}$.

$$G_{ur} = \frac{E_{ur}}{2\,(1 + \nu_{ur})}$$

The values of $G_{ur}^{ref}$ for the **Upper sand layer** and **Lower sand layer** and the ratio to $G_0^{ref}$ are shown in Table 35 (on page 243) . This ratio determines the maximum damping ratio that can be obtained.

**Table 35: $G_{ur}$ values and ratio to $G_0^{ref}$**

| Parameter | Upper sand layer | Lower sand layer | Unit |
|---|---|---|---|
| $G_{ur}$ | $39.10^3$ | $45.10^3$ | $kN/m^2$ |
| $G_0^{ref}/G_{ur}$ | 6.82 | 2.18 | - |

Figure 214 (on page 244) and Figure 215 (on page 244) show the damping ratio as a function of the shear strain for the material used in the model. A more detailed description about the modulus reduction curve to the damping curve can be found in the literature.[6]

[6] Brinkgreve, R.B.J., Kappert, M.H., Bonnier, P.G. (2007). Hysteretic damping in small-strain stiffness model. In Proc. 10th Int. Conf. on Comp. Methods and Advances in Geomechanics. Rhodes, Greece, 737-742.

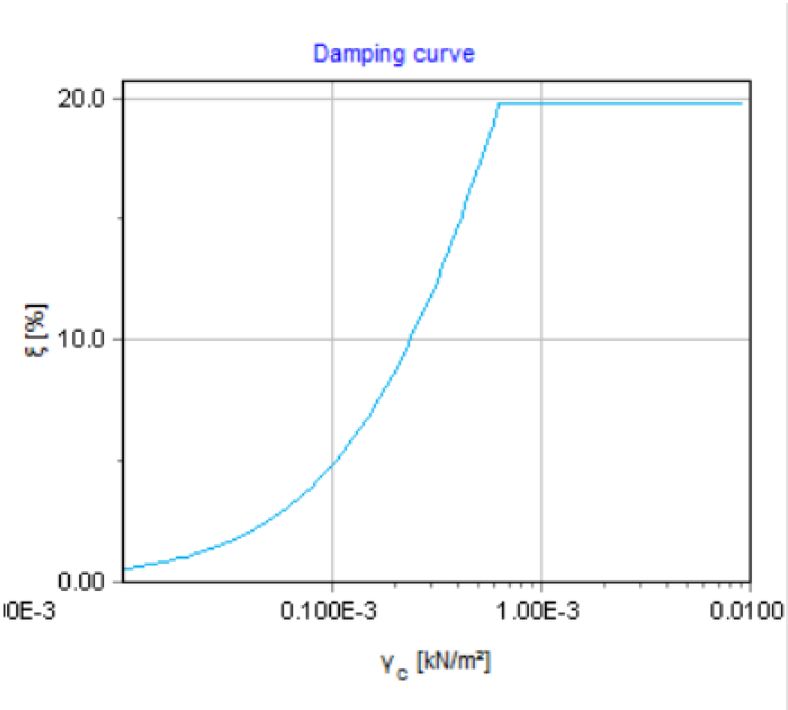



*Figure 214: Damping curve for the upper sand layer*

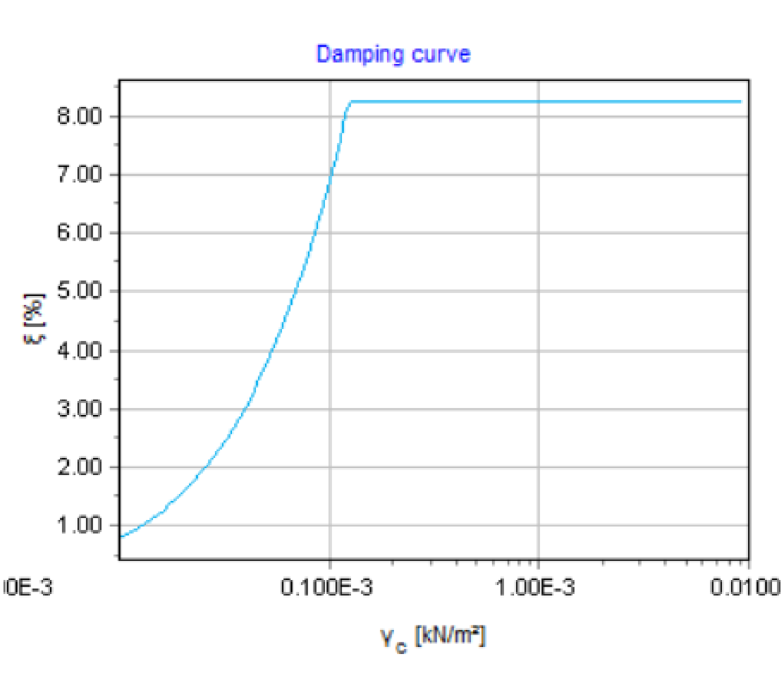



*Figure 215: Damping curve for the lower sand layer*

## 16.4 Define the structural elements

The structural elements of the model are defined in the **Structures mode**.

### 16.4.1 Define the building

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including the basement. A value of 5 kN/m$^2$ is taken as the weight of the floors and the walls. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep.

The plates, representing the walls and the floors in the building, are considered to be linear elastic. Note that two different material datasets are used, one for the basement and the other for the rest of the building. The physical damping in the building is simulated by means of Rayleigh damping. A description of Rayleigh damping parameters is given in the Reference Manual .

**Table 36: Material properties of the building (plate properties)**

| Parameter | Name | Building | Basement | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Material type | - | Elastic | Elastic | - |
| Weight | $w$ | 10 | 20 | kN/m/m |
| Input method | - | Direct | Direct | - |
| Rayleigh α | - | 0.2320 | 0.2320 | - |
| Rayleigh β | - | $8.0 \cdot 10^{-3}$ | $8.0 \cdot 10^{-3}$ | - |
| Prevent punching | - | No | No | - |
| **Mechanical** | | | | |
| Isotropic | - | Yes | Yes | - |
| Axial stiffness | $EA_1$ | $9 \cdot 10^6$ | $12 \cdot 10^6$ | kN/m |
| Bending stiffness | $EI$ | $67.5 \cdot 10^3$ | $160 \cdot 10^3$ | kNm$^2$/m |
| Poisson's ratio | $\nu$ | 0 | 0 | - |

**Table 37: Material properties of the node-to-node anchor**

| Parameter | Name | Column | Unit |
|---|---|---|---|
| Material type | Type | Elastic | - |
| Out-of-plane spacing | $L_{spacing}$ | 3 | m |
| Axial stiffness | $EA$ | $2.5 \cdot 10^6$ | kN |

1. Use plates to create the vertical walls of the building from (-5 0) to (-5 15) and from (5 0) to (5 15).
2. Again with plates, now define the vertical walls of the basement from (-5 -2) to (-5 0) and from (5 -2) to (5 0).
3. Define the basement and ground level of the building as plates from (-5 -2) to (5 -2) and from (-5 0) to (5 0).
4. Define the floors by copying the basement floor 5 times. To do so, select the basement floor and choose the **Array** button. Now specify that we want to copy it in y-direction, 6 copies (be aware: number of copies includes the original) and an intermediate distance of 3 m.
5. Define the material datasets for the structural elements in the building according to Table 36 (on page 245).
6. Assign the Basement material dataset to the vertical plates (2) and the lowest horizontal plate (all below ground level) in the model.
7. Assign the Building material dataset to the remaining plates in the model.
8. Use the Node-to-node anchor feature to define the column at the centre of the building connecting consecutive floors, hence (0 -2) to (0 0), (0 0) to (0 3), (0 3) to (0 6), (0 6) to (0 9), (0 9) to (0 12) and (0 12) to (0 15). Of course this can also be done by drawing one column and use the Array function to copy the others.
9. Define the properties of the anchor according to Table 37 (on page 246) and assign the material dataset to the anchors in the model.
10. Define an interface to model the interaction between soil and the building.

## 16.4.2 Define the loads

1. In order to model the driving force, a point load is created on top of the pile. To create a dynamic load:
   a. Create a point load at the top left corner of the building.
   b. Set $F_{x,ref}$ = 10 kN/m and $F_{y,ref}$ = 0 kN/m.
2. The earthquake is modelled by imposing a prescribed displacement at the bottom boundary. To define the prescribed displacement:
   a. Define a line displacement at the bottom of the model, from (-80 -40) to (80 -40).
   b. Set the x-component of the line displacement to **Prescribed** and assign a value of 1.0. The y-component of the line displacement is **Fixed**. The default distribution (Uniform) should be kept.
3. To define the dynamic multipliers for the line displacement:
   a. Expand the Dynamic line displacement.

b. Click the **Multiplier_x** drop down menu and click on the appearing plus button. The **Multipliers** window pops up and a new displacement multiplier is automatically added.
c. From the **Signal** drop-down menu select the **Table** option.
d. The file containing the earthquake data is available in the Bentley Communities: search for the tutorial Free vibration and earthquake analysis of a building. Download the earthquake signal file `225a.smc`.
e. In the **Multipliers** window click the **Open** button. In the appearing window change in the drop-down menu **Plain text files *.txt** for **Strong motion CD-ROM files** option and select the appearing saved .smc file.
f. Select the **Acceleration** option in the **Data type** drop-down menu.
g. Select the **Drift correction** options and click **OK** to finalize the definition of the multiplier.
The defined multiplier is displayed in Figure 216 (on page 247) :

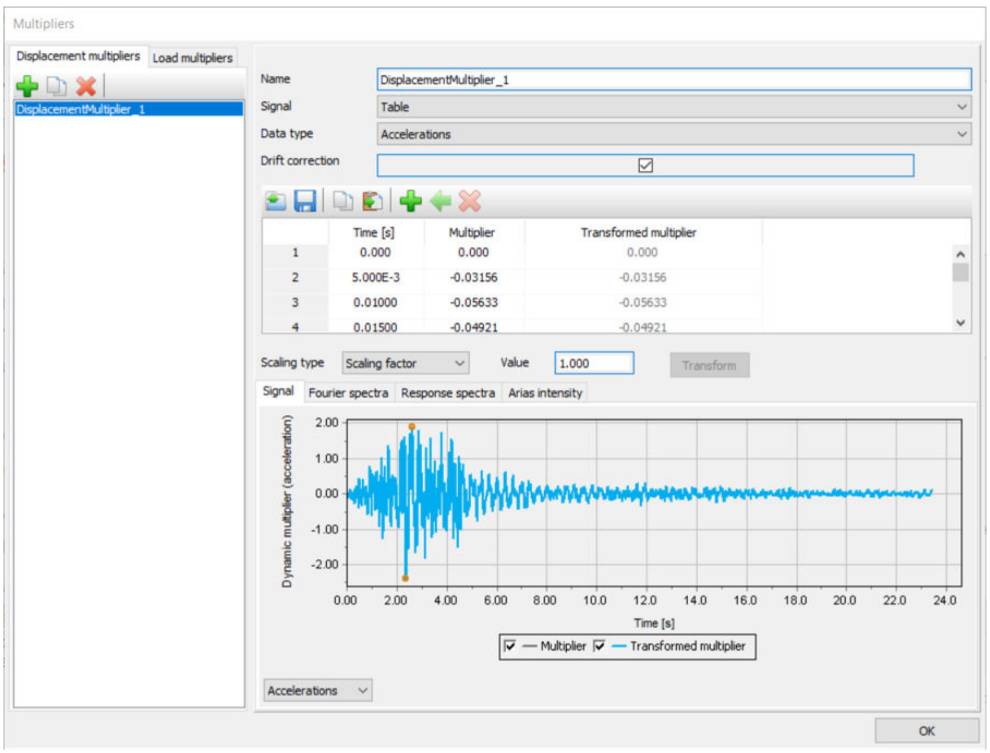



*Figure 216: Dynamic multipliers window*

## 16.4.3 Create interfaces on the boundary

Free-field and Compliant base boundaries require the creation of interface elements along the vertical and bottom boundaries of the model in the Structures mode. The interface elements must be added inside the model, else the Free-field and Compliant base boundary conditions are ignored. To define the interfaces:

1. Click the **Create interfaces on the boundary** button to automatically generate the interfaces at the boundary of the model.

