# Table of Contents

## Chapter 1: Introduction ............................................................................................................. 5

## Chapter 2: Foundation in overconsolidated clay ............................................................................. 6

2.1 Case A: Rigid foundation ................................................................................................................. 7

2.2 Case B: Raft foundation ....................................................................................................................... 22

2.3 Case C: Pile-Raft foundation .............................................................................................................. 30

## Chapter 3: Excavation in sand ............................................................................................................. 36

3.1 Create a new project .......................................................................................................................... 37

3.2 Define the soil stratigraphy .............................................................................................................. 38

3.3 Create and assign material data sets ................................................................................................. 38

3.4 Define the structural elements ......................................................................................................... 40

3.5 Generate the mesh ............................................................................................................................. 43

3.6 Define the calculation ......................................................................................................................... 44

3.7 View the calculation results .............................................................................................................. 45

## Chapter 4: Loading of a suction pile ................................................................................................. 51

4.1 Create a new project .......................................................................................................................... 51

4.2 Define the soil stratigraphy .............................................................................................................. 52

4.3 Create and assign material data sets ................................................................................................. 52

4.4 Define the structural elements ......................................................................................................... 54

4.4.1 Create a suction pile ...................................................................................................................... 54

4.4.2 Create helper objects for local mesh refinements ........................................................................... 58
PLAXIS 3D 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 3D. 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 may be 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 organized on a regular basis at several places in the world to provide hands-on experience and background information on the use of the program.
In this chapter a first application of PLAXIS 3D is considered, namely the settlement of a foundation in clay. This is the first step in becoming familiar with the practical use of the PLAXIS 3D program.

The general procedures for the creation of a geometry, 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 tutorial will be utilised in the following tutorials. Therefore, it is important to complete this first tutorial before attempting any further tutorial examples.

**Geometry**

This exercise deals with the construction and loading of a foundation of a square building in a lightly overconsolidated lacustrine clay. Below the clay layer there is a stiff rock layer that forms a natural boundary for the considered geometry. The rock layer is not included in the geometry; instead an appropriate boundary condition is applied at the bottom of the clay layer. The purpose of the exercise is to find the settlement of the foundation.

The building consists of a basement level and 5 floors above the ground level. To reduce calculation time, only one-quarter of the building is modelled, using symmetry boundary conditions along the lines of symmetry. To enable any possible mechanism in the clay and to avoid any influence of the outer boundary, the model is extended in both horizontal directions to a total width of 75 m.

The model is considered in three different cases:

- Case A: The building is considered very stiff and rough. The basement is simulated by means of non-porous linear elastic volume elements.
- Case B: The structural forces are modelled as loads on a raft foundation.
- Case C: Embedded beams are included in the model to reduce settlements.
2.1 Case A: Rigid foundation

In this case, the building is considered to be very stiff. The basement is simulated by means of non-porous linear elastic volume elements. The total weight of the basement corresponds to the total permanent and variable load of the building. This approach leads to a very simple model and is therefore used as a first exercise, but it has some disadvantages. For example it does not give any information about the structural forces in the foundation.

Objectives

- Starting a new project
- Creation of soil stratigraphy using a single borehole
- Creation of material data sets
- Creation of volumes using Create surface and Extrude tools
- Assigning material
- Local mesh refinement
- Generation of the mesh

Figure 1: Geometry of a square building on a raft foundation
Foundation in overconsolidated clay

Case A: Rigid foundation

- Generating initial stresses using the $K_0$ procedure
- Defining a Plastic calculation

2.1.1 Create a new project

1. Start PLAXIS 3D 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.

2. Click Start a new project.
   The Project properties window appears with the tabsheets: Project, Model and Cloud services.
Foundation in overconsolidated clay  
Case A: Rigid foundation  

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 Tutorial 1 in the Title box and type Settlements of a foundation in the Comments box.
4. Click the Next button at the bottom or click the Model tab. The Model properties are shown:

5. Keep the default units in the Units box (Length = m; Force = kN; Time = day).
6. The General box indicates a fixed gravity of 1.0 G, in the vertical downward direction (-z).
7. In the γ\text{water} box the unit weight of water can be defined. Keep this to the default value of 10 kN/m³.
8. In the **Contour** group set the model dimensions to
   a. \( x_{\text{min}} = 0.0 \) and \( x_{\text{max}} = 75.0 \),
   b. \( y_{\text{min}} = 0.0 \) and \( y_{\text{max}} = 75.0 \).
9. 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 them by selecting the menu **File > Project properties** ...