The geometry of the model is shown in Figure 217 (on page 248) :

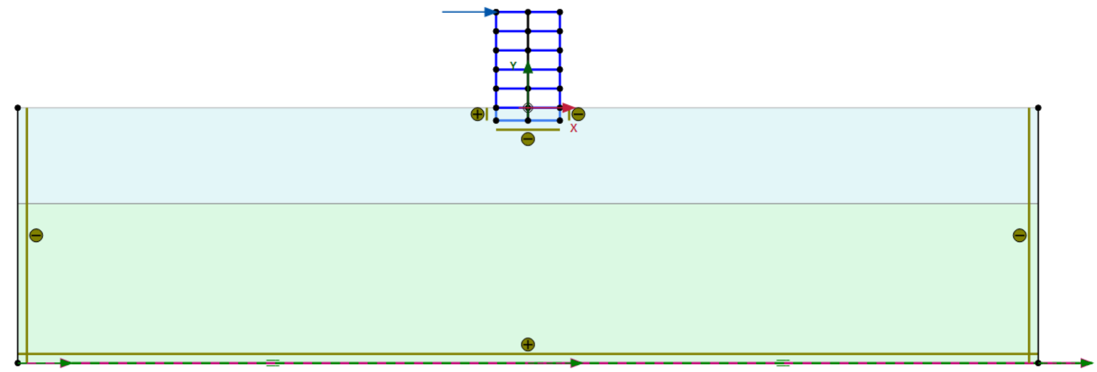


*Figure 217: Geometry of the model*

## 16.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Reset the mesh refinement on the boundaries by changing the **Coarseness factor** on the boundaries to 1.
3. Select both soil layers and set their **Coarseness factor** to 0.3.
4. Click the **Generate mesh** button to generate the mesh. Set the **Element distribution** to **Medium**.
5. Click the **View mesh** button to view the mesh.
   The resulting mesh is shown in Figure 218 (on page 248).:

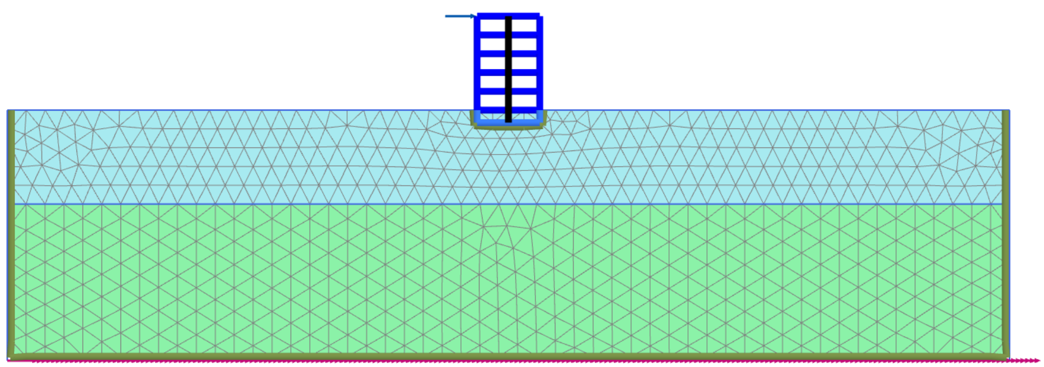


*Figure 218: The generated mesh*

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

## 16.6 Define and perform the calculation

The calculation process consists of the initial conditions phase, simulation of the construction of the building, loading, free vibration analysis and earthquake analysis.

## 16.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with definition of the calculation phases.
2. The initial phase has already been introduced. The default settings of the initial phase will be used in this tutorial.
3. In the **Staged construction** mode check that the building and load are inactive.
4. The model for the initial phase is shown in Figure 219 (on page 249).

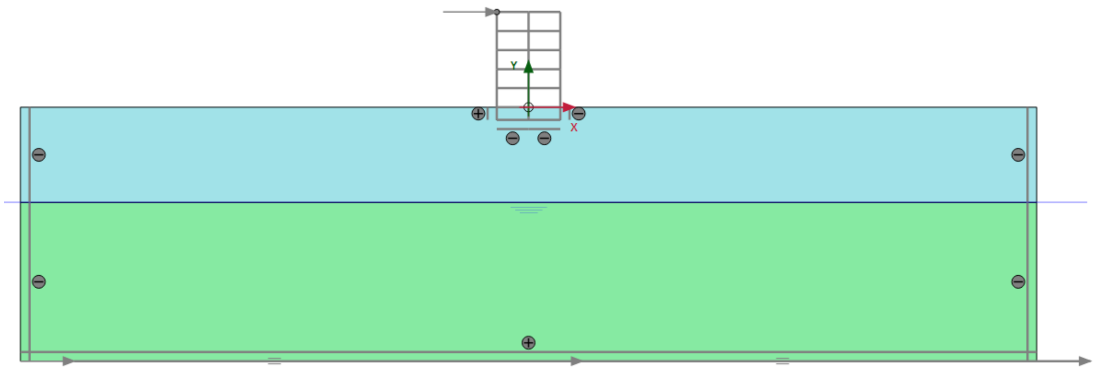


*Figure 219: Initial phase*

## 16.6.2 Phase 1: Building

1. Click the **Add phase** button to create a new phase. The default settings of the added phase will be used for this calculation phase.
2. In the **Staged construction** mode construct the building (activate all the plates, the anchors and only the interfaces of the basement) and deactivate the basement volume.
3. The model for phase 1 is shown in Figure 220 (on page 249).

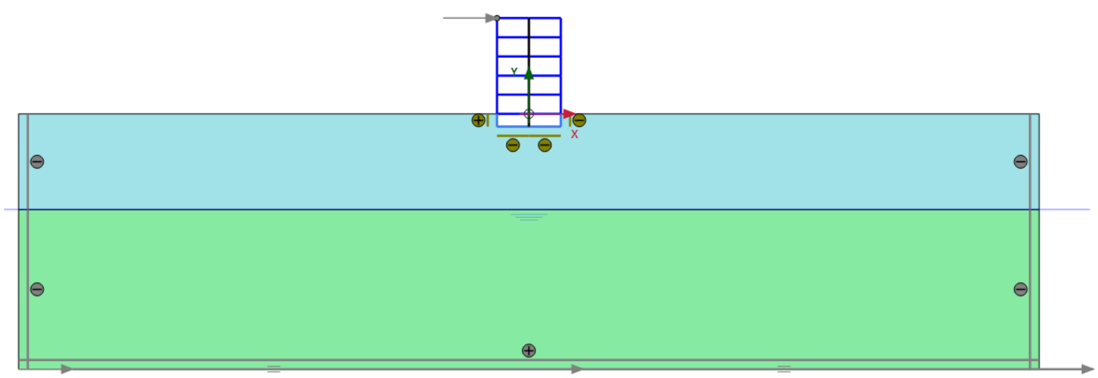


*Figure 220: Construction of the building*

## 16.6.3 Phase 2: Excitation

1. Click the **Add phase** button to create a new phase (Phase_2).
2. In the **Phases** window select the **Reset displacement to zero** in the **Deformation control parameters** subtree. The default values of the remaining parameters will be used in this calculation phase.
3. In the **Staged construction mode** activate the load. The value of the load is already defined in the **Structures mode** .

## 16.6.4 Phase 3: Free vibration

1. Click the **Add phase** button to create a new phase (Phase_3).
2. In the **Phases** window select the **Dynamic** option as **Calculation type**.
3. Set the **Dynamic time interval** parameter to 5 sec.
4. In the **Staged construction** mode release (deactivate) the point load.
5. In the **Model explorer** expand the **Model conditions** subtree.
6. Expand the **Dynamics** subtree. Check the boundary conditions **BoundaryXMin**, **BoundaryXMax** and **BoundaryYMin** are viscous which is shown in Figure 221 (on page 251).

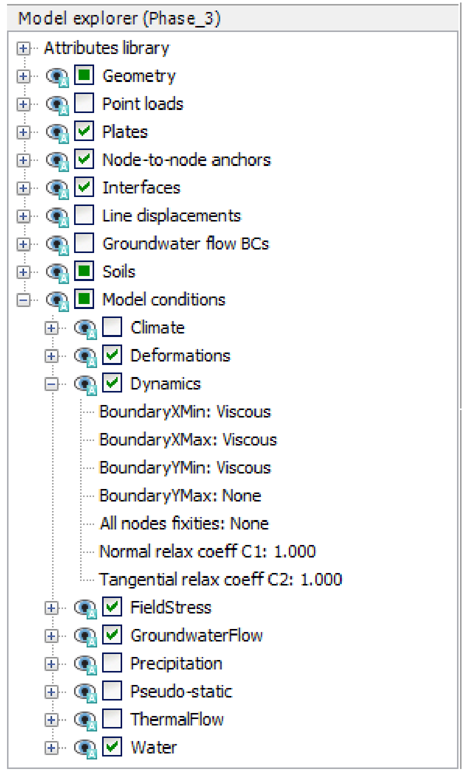



*Figure 221: Boundary conditions for **dynamics calculations***

**Note:**

For a better visualisation of the results, animations of the free vibration and earthquake can be created. If animations are to be created, it is advised to increase the number of the saved steps by assigning a proper value to the **Max steps saved** parameter in the **Parameters** tabsheet of the **Phases** window.

## 16.6.5 Phase 4: Earthquake

1. Click the **Add phase** button to create a new phase (Phase_4).
2. In the **Phases** window set the **Start from phase** option to Phase 1 (construction of building).
3. Select the **Dynamic** option as **Calculation type.**
4. Set the **Dynamic time interval** parameter to 20 sec.
5. Select the **Reset displacement to zero** in the **Deformation control parameters** subtree.
6. In the **Numerical control parameters** unselect the option **Use default iter parameters** and set **Max steps** to 1000 in order to get a more detailed time-acceleration curve.
7. In the **Model explorer** expand the **Model conditions** subtree.
8. Expand the **Dynamics** subtree. Set the **BoundaryXMin** and **BoundaryXMax** to **Free-field.** Set the **BoundaryYMin** to **Compliant base** as shown in Figure 222 (on page 252).

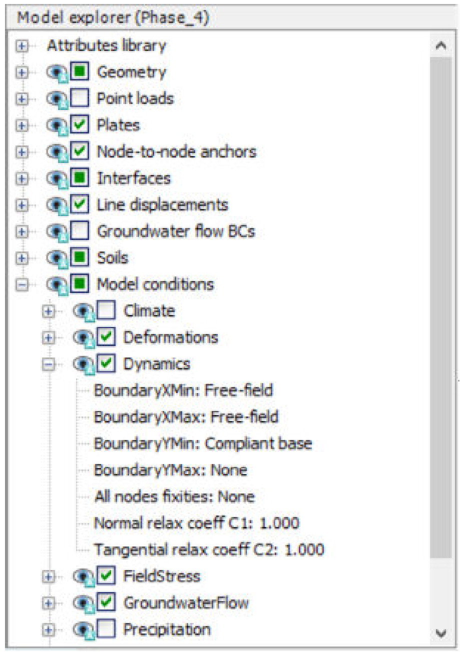



*Figure 222: Boundary conditions for **dynamics calculations** (Phase_4)*

9. Interface elements do not need to be active to enable the use of **Free-field** or **Compliant base** boundaries.
10. In the **Model explorer** activate the **Line displacements** and its dynamic component. Make sure that the value of $u_{x,start,ref}$ is set to 0.5 m. Considering that the boundary condition at the base of the model will be defined using a **Compliant base**, the input signal has to be taken as half of the bedrock (within) motion.

### 16.6.6 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar and select a point at the top of the building, close to or at (0 15).
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button.

## 16.7 Results

Figure 223 (on page 253) shows the deformed structure at the end of the Phase 2 (application of horizontal load).