### 2.1.2 Define the soil stratigraphy

In the **Soil** mode of PLAXIS 3D 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 3D will automatically interpolate between the boreholes and derive the position of the soil layers from the borehole information.

**Note:** PLAXIS 3D can also deal with layers that are discontinuous, i.e. only locally present in the model area. See the info on Multiple boreholes of the Reference Manual for more information.

In the current example, only one soil layer is present, and only a single borehole is needed to define the soil stratigraphy. In order to define the borehole, follow these steps:

1. Click the **Create borehole** button in the side toolbar to start defining the soil stratigraphy.
2. Click on position \((0 \ 0 \ 0)\) in the geometry.
   A borehole will be located at \((x,y) = (0 \ 0)\).
   The **Modify soil layers** window will appear.
3. Add a soil layer by clicking the **Add** button in the **Modify soil layers** window.
4. Keep the top boundary of the soil layer at \(z = 0\) and set the bottom boundary to \(z = -40\) m.
5. Set the **Head** to -2.0 m.
Next, the material data sets are defined and assigned to the soil layers.

### 2.1.3 Create and assign material data sets

In order to simulate the behaviour of the soil, a suitable material model and appropriate material parameters must be assigned to the geometry. In PLAXIS 3D 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 clusters. For structures (like beams, plates, etc.) the system is similar, but different types of structures have different parameters and therefore different types of data sets.

PLAXIS 3D distinguishes between material data sets for Soils and interfaces, Plates, Geogrids, Beams, Embedded beams and Anchors.

The materials used in this tutorial have the following properties:

**Table 1: Material properties**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Lacustrine clay</th>
<th>Building</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Mohr-Coulomb model</td>
<td>Linear Elastic model</td>
<td>-</td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td>Drained</td>
<td>Non-porous</td>
<td>-</td>
</tr>
<tr>
<td>Unit weight above phreatic level</td>
<td>$\gamma_{\text{unsat}}$</td>
<td>17.0</td>
<td>50</td>
<td>kN/m$^3$</td>
</tr>
</tbody>
</table>
Parameter | Name | Lacustrine clay | Building | Unit |
---|---|---|---|---|
Unit weight below phreatic level | $\gamma_{sat}$ | 18.0 | - | kN/m$^3$ |

### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit weight below phreatic level</th>
<th>Building</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\gamma_{sat}$</td>
<td>18.0</td>
<td>-</td>
<td>kN/m$^3$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Young's modulus (constant)</th>
<th>$E'$</th>
<th>1·10$^4$</th>
<th>3·10$^7$</th>
<th>kN/m$^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson's ratio</td>
<td>$\nu'$</td>
<td>0.3</td>
<td>0.15</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>$c'_{ref}$</td>
<td>10</td>
<td>-</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\varphi'$</td>
<td>30.0</td>
<td>-</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>0.0</td>
<td>-</td>
<td>°</td>
</tr>
</tbody>
</table>

### Initial

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit weight below phreatic level</th>
<th>Building</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_0$</td>
<td>-</td>
<td>Automatic</td>
<td>Automatic</td>
</tr>
<tr>
<td>Lateral earth pressure</td>
<td>$K_0$</td>
<td>0.5000</td>
<td>0.5000</td>
</tr>
</tbody>
</table>

To create the material sets for this tutorial, follow these steps:

1. Click the **Materials** button in the **Modify soil layers** window or in the side toolbar. The **Material sets** window pops up.

   **Note:** If necessary the **Modify soil layers** window can be re-opened by double-clicking the borehole in the drawing area or by selecting the menu **Soil > Modify soil layers**.

### Create the Lacustrine clay material set

First create the material set for the clay:

1. Click the **New** button at the lower side of the **Material sets** window.

   The **Soil** window will appear. It contains five tabsheets: **General, Parameters, Groundwater, Interfaces** and **Initial**.

2. In the **General** tabsheet, **Material set** section, **Identification** field, type **Lacustrine Clay**.

3. Select Mohr-Coulomb model from the **Material model** drop-down menu and **Drained** from the **Drainage type** drop-down menu.

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

4. Enter the unit weights in the **General properties** box according to the material data as listed in **Table 1** (on page 11). Keep the unmentioned **Advanced parameters** as their default values.
5. Click the **Next** button or click the **Parameters** tab to proceed with the input of model parameters. The parameters appearing on the **Parameters** tabsheet depend on the selected material model (in this case the Mohr-Coulomb model). The Mohr-Coulomb model involves only five basic parameters ($E'$, $v'$, $c'_{ref}$, $\varphi'$, $\psi'$). See the Material Models Manual for a detailed description of the different soil models and their corresponding parameters.

6. Enter the model parameters $E'$, $v'$, $c'_{ref}$, $\varphi'$ and $\psi$ of **Lacustrine clay** according to Table 1 (on page 11) in the corresponding boxes of the **Parameters** tabsheet.
7. No consolidation will be considered in this exercise. As a result, the permeability of the soil will not influence the results and the Groundwater window can be skipped.

8. Since the geometry model does not include interfaces, the Interfaces tab can be skipped.

9. Click the Initial tab and check that the \textit{K0 determination} is set to \textit{Automatic}. In that case $K_0$ is determined from Jaky’s formula: $K_0 = 1 - \sin \phi$.

10. Click the \textit{OK} button to confirm the input of the current material data set. The created data set appears in the tree view of the Material sets window.

11. Drag the set \textit{Lacustrine clay} 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).

\textbf{Note:} Notice that the cursor changes shape to indicate whether or not it is possible to drop the data set. Correct assignment of the data set to the soil layer is indicated by a change in the colour of the layer.

Create the Building material set

The building is modelled by a linear elastic non-porous material. To define this data set, follow these steps:

1. Click the \textit{New} button in the Material sets window.

2. In the \textit{General} tabsheet, \textit{Material set} section, \textit{Identification} field, type Building.

3. Select Linear Elastic model from the \textit{Material model} drop-down menu and \textit{Non-porous} from the \textit{Drainage type} drop-down menu.

4. Enter the unit weight in the \textit{General properties} box according to the material data as listed in Table 1 (on page 11). This unit weight corresponds to the total permanent and variable load of the building.

5. Click the \textit{Next} button or click the Parameters tab to proceed with the input of model parameters. The linear elastic model involves only two basic parameters ($E', \nu'$).

6. Enter the model parameters of Table 1 (on page 11) in the corresponding edit boxes of the Parameters tabsheet.

7. Click the \textit{OK} button to confirm the input of the current material data set. The created data set will appear in the tree view of the Material sets window, but it is not directly used.

8. Click the \textit{OK} button to close the Material sets window.

9. Click the \textit{OK} button to close the Modify soil layers window.

\textbf{Note:} PLAXIS 3D 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 \textit{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.

2.1.4 Define the structural elements

The structural elements are created in the Structures mode of the program.

To model the building:

1. Click the Structures tab to proceed with the input of structural elements in the Structures mode.
2. Click the *Create surface* button 🍁. Position the cursor at the coordinate (0 0 0). Check the cursor position displayed in the cursor position indicator. As you click, the first surface point of the surface is defined.

3. Define three other points with coordinates (0 18 0), (18 18 0), (18 0 0) respectively. Right-click or press `<Esc>` to finalize the definition of the surface. Note that the created surface is still selected and displayed in red.

4. Click the *Extrude object* button ⤵️ to create a volume from the surface. The *Extrude* window pops up.

![Extrude window](image)

*Figure 5: Extrude window*

5. Change the value of z to -2 and click *Apply* to close the window.

6. Click the *Select* button 📢.

7. Right-click the created surface and select *Delete* from the appearing menu. This will delete the surface but the building volume is retained.

The building volume, as well as the corresponding material data sets have now been created.

### 2.1.5 Generate the mesh

The model is complete. PLAXIS 3D allows for a fully automatic mesh generation procedure, in which the geometry is divided into volume elements and compatible structure elements, if applicable. The mesh generation takes full account of the position of the geometry entities in the geometry model, so that the exact position of layers, loads and structures is accounted for in the finite element mesh. A local refinement will be considered in the building volume.

**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 (see *Mesh Generation* in 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.

To generate the mesh, follow these steps:

1. Proceed to the **Mesh** mode by clicking the corresponding tab.
2. Click the **Refine mesh** button in the side toolbar and click the created building volume to refine the mesh locally. It will colour green.
3. Click the **Generate mesh** button in the side toolbar or select the menu **Mesh > Generate mesh**.
4. Change the **Element distribution** to **Coarse** in the **Mesh options** window.
5. Click **OK** to start the mesh generation.

6. After the mesh is generated, click the **View mesh** button 🌐.
   A new window is opened displaying the generated mesh.

7. Click on the **Close** tab to close the Output program and go back to the **Mesh** mode of the **Input** program.
2.1.6 Define and perform the calculation

Once the mesh has been generated, the finite element model is complete. Now the calculation phases have to be defined.

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. The initial water level has been entered already in the Modify soil layers window. This level is taken into account to calculate the initial effective stress state. It is therefore not needed to enter the Flow conditions mode.