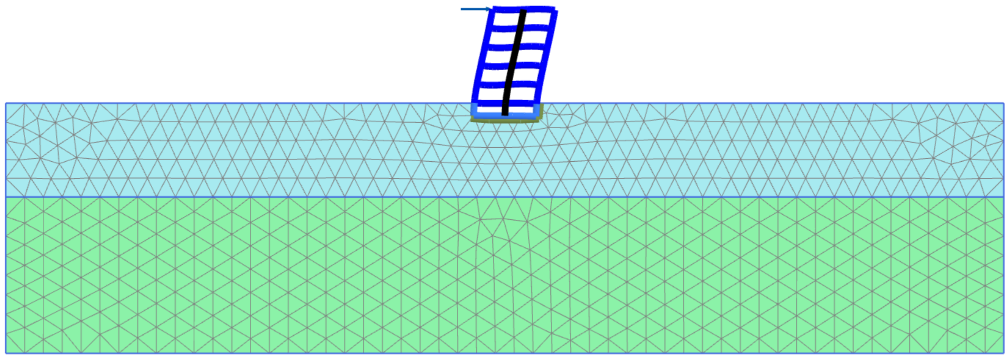


*Figure 223: Deformed structure in phase 2*

Figure 224 (on page 253) shows the time history of displacements of the selected points A (0 15) for the free vibration phase. It may be seen from the figure that the vibration slowly decays with time due to damping in the soil and in the building.

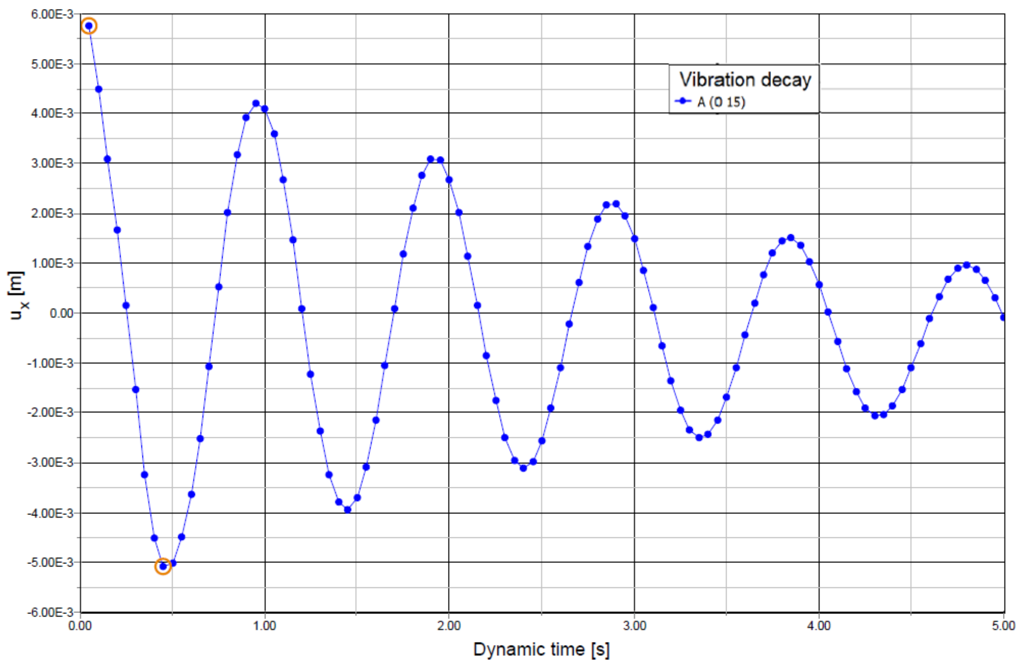



*Figure 224: Time history of displacements at selected points*

In the **Fourier** tabsheet of the **Curve generation** window select the **Power (spectrum)** > **Total displacements** > **Ux** and click **OK** to create the plot. The plot is shown in Figure 225 (on page 254). From this figure it can be evaluated that the dominant building frequency is around 1 Hz.

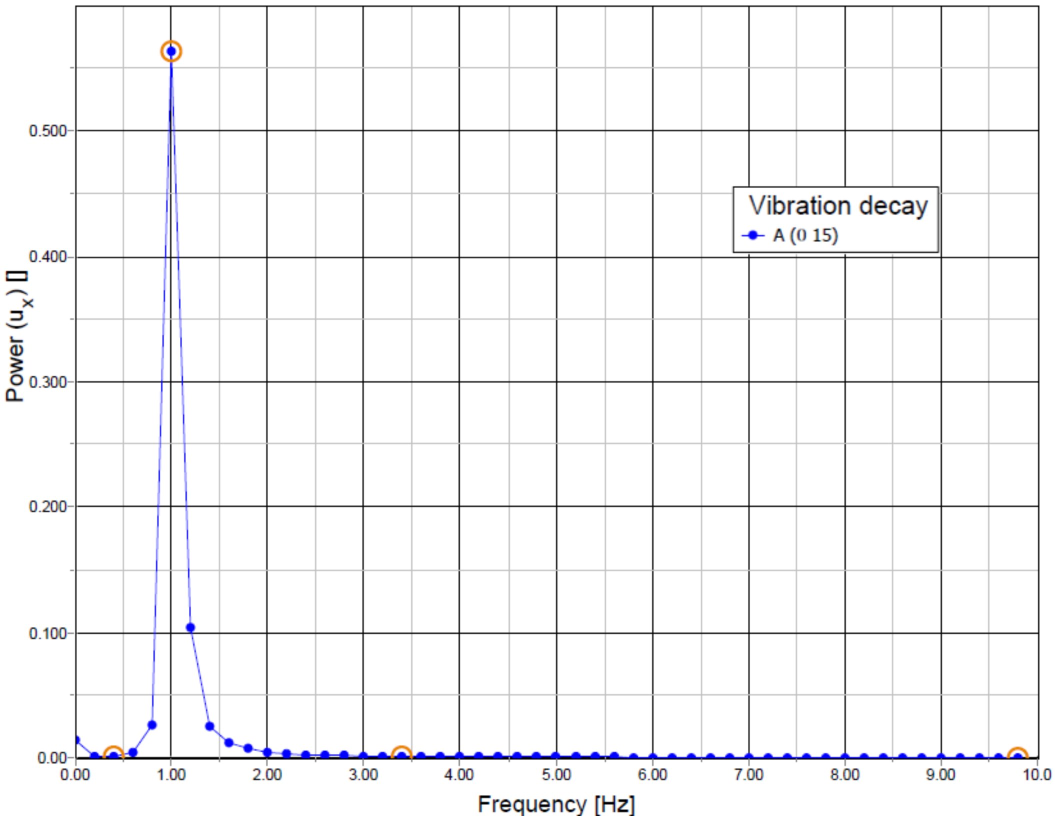



*Figure 225: Frequency representation (spectrum)*

Figure 226 (on page 255) shows the time history of the lateral acceleration of the selected point at (0 15) for the earthquake phase (dynamic analysis). For a better visualisation of the results animations of the free vibration and earthquake can be created.

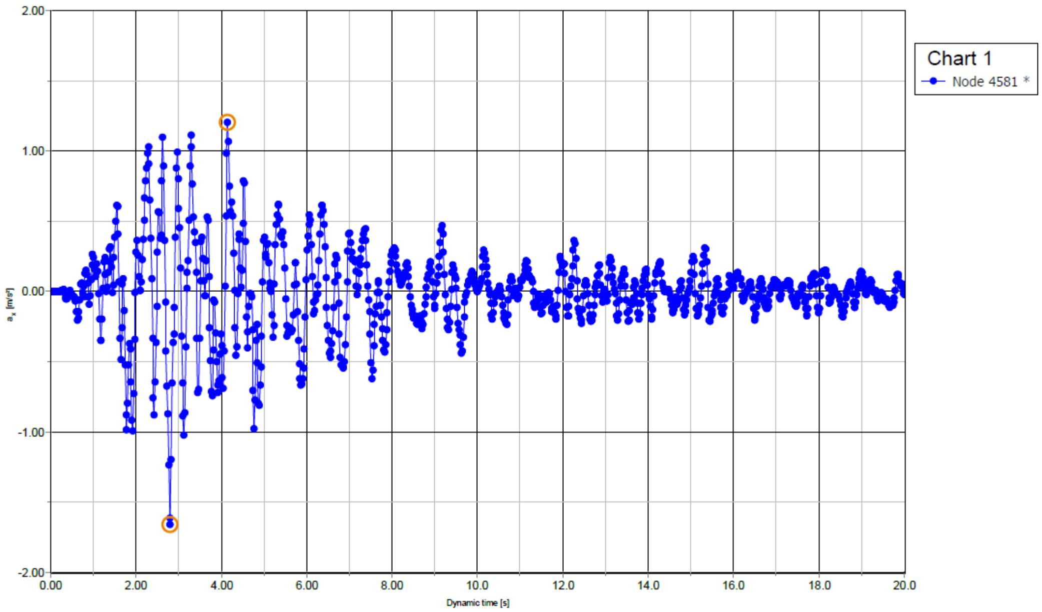



*Figure 226: Variation of acceleration in dynamic time*

17

# Thermal expansion of a navigable lock [ULT]

A navigable lock is temporarily 'empty' due to maintenance. After some time there is significant increase of the air temperature, which causes thermal expansion of the inner side of the lock, while the soil-side of the concrete block remains relatively cold. This leads to backward bending of the wall and, consequently, to increased lateral stress in the soil behind the wall and increased bending moments in the wall itself.

**Objectives**

This example demonstrates the use of the **Thermal** module to analyse this kind of situations.

- Defining a thermal temperature function
- Use of thermal expansion
- Performing a fully coupled analysis for THM calculation

The geometry of the project is shown in Figure 227 (on page 256).

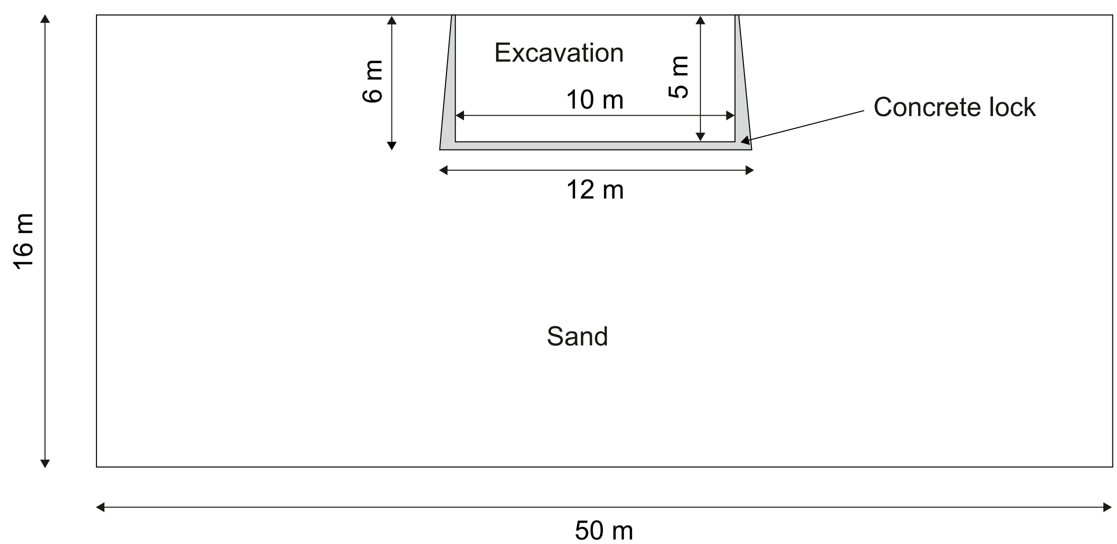



*Figure 227: Geometry of the project*

## 17.1 Create new project

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet, the default options for **Model** and **Elements** are used for this project. Also the default options for the units are used in this tutorial.
4. Set the model **Contour** to $x_{min}$ = 0 m, $x_{max}$ = 25 m, $y_{min}$ = -16 m and $y_{max}$ = 0 m.
5. Click **OK** to close the **Project properties** window.

## 17.2 Define the soil stratigraphy

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Create a single soil layer with top level at 0.0 m and bottom level at -16 m. Set the head at -4 m.