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

1. Click the Staged construction tab to proceed with the definition of calculation phases.

   When a new project has been defined, a first calculation phase named 'Initial phase', is automatically created and selected in the Phases explorer:

   ![Phases explorer]

   Figure 9: Phases explorer

   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.

   The Phases window is displayed.
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.

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>By default the $K_0$ procedure is selected as <strong>Calculation type</strong> in the <strong>General</strong> subtree of the <strong>Phases window</strong>. This option will be used in this project to generate the initial stresses.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The <strong>Staged construction</strong> option is selected as <strong>Loading type</strong>. This is the only option available for the <strong>$K_0$ procedure</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The <strong>Phreatic</strong> option is selected by default as the <strong>Pore pressure calculation type</strong>.</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** The $K_0$ procedure may only be used for horizontally layered geometries with a horizontal ground surface and, if applicable, a horizontal phreatic level. See "Types of Analysis" in the Reference Manual for more information on the **$K_0$ procedure**.

3. The other default options in the **Phases** window will be used as well in this tutorial.
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 generated according to the **Head** value assigned to boreholes in the **Modify soil layers** window (BoreholeWaterLevel_1) is automatically assigned to **GlobalWaterLevel**.
7. Make sure that all the soil volumes in the project are active and the material assigned to them is **Lacustrine clay**.

**Phase 1: Construction stage**

After the definition of the initial conditions, the construction of the building can be modelled. This will be done in a separate calculation phase, which needs to be added as follows:

1. Click the **Add phase** button in the **Phases explorer**.
A new phase, named Phase 1 will be added in the **Phases explorer**.

**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.

2. Double click **Phase 1** to open the **Phases** window.
3. In the **ID** box of the **General** subtree, write (optionally) an appropriate name for the new phase (for example **Building**).
4. The current phase starts from **Initial phase**, which contains the initial stress state. The default options and values assigned are valid for this phase.

![Figure 11: The Phases window for Building phase](image)

5. Click **OK** to close the **Phases** window.
6. Right-click the building volume that was created earlier. Select the menu **Set material > Building**.
   The Building data set is now assigned to the building volume.

**Execute the calculation**

All calculation phases (two phases in this case) are marked for calculation, indicated by a blue arrow. The execution order is controlled by the **Start from phase** parameter.

1. Click the **Calculate** button to calculate the project. 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.
Figure 12: Active task window displaying the calculation progress

The information, which is continuously updated, shows, amongst others, 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.

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

3. Click the Save button to save the project before viewing results.

2.1.7 View the calculation results

Once the calculation has been completed, the results can be displayed in the Output program. In the Output program, the displacement and stresses in the full three-dimensional model as well as in cross sections or structural elements can be viewed. The computational results are also available in tabular form.

To view the current results, follow these steps:

1. Select the last calculation phase (Building) in the Phases explorer tree.
2. Click the **View calculation results** button in the side toolbar to open the **Output** program. The **Output** program will, by default, show the three-dimensional deformed mesh at the end of the selected calculation phase. The deformations are scaled to ensure that they are clearly visible.

3. Select the menu **Deformations > Total Displacements > |u|**.
   The plot shows colour shadings of the total displacements. A legend is presented with the displacement values at the colour boundaries. When the legend is not present, select the menu **View > Legend** to display it.

![Figure 13: Shadings of Total displacements at the end of the last phase](image)

4. In the **Output** window click the **Iso surfaces** button to display the areas having the same displacement.

**Note:**

- The **Deformations** menu, in addition to the **Total displacements**, allows for the presentation of **Incremental displacements** and **Phase displacements**.
  - Incremental displacements are the displacements that occurred in one calculation step (in this case the final step). Incremental displacements may be helpful in visualising failure mechanisms.
  - Phase displacements are the displacements that occurred in one calculation phase (in this case the last phase). Phase displacements can be used to inspect the impact of a single construction phase, without the need to reset displacements to zero before starting the phase.

### 2.2 Case B: Raft foundation

In this case, the model is modified so that the basement consists of structural elements. This allows for the calculation of structural forces in the foundation.

The raft foundation consists of a 50 cm thick concrete floor stiffened by concrete beams. The walls of the basement consist of 30 cm thick concrete. The loads of the upper floors are transferred to the floor slab by a
column and by the basement walls. The column bears a load of 11650 kN and the walls carry a line load of 385 kN/m, as sketched in the following figure.

![Figure 14: Geometry of the basement](image)

In addition, the floor slab is loaded by a distributed load of 5.3 kN/m$^2$. The properties of the clay layer will be modified such that stiffness of the clay will increase with depth.

**Objectives**

- Saving project under a different name
- Modifying existing data sets
- Defining a soil stiffness that increases with depth
- Modelling of plates and defining material data set for plates
- Modelling of beams and defining material data set for beams
- Assigning point loads
- Assigning line loads
- Assigning distributed loads to surfaces
- Deleting phases
- Activation and deactivation of soil volumes
- Activation and deactivation of structural elements
- Activation of loads
- Zooming in Output
- Drawing cross sections in Output
- Viewing structural output

### 2.2.1 Create a new project

The geometry used in this exercise is the same as the previous one, except that additional elements are used to model the foundation. You can start from the previous project, store it under a different name and then modify it. To perform this, follow these steps:

1. Start PLAXIS 3D by double clicking the icon of the Input program.
The **Quick select** dialog box appears.
2. In the list Recent projects select the project of Case A.
   The previous project opens.
3. Select **File > Save project as** to save the project under a different name (e.g. **Tutorial 1b**).

### 2.2.2 Create and assign a material data set

The material set for the clay layer has already been defined. To modify this material set to take into account the stiffness of the soil increasing with depth, follow these steps:

1. Click the **Materials** button in the side toolbar. The **Material sets** window pops up.
2. Make sure that the option **Soil and interfaces** is selected as **Set type**.
3. Select the **Lacustrine clay** material set and click the **Edit** button.
4. In the **Parameters** tabsheet, change the stiffness of the soil $E'$ to 5000 kN/m$^2$.
5. Enter a value of 500 in the $E'_{inc}$ box in the **Advanced** parameters. Keep the default value of 0.0 m for $z_{ref}$. Now the stiffness of the soil is defined as 5000 kN/m$^2$ at $z=0.0$ m and increases with 500 kN/m$^2$ per meter depth.
6. Click **OK** to close the **Soil** window.
7. Click **OK** to close the **Material sets** window.

### 2.2.3 Define the structural elements

Proceed to the **Structures** mode to define the structural elements that compose the basement.
A number of material data sets will be created with the following material properties.

**Table 2: Material properties of the basement floor and basement walls**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Basement floor</th>
<th>Basement wall</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of behaviour</td>
<td>Type</td>
<td>Elastic, isotropic</td>
<td>Elastic, isotropic</td>
<td>-</td>
</tr>
<tr>
<td>Thickness</td>
<td>$d$</td>
<td>0.5</td>
<td>0.3</td>
<td>m</td>
</tr>
<tr>
<td>Weight</td>
<td>$\gamma$</td>
<td>15</td>
<td>15.5</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>$E_1$</td>
<td>$3\cdot10^7$</td>
<td>$3\cdot10^7$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu_{12}$</td>
<td>0.15</td>
<td>0.15</td>
<td>-</td>
</tr>
</tbody>
</table>

**Note:** When specifying a unit weight, please consider the fact that the element itself does not occupy any volume and overlaps with the soil elements. Hence, it might be considered to subtract the unit soil weight from the real unit weight of the plate, beam or embedded beam material in order to compensate for the overlap. For partially overlapping plates, beams or embedded beams the reduction of the unit weight should be proportional.
### Table 3: Material properties of the basement column and basement beams

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Basement column</th>
<th>Basement beam</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material type</td>
<td>Type</td>
<td>Elastic</td>
<td>Elastic</td>
<td>-</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>$3 \times 10^7$</td>
<td>$3 \times 10^7$</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Volumetric weight</td>
<td>γ</td>
<td>24.0</td>
<td>6.0</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Cross section area</td>
<td>A</td>
<td>0.49</td>
<td>0.7</td>
<td>m²</td>
</tr>
<tr>
<td>Moment of Inertia</td>
<td>$I_2$</td>
<td>0.020</td>
<td>0.029</td>
<td>m⁴</td>
</tr>
<tr>
<td></td>
<td>$I_3$</td>
<td>0.020</td>
<td>0.058</td>
<td>m⁴</td>
</tr>
</tbody>
</table>

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures** mode.
2. Click the **Selection** button.
3. Right-click the volume representing the building. Select the **Decompose into surfaces** option from the appearing menu.
4. Delete the top surface by selecting it and pressing **<Delete>**.
5. Select the volume representing the building. Click the visualisation toggle in the **Selection explorer** to hide the volume.
6. Right-click the bottom surface of the building. Select the **Create plate** option from the appearing menu.
7. Assign plates to the two vertical basement surfaces that are inside the model. Delete the remaining two vertical surfaces at the model boundaries.
   - Multiple entities can be selected by holding the **<Ctrl>** key pressed while clicking on the entities.
   - A feature can be assigned to multiple similar objects the same way as to a single selection.
8. Click the **Materials** button to open the material data base, then set the **Set type to Plates**.