## 17.3 Create and assign material data sets

Two data sets need to be created; one for the sand layer and one for the concrete block.

1. Open the **Material sets** window.
2. Define a data set for the **Sand** layer with the parameters given in Table 38 (on page 257), for the **General**, **Mechanical**, **Groundwater**, **Thermal**, **Interfaces** and **Initial** tabsheets.
3. Create another dataset for **Concrete** according to Table 38 (on page 257).
4. Assign the material dataset **Sand** to the borehole soil layer.

**Table 38: Material properties**

| Parameter | Name | Sand | Concrete | Unit |
|---|---|---|---|---|
| **General** | | | | |
| Soil model | - | HS small | Linear elastic | - |
| Drainage type | - | Drained | Non-porous | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 20 | 24 | kN/m$^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 20 | - | kN/m$^3$ |

| **Mechanical** | | | | |
|---|---|---|---|---|
| Young's modulus | $E_{ref}$ | - | $25 \cdot 10^6$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | - | 0.15 | - |
| Secant stiffness in standard drained triaxial test | $E_{50}{}^{ref}$ | $40 \cdot 10^3$ | - | $kN/m^2$ |
| Tangent stiffness for primary oedometer loading | $E_{oed}{}^{ref}$ | $40 \cdot 10^3$ | - | $kN/m^2$ |
| Unloading / reloading stiffness | $E_{ur}{}^{ref}$ | $1.2 \cdot 10^5$ | - | $kN/m^2$ |
| Power for stress-level dependency of stiffness | $m$ | 0.5 | - | - |
| Shear modulus at very small strains | $G_0{}^{ref}$ | $80 \cdot 10^3$ | - | $kN/m^2$ |
| Shear strain at which $G_s = 0.722\ G_0$ | $\gamma_{0.7}$ | $0.1 \cdot 10^{-3}$ | - | - |
| Cohesion | $c'_{ref}$ | 2 | - | $kN/m^2$ |
| Friction angle | $\varphi'$ | 32 | - | ° |
| Dilatancy angle | $\psi$ | 2 | - | ° |

| **Groundwater** | | | | |
|---|---|---|---|---|
| Data set | - | USDA | - | - |
| Model | - | Van Genuchten | - | - |
| Soil - Type | - | Sand | - | - |
| Flow parameters - Use defaults | - | From data set | - | - |

| **Thermal** | | | | |
|---|---|---|---|---|
| Specific heat capacity | $c_s$ | 860 | 900 | kJ/t/K |
| Thermal conductivity | $\lambda_s$ | $4 \cdot 10^{-3}$ | $1 \cdot 10^{-3}$ | kW/m/K |
| Soil density | $\rho_s$ | 2.6 | 2.5 | $t/m^3$ |
| Thermal expansion type | - | Isotropic | Isotropic | - |
| Volumetric Thermal expansion | $\alpha_{sv}$ | $1.5.10^{-6}$ | $0.03.10^{-3}$ | 1/K |

| | | | | |
|---|---|---|---|---|
| **Interfaces** | | | | |
| Strength determination | - | Rigid | Manual | - |
| Interface reduction factor | $R_{inter}$ | 1.0 | 0.67 | - |
| **Initial** | | | | |
| $K_0$ determination | - | Automatic | Automatic | - |

## 17.4 Define the structural elements

The lock will be modelled as a concrete block during the staged construction.

1. Proceed to **Structures mode** .
2. Click the **Create soil polygon** button in the side toolbar and select the **Create soil polygon** option.
3. Define the lock in the drawing area by clicking on (0 -5), (5 -5), (5 0), (5.5 0), (6 -6), (0 -6) and (0 -5).

**Note:**

The **Snapping options** can be selected, and the **Spacing** can be set to 0.5 to easily create the polygon.

The **Concrete** material will be assigned later in the **Staged construction**.

4. Click the **Create line** button in the side toolbar.
5. Select the **Create thermal flow bc** option in the expanded menu.
6. Create thermal boundaries at vertical boundaries and the bottom boundary ($X_{min}$, $X_{max}$ and $Y_{min}$).
7. The vertical boundaries have the default option of **Closed** for the **Behaviour**.
8. Select the bottom boundary, in the **Selection explorer** set the **Behaviour** to **Temperature**.
9. Set the reference temperature, $T_{ref}$ to 283.4 K which is shown in Figure 228 (on page 259).

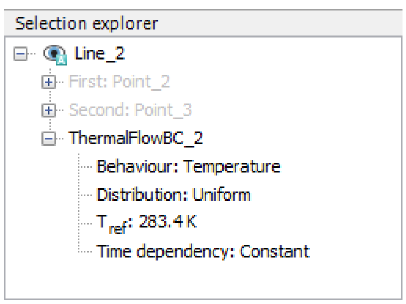



*Figure 228: Thermal boundary condition in the **Selection explorer***

The geometry of the model is now complete as shown in Figure 229 (on page 260) .

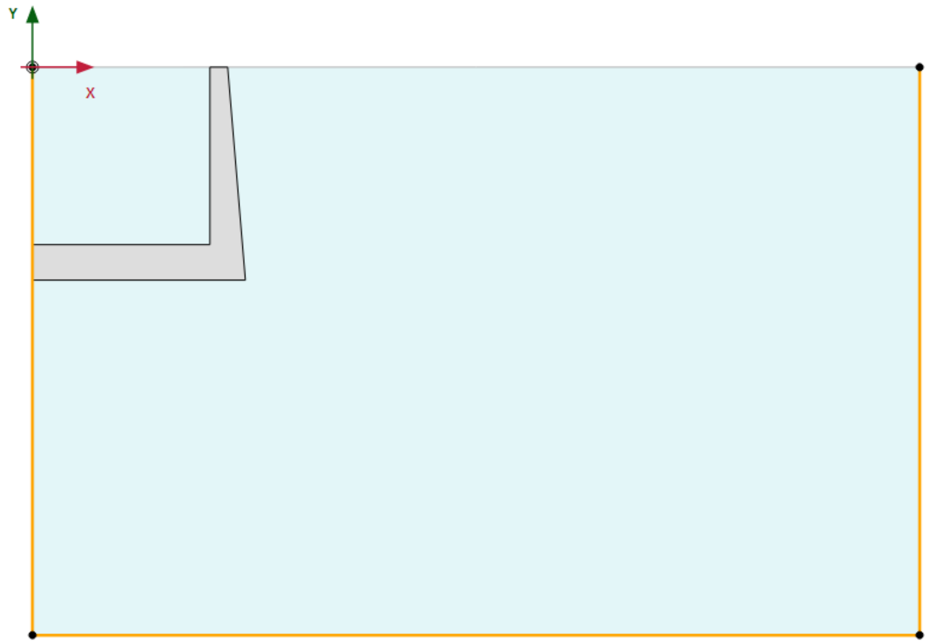



*Figure 229: Geometry of the model*

## 17.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Select the polygon representing the concrete block, and in the **Selection explorer** set the **Coarseness factor** to 0.25.
3. Click the **Generate mesh** button to generate the mesh. The default element distribution of **Medium** is used for this example.
4. Click the **View mesh** button to view the mesh.
   The resulting mesh is shown is shown in Figure 230 (on page 261):

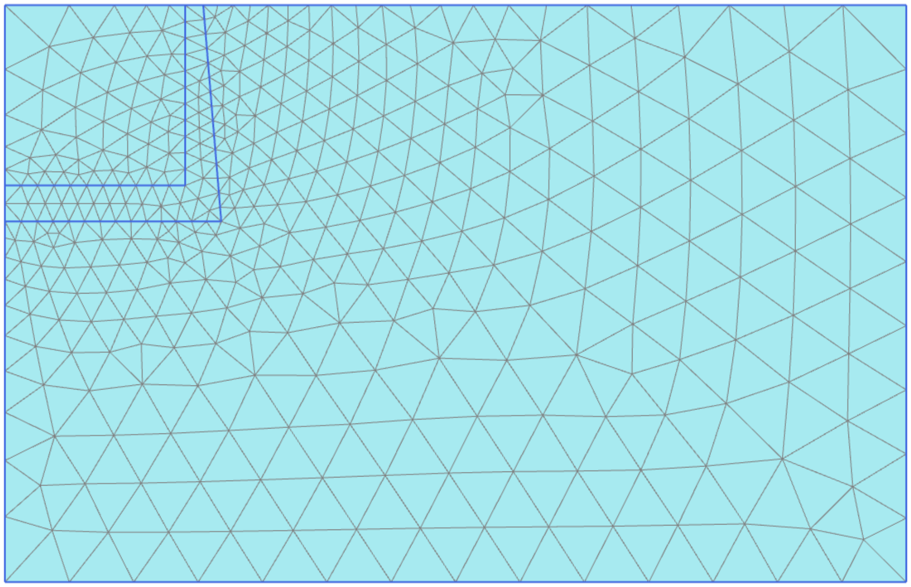


*Figure 230: The generated mesh*

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

## 17.6 Define and perform the calculation

The calculations for this tutorial is carried out in three phases. The concrete lock is activated in a plastic calculation, after which the temperature increase is defined as a fully coupled flow deformation analysis.

### 17.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with the definition of the calculation phases.
2. Double click on **Initial phase** in the **Phases explorer**.
3. The default options for **Calculation type** and **Pore pressure calculation type** are used in this example.
4. Select **Earth gradient** for the **Thermal calculation type** option and close the **Phases** window.
5. In the **Staged construction** activate the **ThermalFlow** under the **Model conditions** subtree and set the value for $T_{ref}$ to 283 K. The default values for $h_{ref}$ and **Earth gradient** are valid. The Thermal flow parameters are shown in Figure 231 (on page 262) and the model for initial phase is shown in Figure 232 (on page 262).

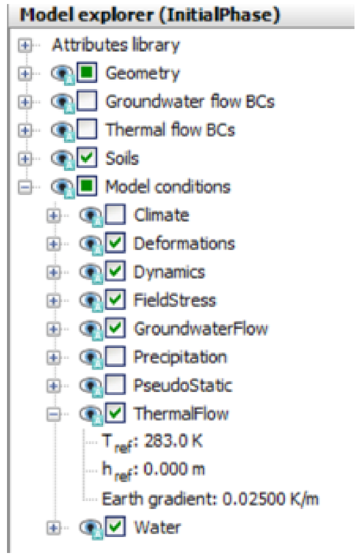



*Figure 231: Thermal flow in the Model explorer*

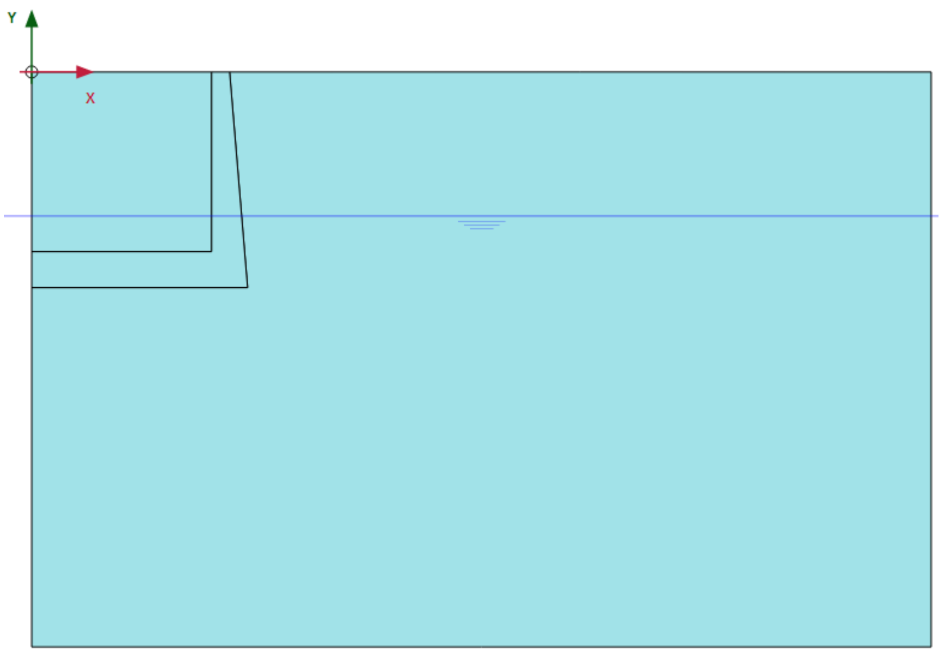



*Figure 232: Initial phase*

## 17.6.2 Phase 1: Construction

1. Click the **Add phase** button to create a new phase (Phase_1).

2. Double click on Phase_1 in the **Phases explorer**.
3. In the **Phases** window, enter an appropriate name for the phase ID and select **Steady state groundwater flow** as **Pore pressure calculation type**.
4. Set the **Steady state thermal flow** for the **Thermal calculation type**.
5. Make sure that the **Reset displacements to zero** and **Ignore suction** options are selected.
6. In the **Staged construction** mode, assign the **Concrete** dataset to the created polygon which represents the navigable lock as shown in Figure 233 (on page 263).