9. Create data sets for the basement floor and for the basement walls according to Table 2 (on page 24).

10. Drag and drop the data sets to the basement floor and the basement walls accordingly. It may be needed to move the **Material sets** window by clicking at its header and dragging it.

11. Click the **OK** button to close the **Material sets** window.

12. Right-click the bottom of the surface of the building volume and select the **Create surface load** option from the appearing menu. The actual value of the load can be assigned in the **Structures** mode as well as when the calculation phases will be defined (Phase definition mode). In this example, the value will be assigned in the Phase definition modes.

13. Click the **Create line** button in the side toolbar.

14. Select the **Create line load** option from the additional tools displayed.

15. Click the command input area, type 0 18 0 18 0 18 0 0 and press **Enter**. Line loads will now be defined on the basement walls. The defined values are the coordinates of the three points of the lines. Click the right mouse button to stop drawing line loads.

16. Click the **Create line** button in the side toolbar.

17. Select the **Create beam** option from the additional tools displayed.

18. Click on (6 6 0) to create the first point of a vertical beam. Keep **Shift** pressed and move the mouse cursor to (6 6 -2). Note that while the **Shift** key is pressed the cursor will move only vertically. As it can be seen in the cursor position indicator, the z coordinate changes, while x and y coordinates will remain the same. Click on (6 6 -2) to define the second point of the beam. To stop drawing click the right mouse button.

19. Create horizontal beams from (0 6 -2) to (18 6 -2) and from (6 0 -2) to (6 18 -2).
Note: By default, the cursor is located at z=0. To move in the vertical direction, keep the <Shift> key while moving the mouse.

20. Click the Materials button to open the material data base and set the Set type to Beams.

21. Create data sets for the horizontal beams according to Table 3 (on page 25).

22. Assign the data set to the corresponding beam elements by drag and drop.

23. Click the Create load button in the side toolbar.

24. Select the Create point load option from the additional tools displayed. Click at (6 6 0) to add a point load at the top of the vertical beam.

2.2.4 Generate the mesh

To generate the mesh, follow these steps:

1. Proceed to the Mesh mode by clicking the corresponding tab.
2. Click the Generate mesh button. Keep the Element distribution as Coarse.
3. Click the View mesh button and inspect the generated mesh.
4. Click on the Close tab to close the Output program and go back to the Mesh mode of the Input program.

As the geometry has changed, all calculation phases have to be redefined.

2.2.5 Define and perform the calculation

Proceed to the Staged construction mode.

Initial phase

1. Click the Staged construction tab to proceed with the definition of calculation phases.
2. As in the previous example, the K₀ procedure will be used to generate the initial conditions.
3. All the structural elements should be inactive in the Initial Phase.
4. No excavation is performed in the Initial phase. So, the basement volume should be active and the material assigned to it should be Lacustrine clay.

Phase 1 to 3: Construction stages

Instead of constructing the building in one calculation stage, separate calculation phases will be used. In Phase 1, the construction of the walls and the excavation is modelled. In Phase 2, the construction of the floor and beams is modelled. The activation of the loads is modelled in the last phase (Phase 3).
1. The calculation type for the phases representing the construction stages is set by default to Plastic.
2. In the Phases window rename Phase_1 to Excavation.
3. In the Staged construction mode deactivate the soil volume located over the foundation by selecting it and by clicking on the checkbox in front of it in the Selection explorer.
4. In the Model explorer click the checkbox in front of the plates corresponding to the basement walls to activate them.
5. In the Phases explorer click the Add phase button. A new phase (Phase_2) is added. Double-click Phase_2.
6. Rename the phase by defining its ID as Construction. Keep the default settings of the phase and close the Phases window.
7. In the Model explorer click the checkbox in front of the plate corresponding to the basement floor to activate it.
8. In the Model explorer click the checkbox in front of the beams to activate all the beams in the project.
9. Add a new phase following the Construction phase. Rename it to Loading.
10. In the Model explorer click the checkbox in front of the Surface loads to activate the surface load on the basement floor. Set the value of the z-component of the load to -5.3. This indicates a load of 5.3 kN/m², acting in the negative z-direction.
11. In the Model explorer, click the checkbox in front of Line loads to activate the line loads on the basement walls. Set the value of the z-component of each load to -385. This indicates a load of 385 kN/m, acting in the negative z-direction.
12. In the Model explorer click the checkbox in front of Point loads to activate the point load on the basement column. Set the value of the z-component of the load to -11650. This indicates a load of 11650 kN, acting in the negative z-direction.

Execute the calculation

1. Click the Preview phase button to check the settings for each phase.
2. Click the Calculate button to calculate the project. Ignore the warning that no nodes and stress points have been selected for curves.
3. Click the Save button to save the project after the calculation.

2.2.6 View the calculation results

1. Select Construction phase in the Phases explorer.
2. Click the View calculation results button to open the Output program. The deformed mesh at the end of this phase is shown.
3. Select the last phase in the Displayed step drop-down menu to switch to the results at the end of the last phase.
4. In order to evaluate stresses and deformations inside the geometry, select the Vertical cross section tool.
A top view of the geometry is presented and the **Cross section points** window appears. As the largest displacements appear under the column, a cross section here is most interesting.

5. Enter (0.0 6.0) and (75.0 6.0) as the coordinates of the first point (A) and the second point (A’) respectively in the **Cross section points** window.

6. Click **OK**.
   A vertical cross section is presented. The cross section can be rotated in the same way as a regular 3D view of the geometry.

7. Select the menu **Deformations > Total displacements > u_z**
   The maximum and minimum values of the vertical displacements are shown in the caption. If the title is not visible, select this option from the **View** menu.

![Figure 16: Cross section showing the total vertical displacement](image)

8. Press `<CTRL+>` and `<CTRL->` to move the cross section.

9. Return to the three-dimensional of the geometry by selecting this window from the list in the **Window** menu.

10. Double-click the floor.
    A separate window will appear showing the displacements of the floor. To look at the bending moments in the floor, select the menu **Forces > M_11**.

11. Click the **Shadings** button.
    The plot will be displayed.
12. To view the bending moments in tabulated form, select **Tools > Table**. A new window is opened in which a table is presented, showing the values of bending moments in each node of the floor.

### 2.3 Case C: Pile-Raft foundation

As the displacements of the raft foundation are rather high, embedded beams will be used to decrease these displacements. These embedded beams represent bored piles with a length of 20 m and a diameter of 1.5 m.

**Objectives**

- Using embedded beams.
- Defining material data set for embedded beams.
- Creating multiple copies of entities.

#### 2.3.1 Create a new project
The geometry used in this exercise is the same as the previous one, except for the pile foundation. It is not necessary to create a new model; you can start from the previous model, store it under a different name and modify it. To perform this, follow these steps:

1. Start PLAXIS 3D by double clicking the icon of the Input program.
   The Quick select dialog box appears.
2. Select the project of Case B.
   The project opens.
3. Select the menu File > Save project as to save the project under a different name (e.g. Tutorial 1c).

### 2.3.2 Define the structural elements: Foundation piles

The foundation consists of piles. These will be modelled as embedded beams. A new material is needed for the piles. The material properties are as follows:

**Table 4: Material properties of embedded beam**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Pile foundation</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>$3 \times 10^7$</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Unit weight</td>
<td>$\gamma$</td>
<td>6.0</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Beam type</td>
<td>-</td>
<td>Predefined</td>
<td>-</td>
</tr>
<tr>
<td>Predefined beam type</td>
<td>-</td>
<td>Massive circular beam</td>
<td>-</td>
</tr>
<tr>
<td>Diameter</td>
<td>Diameter</td>
<td>1.5</td>
<td>m</td>
</tr>
<tr>
<td>Axial skin resistance</td>
<td>Type</td>
<td>Linear</td>
<td>-</td>
</tr>
<tr>
<td>Maximum traction allowed at the top of the embedded beam</td>
<td>$T_{\text{skin, start, max}}$</td>
<td>200</td>
<td>kN/m</td>
</tr>
<tr>
<td>Maximum traction allowed at the bottom of the embedded beam</td>
<td>$T_{\text{skin, end, max}}$</td>
<td>500</td>
<td>kN/m</td>
</tr>
<tr>
<td>Base resistance</td>
<td>$F_{\text{max}}$</td>
<td>$1 \times 10^4$</td>
<td>kN</td>
</tr>
</tbody>
</table>

To model the foundation piles:

1. Click the Structures tab to proceed with the input of structural elements in the Structures mode.
2. Click the Create line button at the side tool bar and select the Create embedded beam from the additional tools that appear.
3. Define a pile from (6 6 -2) to (6 6 -22).
4. Click the *Materials* button to open the material data base and set the *Set type* to *Embedded beams*.