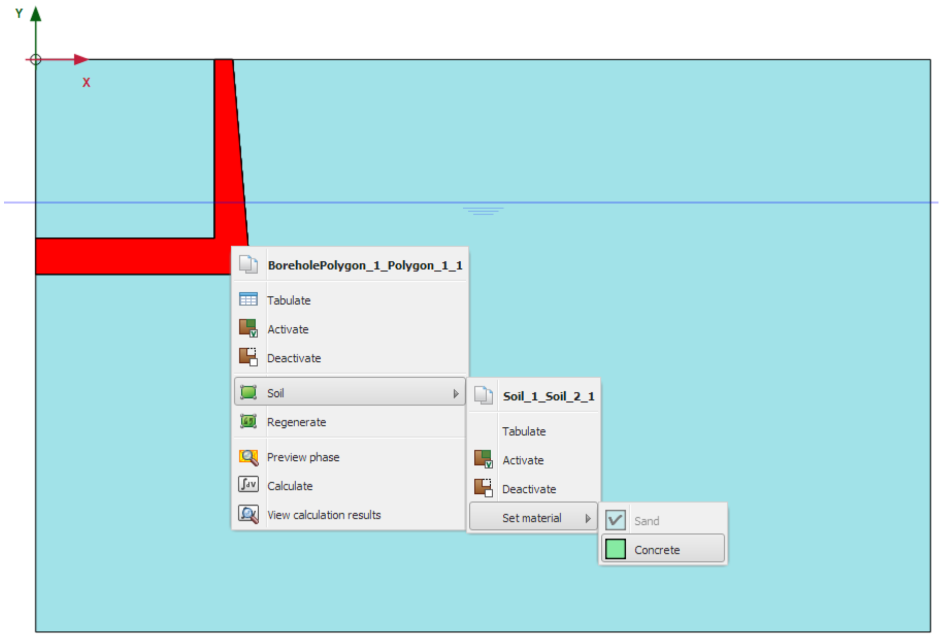



*Figure 233: Concrete set as material for polygon*

7. Right click the soil cluster which is cut-off by the polygon and select the option **Deactivate** from the appearing menu.
8. In the **Selection explorer**, set the **WaterConditions** of this cluster to **Dry**.
9. In the **Model explorer**, activate all the **Thermal flow boundary conditions**.
10. In the **Model explorer**, activate the **Model conditions** > **Climate** condition.
11. Set the **Air temperature** to 283 K and the **Surface transfer** to 1 kW/m$^2$/K as shown in Figure 234 (on page 264).

    This will define the thermal conditions at the ground surface and the inside of the lock.

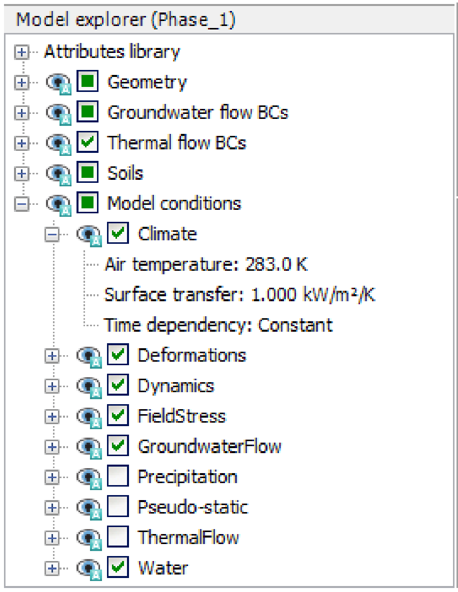



*Figure 234: Model conditions for Phase_1*

**12.** Deactivate the **ThermalFlow** option. This is because the thermal flow boundary conditions, including climate condition, are used in a steady state thermal flow calculation, instead of the earth gradient option.

Figure 235 (on page 264) shows the model at the end of **Phase_1**.

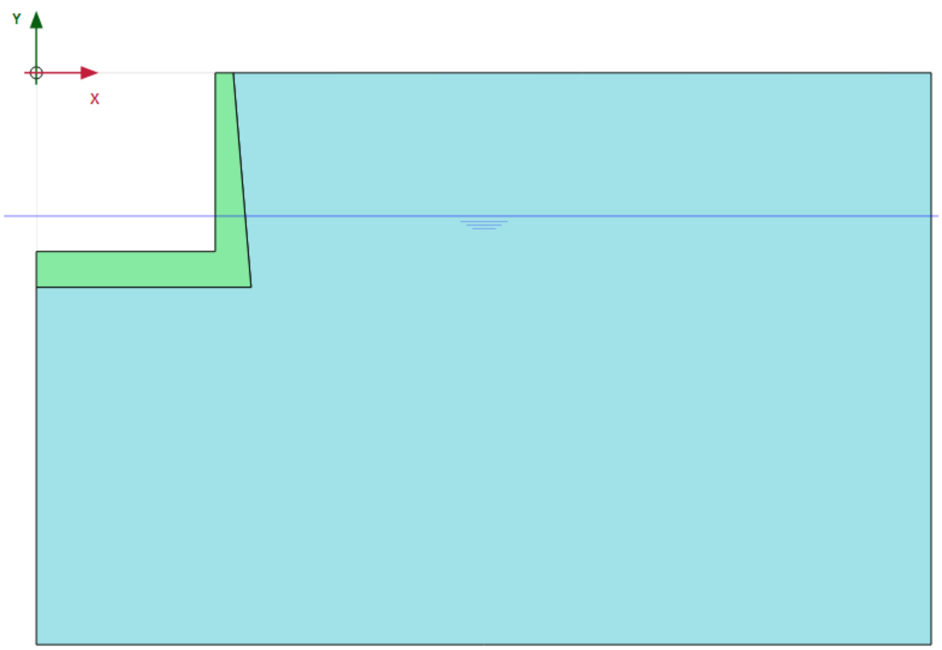



*Figure 235: The model at the end of Phase_1*

### 17.6.3 Phase 2: Heating

1. Click the **Add phase** button to create a new phase (Phase_2).
2. Double click on Phase_2 in the **Phases explorer**.
3. Set the **Calculation type** to **Fully coupled flow deformation**.
4. The **Thermal calculation type** is set to **Use temperatures from previous phase**. This is to indicate that temperature needs to be considered and that the initial temperature is taken from the previous phase.
5. The **Time interval** is set to 10 days.
6. Make sure that the **Reset displacements to zero** and **Reset small strain** options are selected in the **Deformation control parameters** subtree. The **Ignore suction** option is unchecked by default.
7. A temperature function is defined for the **Time dependency** in **Climate** which is used for this phase. Follow these steps to create a temperature function.
   a. Right-click the **Thermal functions** option in the **Attributes library** in the **Model explorer** and select **Edit** option in the appearing menu.
   The **Thermal functions** window is displayed.
   b. In the **Temperature functions** tabsheet add a new function by clicking on the corresponding button.
   The new function is highlighted in the list and options to define the function are displayed.
   c. The default option of **Harmonic** is used for this signal.
   d. Assign a value of 15 for the **Amplitude** and 40 days for the **Period**.

   A graph is displayed in Figure 236 (on page 266) showing the defined function. Since the time interval of the phase is 10 days, only a quarter of a temperature cycle is considered in this phase, which means that after 10 days the temperature has increased by 15 K.

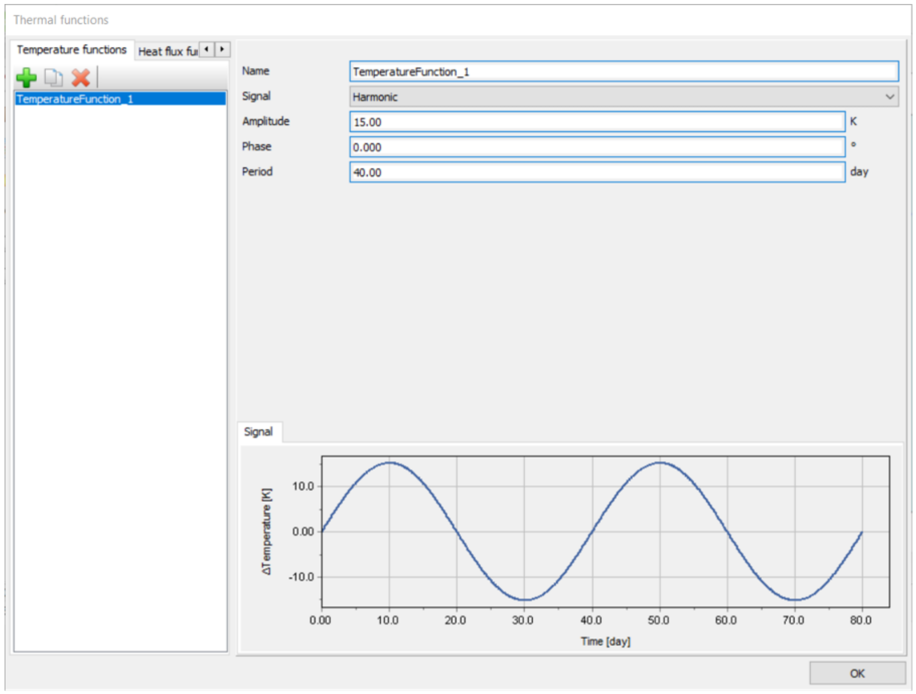



*Figure 236: The temperature function*

   **e.** Click **OK** to close the **Thermal functions** window.

8. Expand the subtree **Model conditions** in the **Model explorer** shown in Figure 237 (on page 266).
9. In the **Climate** option, set the **Time dependency** to **Time dependent** and assign the temperature function which was created.

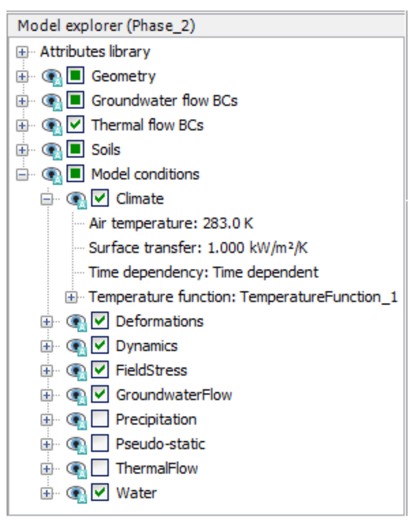



*Figure 237: Model conditions for Phase_2*

The calculation definition is now complete.

### 17.6.4 Execute the calculation

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of curves.

1. Click the **Select points for curves** button in the side toolbar and select some characteristic points for curves (for example at the top of the excavation, (5.0, 0.0)).
2. Click the **Calculate** button to calculate the project, a warning regarding different stress type used in the **Fully coupled flow deformation** analysis will appear. This warning appears because the **Fully coupled flow deformation** analysis always calculates with suction while the other calculation types by default do not calculate suction, and mixing phases with and without suction may lead to unexpected results. However, since in this tutorial we are dealing with sand the influence of suction will be very small and thus the warning can be ignored.
3. After the calculation has finished, save the project by clicking the **Save** button.

## 17.7 Results

In the **Phases explorer**, select the **Initial phase** and click the **View calculation results** button on the toolbar. In the Output program, select the menu **Stresses** > **Heat flow** > **Temperature**.

Figure 238 (on page 268) shows the initial temperature distribution, which is obtained from the reference temperature at the ground surface and the earth gradient. This gives a temperature of 283.0 K at the ground surface and 283.4 at the bottom of the model.

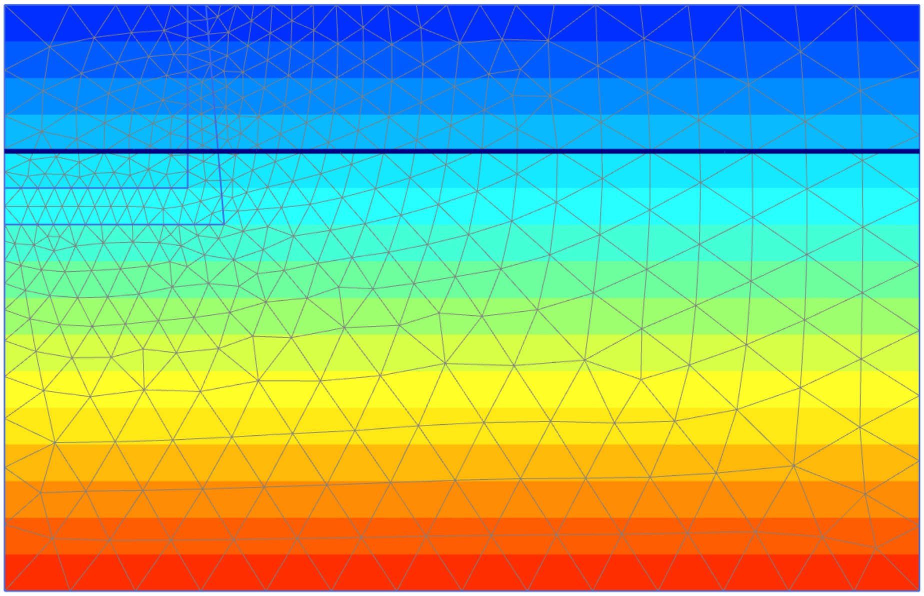


*Figure 238: Initial temperature distribution*

Figure 239 (on page 268) shows the temperature distribution obtained from Phase_1 using a steady-state thermal flow calculation. In fact, the temperatures at the top and bottom are equal to the temperatures as defined in the **Initial phase**; however, since the temperature at the ground surface is now defined in terms of **Climate** conditions (air temperature), this temperature is also applied at the inner side of the lock and affects the temperature distribution in the ground.

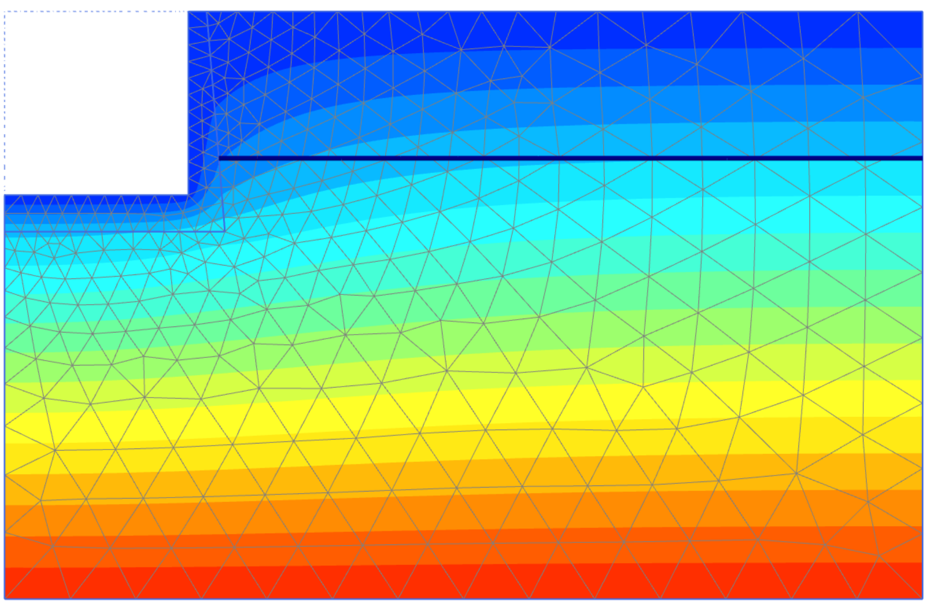


*Figure 239: Steady-state temperature distribution in Phase_1*

The most interesting results are obtained in Phase_2 in which the air temperature in the **Climate** condition increases gradually from 283 K to 298 K (defined by a quarter of a harmonic cycle with an amplitude of 15K). Figure 240 (on page 269) shows the temperature at the ground surface as a function of time.

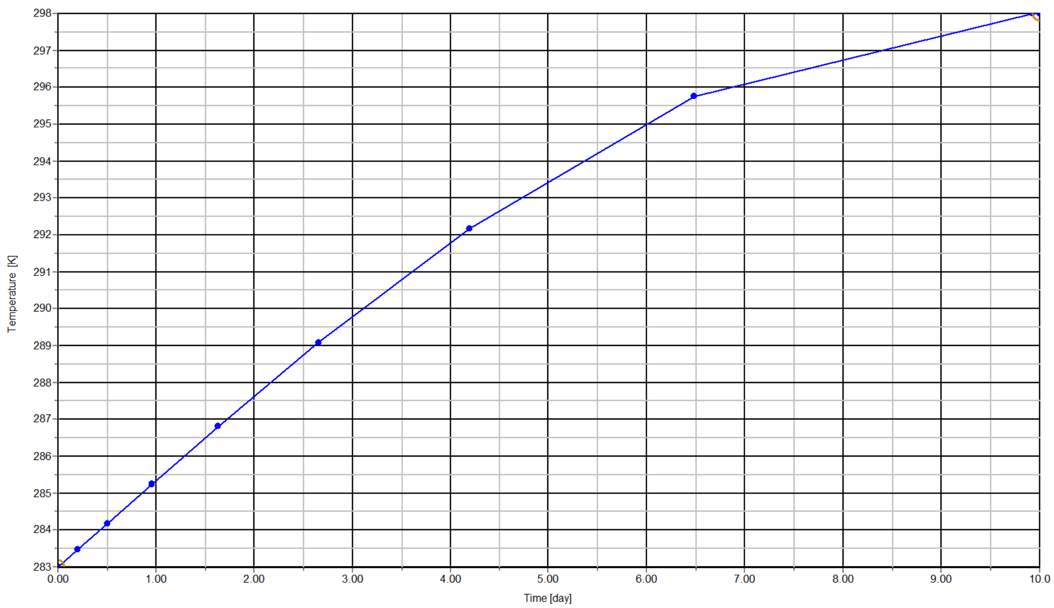



*Figure 240: Temperature distribution in Point A as a function of time*

As a result of the short increase in temperature at the inside of the concrete block, while the outer side (soil side) remains 'cold', the wall will bend towards the soil. Figure 241 (on page 269) shows the deformed mesh at the end of Phase_2.

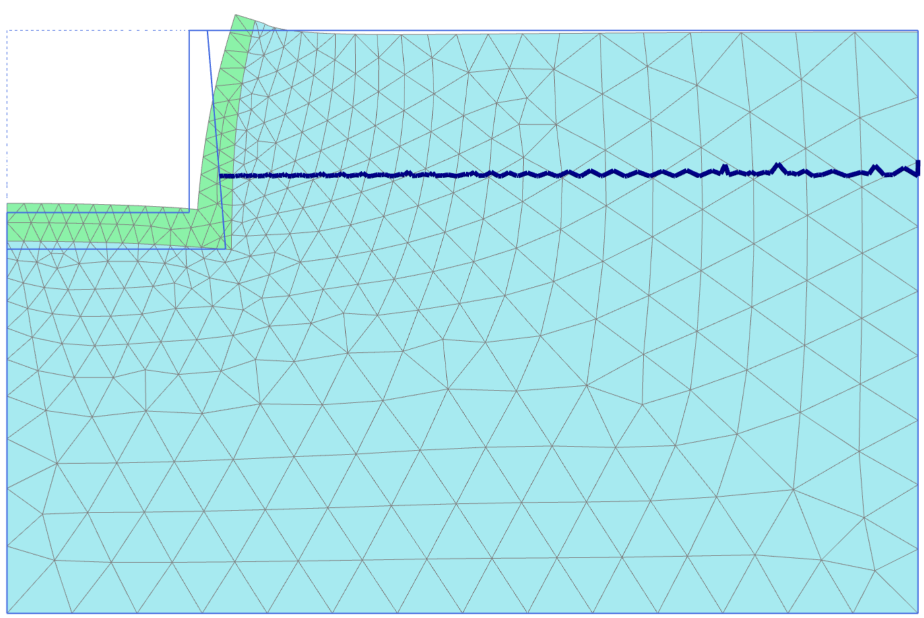


*Figure 241: Deformed mesh at the end of Phase_2*

As a result of this backward bending, the lateral stresses in the soil right behind the concrete block will increase, tending towards a passive stress state.

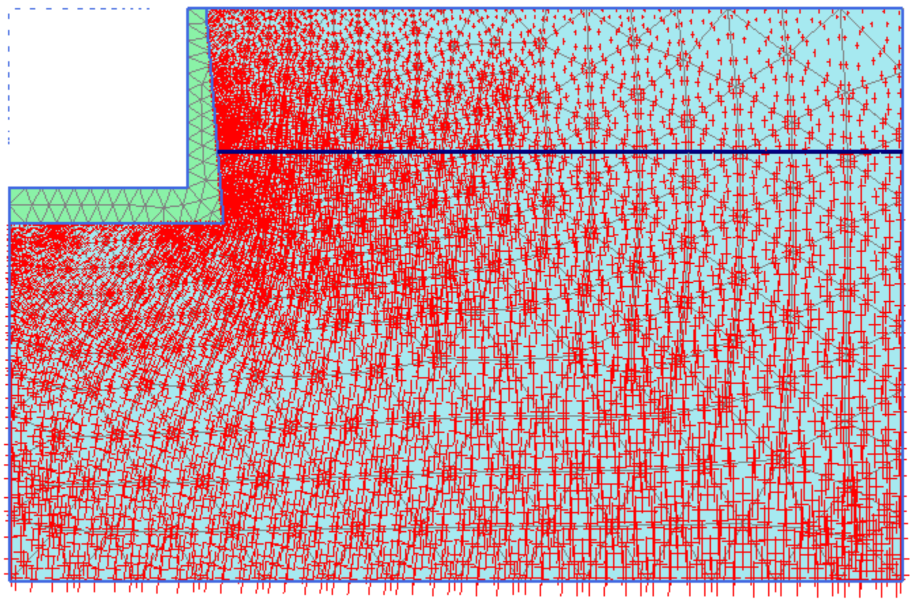


*Figure 242: Effective principal stresses at the end of Phase_2 in the Principal directions*

**Note:** Note that the visualisation is different for Figure 242 (on page 270), because it displays the stresses in the porous materials. This can be changed in **View** > **Settings** on the tab **Results** (see the Reference Manual for more information).

**Note:** Note that Figure 242 (on page 270) shows the principal stresses for all stress points whereas by default the principal stresses are only shown for the 3 center stress points. This can be changed using the  and  buttons on the navigation bar.

# 18 Freeze pipes in tunnel construction [ULT]

This tutorial illustrates change in coupling of groundwater flow and thermal flow as a result of ground freezing. A tunnel is constructed with the use of freeze pipes. By first installing freeze pipes in the soil, the soil freezes and becomes watertight so that tunnel construction can take place. This method of construction requires a lot of energy for the cooling of the soil, so by being able to model the cooling behaviour while groundwater flow is present an optimal freezing system can be designed.

**Objectives**

- Modelling soil freezing, coupling between thermal flow and groundwater flow
- Modelling unfrozen water content.
- Using the command line for structure definition.

**Geometry**

In this tutorial a tunnel with a radius of 3.0 m will be constructed in a 30 m deep soil layer. A groundwater flow from left to right is present, influencing the thermal behaviour of the soil. First the soil will be subjected to the low temperatures of the freeze pipes, and once the soil has frozen sufficiently, tunnel construction can take place. The latter is not included in this tutorial.

Because groundwater flow causes an asymmetric temperature distribution, the whole geometry needs to be modelled, where in previous examples only half of the geometry was sufficient. The geometry of the project is shown in Figure 243 (on page 271).

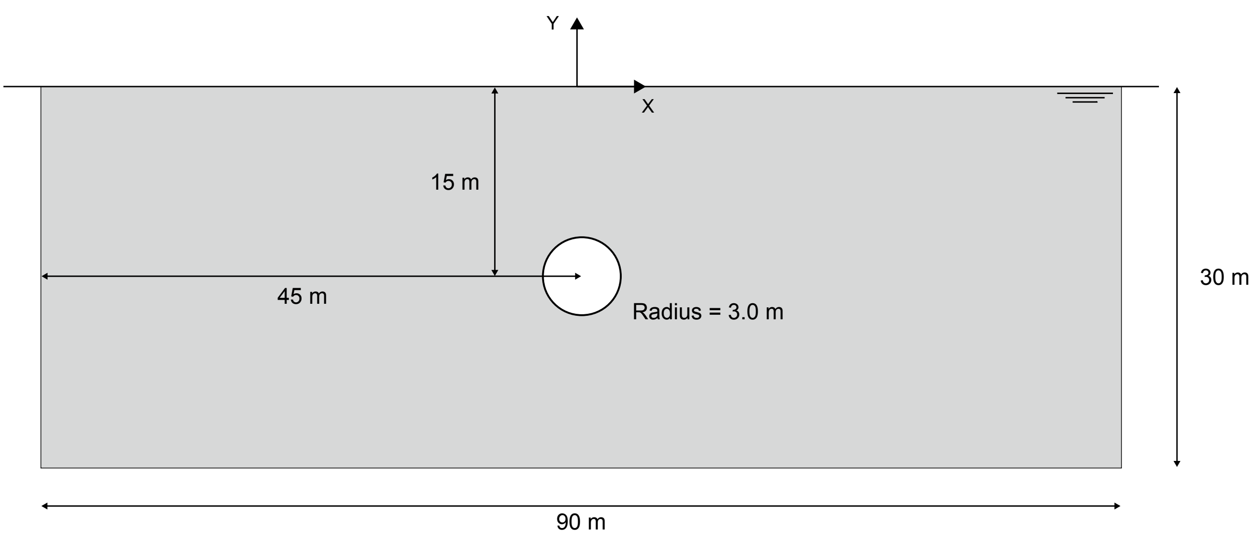



*Figure 243: Geometry of the project*

## 18.1 Create new project

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet, the default options for **Model** and **Elements** are used for this project. Also the default options for the units are used in this tutorial. Note that the unit of **Mass** is set automatically to **tonnes**.
4. Set the model **Contour** to $x_{min}$ = -45 m, $x_{max}$ = 45 m, $y_{min}$ = -30 m and $y_{max}$ = 0 m.
5. In the **Constants** tabsheet, set $T_{water}$ and $T_{ref}$ to 283 K, other constants keep their default values. A description of constants can be found in the Reference Manual.
6. Click **OK** to close the **Project properties** window.

## 18.2 Define the soil stratigraphy

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Create a single soil layer with top level at 0 m and bottom level at -30 m. Set the head at ground level (0 m).

## 18.3 Create and assign material data sets

1. Click the **Materials** button in the **Modify soil layers** window.
2. Define a data set for soil with the parameters given in Table 39 (on page 273), for the **General Parameters** and **Groundwater** tabsheets.

**Table 39: Material properties**

| Parameter | Name | Sand | Unit |
|---|---|---|---|
| **General** | | | |
| Soil model | - | Mohr-Coulomb | - |
| Drainage type | - | Drained | - |
| Unsaturated unit weight | $\gamma_{unsat}$ | 18 | $kN/m^3$ |
| Saturated unit weight | $\gamma_{sat}$ | 18 | $kN/m^3$ |
| Initial void ratio | $e_{init}$ | 0.5 | - |
| **Mechanical** | | | |
| Young's modulus | $E'_{\mathrm{ref}}$ | $100 \cdot 10^3$ | $kN/m^2$ |
| Poisson's ratio | $\nu$ | 0.3 | - |
| Cohesion | $c'_{ref}$ | 0 | $kN/m^2$ |
| Friction angle | $\varphi'$ | 37 | ° |
| Dilatancy angle | $\psi$ | 0 | ° |
| **Groundwater** | | | |
| Classification type | - | Standard | - |
| Soil - class | - | Medium | - |
| Flow parameters - Use defaults | - | None | - |
| Horizontal permeability | $k_x$ | 1 | m/day |
| Vertical permeability | $k_y$ | 1 | m/day |
| **Thermal** | | | |
| Specific heat capacity | $c_s$ | 860 | kJ/t/K |
| Thermal conductivity | $\lambda_s$ | $4 \cdot 10^{-3}$ | kW/m/K |
| Soil density | $\rho_s$ | 2.6 | $t/m^3$ |
| Thermal expansion type | - | Isotropic | - |
| Thermal expansion | $\alpha_{sv}$ | $0.015 \cdot 10^{-3}$ | 1/K |

| | | | |
|---|---|---|---|
| **Thermal** | | | |
| Unfrozen water saturation method | - | User defined (see table below) | - |
| **Interfaces** | | | |
| Strength determination | - | Rigid | - |
| Thermal resistance factor | $R_{thermal}$ | 0 | $m^2K/kW$ |
| **Initial** | | | |
| $K_0$ determination | - | Automatic | - |

To model the amount of (fluid) water available to flow through the soil at certain temperatures, a curve for unfrozen water content needs to be determined by defining a table with values for unfrozen water content at certain temperatures. The same curve can be applied in other projects, hence the table can be saved and loaded into the soil properties of other projects. For more information, refer to the Reference Manual.

3. Click the **Thermal** tab. Enter the values as given in Table 40 (on page 274) .

**Table 40: Input for unfrozen water content curve for sand**

| # | Temperature [K] | Unfrozen water content [-] |
|---|---|---|
| 1 | 273.0 | 1.00 |
| 2 | 272.0 | 0.99 |
| 3 | 271.6 | 0.96 |
| 4 | 271.4 | 0.90 |
| 5 | 271.3 | 0.81 |
| 6 | 271.0 | 0.38 |
| 7 | 270.8 | 0.15 |
| 8 | 270.6 | 0.06 |
| 9 | 270.2 | 0.02 |
| 10 | 269.5 | 0.00 |

4. Select the option **User defined** from the drop down menu for **Unfrozen water content** at the bottom of the tabsheet.
5. Add rows to the table by clicking the **Add row** button. Complete the data using the values given in Table 40 (on page 274).
6. Enter the values for **Interfaces** and **Initial** tabsheets as given in Table 39 (on page 273).
7. Click **OK** to close the dataset.

8. Assign the material dataset to the soil layer.

**Note:**

The table can be saved by clicking the **Save** button in the table. The file must be given an appropriate name. For convenience, save the file in the same folder as the project is saved.

## 18.4 Define the structural elements

The freeze pipes are modelled by defining lines with a length similar to the freeze pipe diameter (10 cm), containing a convective boundary condition. For simplicity, in this tutorial only 12 cooling elements are defined, while in reality more elements may be implemented in order to achieve a sufficient share of frozen soil.

1. Proceed to **Structures mode** .
2. Click the **Create line** button in the side toolbar.
3. Click the command line and type

   ```
   line 0 -12.05 0 -11.95
   ```

   Press **Enter** to create the first freezing pipe. For more information regarding command line, see Reference Manual .
4. The remaining freeze pipes will be created by copying the first freeze pipe 12 times. To do so select the line representing the freeze pipe that was just created.
5. From the side toolbar select the **Create array** option . Set the **Array pattern** to **Polar** as we want to create a circle of freeze pipes, the **Center point** is (x y) = (0 -15), the **Total number of items** is 12 (the original plus 11 copies) and finally the **Angle to fill** must be set to 360 degrees as we want to create a full circle. Press OK to create the 11 additional freeze pipes.
6. Multi select the 12 lines representing the freeze pipes using the **Select lines** option from the side toolbar.
7. Right click the selected lines and select **Thermal flow BC** to create the thermal flow boundary conditions for the freeze pipes.
8. For the selected freeze pipes, in the **Selection explorer** expand the subtree for the **ThermalFlowBC**.
9. The **Behaviour** is set to **Convection**, the $T_{fluid}$ to 250 K and the **Transfer coefficient** to 1.0 kW/m$^2$/K.
10. Click the **Create line** button in the side toolbar.
11. Select the **Create thermal flow BC** option in the expanded menu. In the drawing area create a thermal boundary condition along the perimeter of the model, hence from (x y) = (-45 0) to (45 0), (45 -30), (-45 -30) and back to (-45 0).
12. Select the four boundaries just have been created and right-click on them. From the menu that pops up select **Create** and **Create groundwater flow BC** to add groundwater flow boundary conditions to the thermal flow boundary conditions.

PLAXIS 2D allows different types of **Thermal** boundary conditions to be applied. In this tutorial the freeze pipes will be modelled as convective boundary conditions.

1. Multi select the created boundaries.
2. For the **ThermalFlowBC**, set the **Behaviour** to **Temperature** and $T_{ref}$ to 283 K.

To assign the groundwater boundary conditions, the following steps are followed:

1. Multi select the top and bottom boundary.
2. For the **GWFlowBC**, set the **Behaviour** to **Closed**.
3. Select the left boundary, set the **Behaviour** to **Inflow** with a $q_{ref}$ value of 0.1 m/day.
4. The right boundary has the default behaviour of **Seepage**.

The tunnel is created with the help of the **Tunnel designer**. Because deformations are not considered in this calculation, there is no need to assign a plate material to the tunnel. The generated tunnel will only be used for generating a more dense and homogeneous mesh around the freezing pipes. The tunnel will not be activated during any calculation phase, but PLAXIS 2D will detect the line elements and will generate the mesh according to these elements. Changing the coarseness factor of the pipe elements will cause a denser, but not a more homogeneous mesh.

1. Click the **Create tunnel** button in the side toolbar and click on (0 -18) in the drawing area.
2. The option **Circular** is selected for **Shape type**. Note that the default option is **Free**.
3. The default option of **Define whole tunnel** is used in this example.
4. Proceed to the **Segments** tab and set **Radius** to 3 m to the two multi selected segments.
5. Click on **Generate** to generate the defined tunnel in the model. Close the **Tunnel designer** window.

The geometry of the model is shown in Figure 244 (on page 276) .

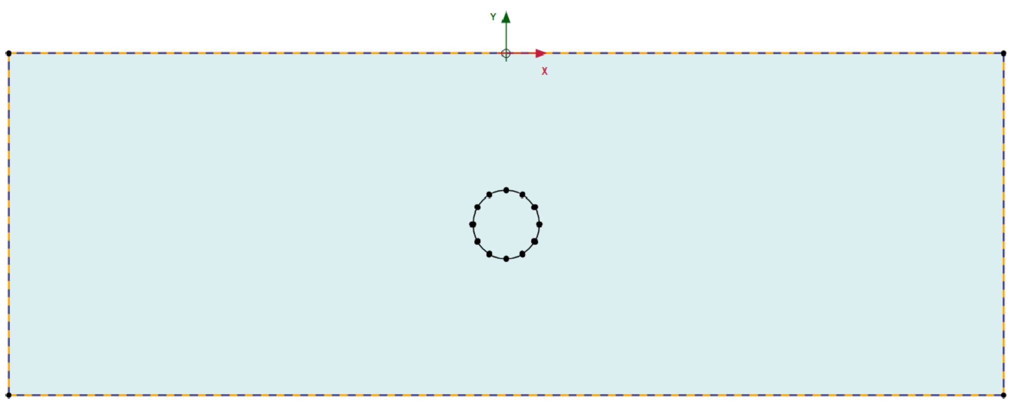


*Figure 244: Geometry of the model*

## 18.5 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Click the **Generate mesh** button to generate the mesh. The default element distribution of **Medium** is used for this example.
3. Click the **View mesh** button to view the mesh.

   The resulting mesh is shown in Figure 245 (on page 277):

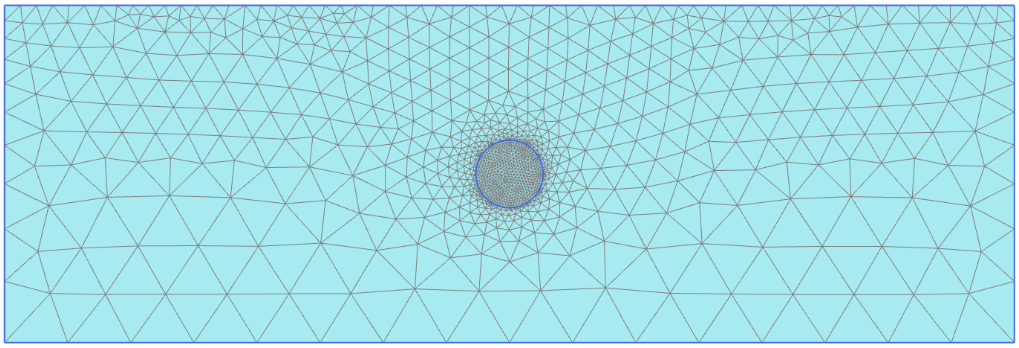


*Figure 245: The generated mesh*

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

## 18.6 Define and perform the calculation

The calculations for this tutorial are carried out in the **Flow only mode**.

### 18.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with the definition of the calculation phases.
2. Double click on **Initial phase** in the **Phases explorer**.
3. In the **Phases** window select the **Flow only** option from the **Calculation type** drop-down menu.
4. Choose the **Earth gradient** option for the **Thermal calculation type**.
5. In the **Staged construction** activate the **ThermalFlow** under the **Model conditions** subtree and set the value for $T_{ref}$ to 283 K, $h_{ref}$ to 0 m and 0 K/m for the **Earth gradient**.
6. The model for initial phase is shown in Figure 246 (on page 277).

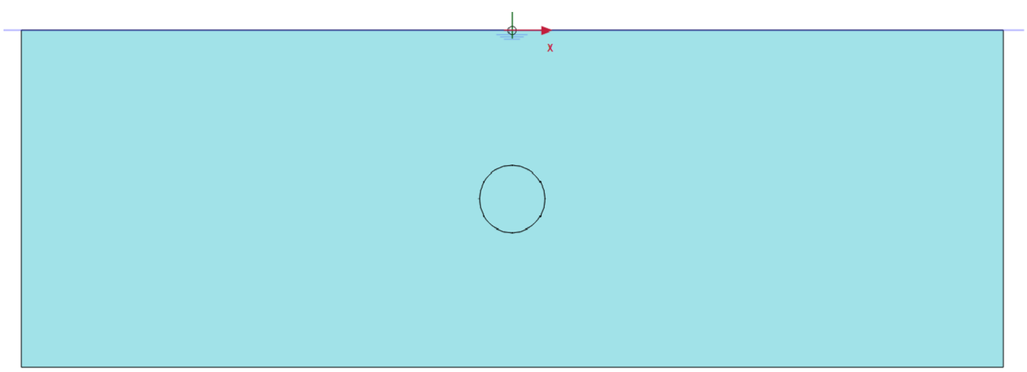


*Figure 246: Initial phase*

### 18.6.2 Phase 1: Transient calculation

1. Click the **Add phase** button to create a new phase.
2. Double click the new phase in the **Phases explorer**.
3. In the **Phases** window, enter an appropriate name for the phase ID (e.g. `Transient calculation`).
4. Set **Transient groundwater flow** as the option for the **Pore pressure calculation type**.
5. Set **Transient thermal flow** as the option for the **Thermal calculation type**.
6. Set **Time interval** to 180 days and the **Max number of steps stored** to 100. This is to be able to view intermediate time steps after the calculation.
7. In **Staged construction** mode, activate all the thermal boundary conditions by clicking the check box for the **Thermal flow BCs** in the **Model explorer**.
8. In the **Model explorer**, activate the four groundwater flow boundary conditions corresponding to the left, top, right and bottom boundary conditions in the **Groundwater flow BCs** subtree.

### 18.6.3 Execute the calculation

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of curves.

1. Click the **Select points for curves** button in the side toolbar. Select at least the node in the middle of the model and some characteristic points for curves (for example between two freezing pipes).
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button.

## 18.7 Results

Interesting results from this calculation can be the point in time when there is no groundwater flow in between two freezing pipes, groundwater flow over the whole model and temperature distribution for both steady state and transient calculations.

To view the results in the Output program:

1. Click the **View calculation results** button on the toolbar.
2. Select the menu **Stresses** > **Heat flow** > **Temperature**.
3. Figure 247 (on page 279) shows the spatial distribution of the temperature for transient calculation in the final step. Note that the element contours have been switched off to better show that the temperature of the soil inside the tunnel is below freezing.

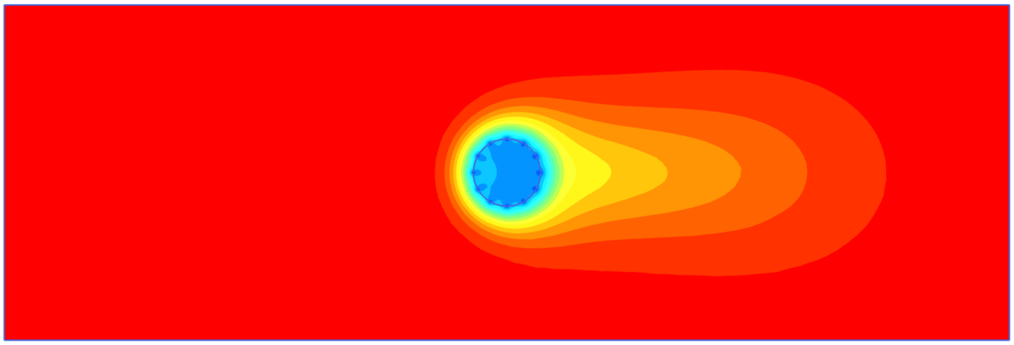


*Figure 247: Temperature distribution for transient phase*

1. Select the menu **Stresses** > **Groundwater flow** > **|q|**.
2. Select the menu **View** > **Arrows** or click the corresponding button in the toolbar to display the results arrows.

In the Output program, it is possible to view the results for the intermediate saved steps. More information is available in the Reference Manual . It is possible to view the progression of the freezing of the tunnel.

Figure 248 (on page 279) shows the distribution of the of groundwater flow field for an intermediate step for the transient calculation (around 38 days).

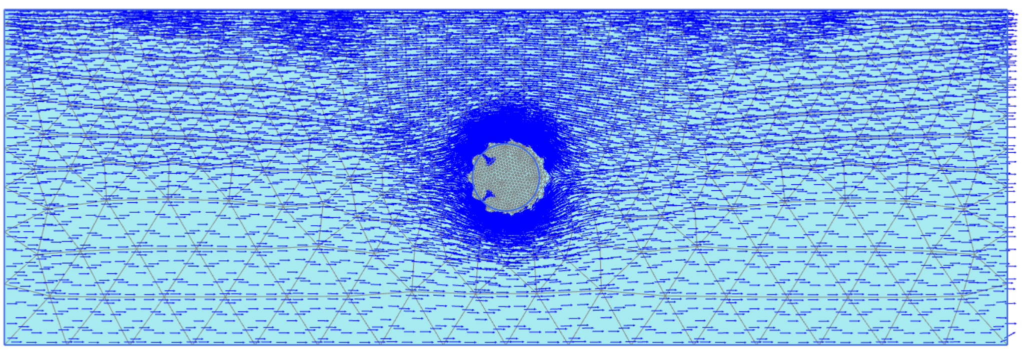


*Figure 248: Groundwater flow field for transient phase for an intermediate step ($t_{approx}$ 38 days)*

Figure 249 (on page 280) shows the groundwater flow field for the last time step for the transient flow calculation. Here it is clearly noticeable that the entire tunnel area is frozen and no flow occurs.

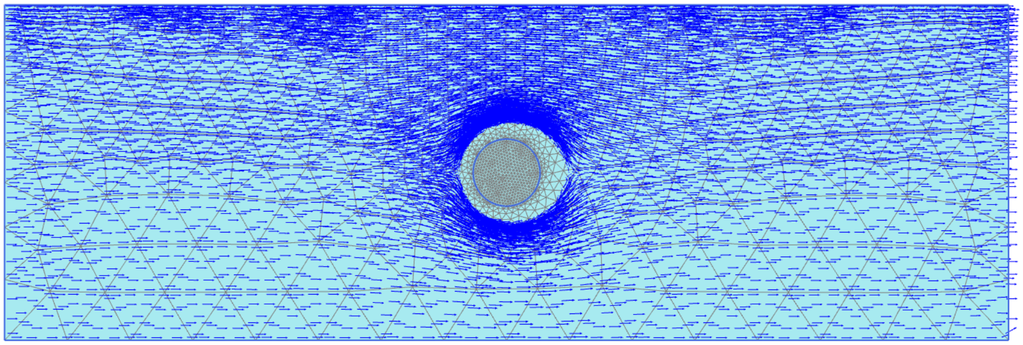


*Figure 249: Groundwater flow field after 180 days*

Figure 250 (on page 280) shows the change of temperature in time for the point in the middle of the tunnel. It can be seen that the temperature drops quite fast until about 273 K when the pore water starts to change from water to ice. During this process the temperature remains almost constant and only until after all pore water has turned to ice (at t = 122 s) the ice temperature drops further.

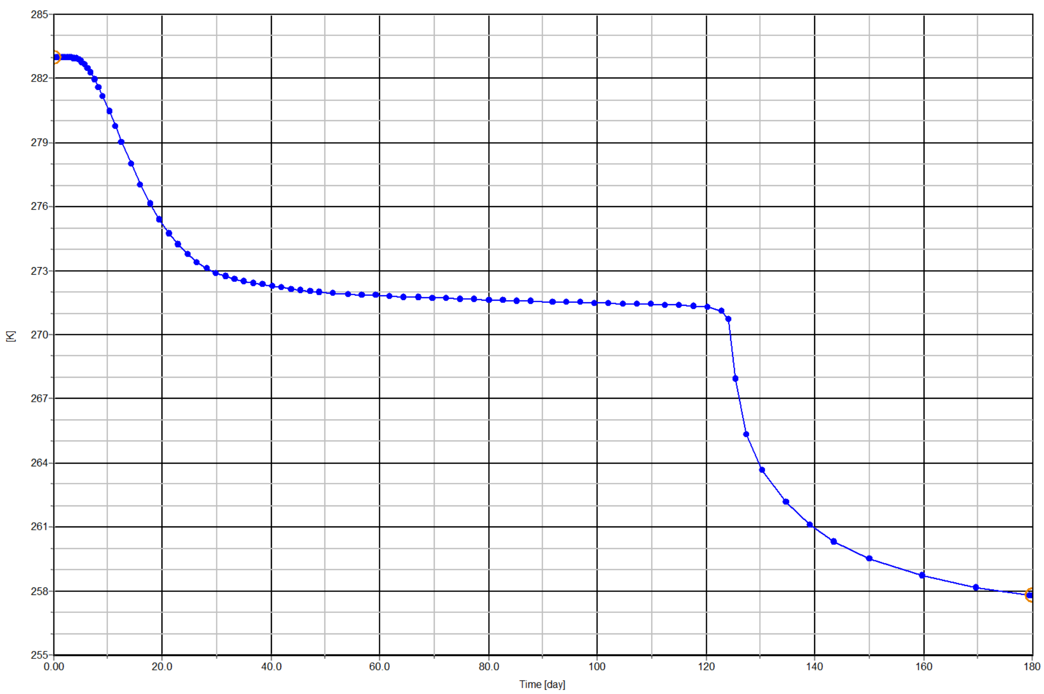



*Figure 250: Temperature drop in time at the center of the tunnel*