5. Create a data set for the embedded beam according to Table 4 (on page 31). The value for the cross section area A and the moments of inertia \( I_2 \) and \( I_3 \) are automatically calculated from the diameter of the massive circular pile. Confirm the input by clicking *OK*.

6. Drag and drop the *Embedded beam* data to the embedded beam in the drawing area. The embedded beam will change colour to indicate that the material set has been assigned successfully.

7. Click the *OK* button to close the *Material sets* window.

**Note:**

A material set can also be assigned to an embedded beam by right-clicking it either in the drawing area or in the *Selection explorer* and the *Model explorer* and selecting the material from the *Set material* option in the displayed menu.

8. Click the *Select* button and select the embedded beam.

9. Click the *Create array* button.

10. In the *Create array* window, select the *2D, in xy plane* option for shape.

11. Keep the number of columns as 2. Set the distance between the columns to \( x=12 \) and \( y=0 \).

12. Keep the number of rows as 2. Set the distance between the rows to \( x=0 \) and \( y=12 \).

![Create array window](image)

*Figure 18: Create array window*

13. Press *OK* to create the array. A total of \( 2 \times 2 = 4 \) piles will be created.

2.3.3 Generate the mesh
As the geometry model is complete now, the mesh can be generated.

To generate the mesh, follow these steps:

1. Proceed to the **Mesh** mode.
2. Click the **Generate mesh** button in the side toolbar. Keep the **Element distribution** as **Coarse**
3. Click the **View mesh** button to view the mesh.
4. Click the eye button in front of the **Soil** subtree in the **Model explorer** to hide the soil. The embedded beams can be seen now:

![Figure 19: Partial geometry of the model in the Output](image)

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

### 2.3.4 Define and perform the calculation

After generation of the mesh, all construction stages must be redefined. Even though in practice the piles will be constructed in another construction stage than construction of the walls, for simplicity both actions will be done in the same construction stage in this tutorial. To redefine all construction stages, follow these steps:

1. Switch to the **Staged construction** mode.
2. Check if the **K0 procedure** is selected as **Calculation type** for the initial phase. Make sure that all the structural elements are inactive and all soil volumes are active.
3. In the **Phases explorer** select the **Excavation** phase.
4. Make sure that the basement soil is excavated and the basement walls are active.
5. Activate all the embedded beams.
6. In the **Phases explorer** select the **Construction** phase. Make sure that all the structural elements are active.
7. In the **Phases explorer** select the **Loading** phase. Make sure that all the structural elements and loads are active.
8. Click the Calculate button 🗝️ to calculate the project.
9. Click the Save button 🗂️ to save the project after the calculation.

2.3.5 View the calculation results

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

To view the results, follow these steps:

1. Select the Loading phase and view the calculation results.
2. Double-click the basement floor. Select the menu Forces > M_11.
   The results are shown:

   ![Figure 20: Bending moments in the basement floor](image)

3. Adjust the legend by changing scaling settings into:
   a. Scaling: manual
   b. Minimum value: -450
   c. Maximum value: 550
   d. Number of intervals: 18
4. Select the view corresponding to the deformed mesh in the **Window** menu.

5. Click the **Toggle visibility** button in the side toolbar.

6. To view the embedded beams press **<Shift>** and keep it pressed while clicking on the soil volume in order to hide it.

7. Click the **Select structures** button. To view all the embedded beams, press **<Ctrl>+<Shift>** and double click on one of the piles.

8. Select the menu **Forces** > **N** to view the axial loads in the embedded beams.

The plot is shown:

![Figure 21: Resulting axial forces (N) in the embedded beams](image)

*Figure 21: Resulting axial forces (N) in the embedded beams*
This tutorial describes the construction of an excavation pit in soft clay and sand layers. The pit is a relatively small excavation of 12 by 20 m, excavated to a depth of 6.5 m below the surface. Struts, walings and ground anchors are used to prevent the pit from collapsing. After the full excavation, an additional surface load is added on one side of the pit.

**Objectives**

- Using the Hardening Soil model
- Modelling of ground anchors
- Using interface features
- Defining over-consolidation ratio (OCR)
- Prestressing a ground anchor
- Changing water conditions
- Selection of stress points to generate stress/strain curves
- Viewing plastic points

**Geometry**

The proposed geometry for this exercise is 80 m wide and 50 m long. The excavation pit is placed in the center of the geometry.
Excavation in sand
Create a new project

The figure below shows a cross section of the excavation pit with the soil layers. The clay layer is considered to be impermeable.

3.1 Create a new project

To create the geometry model, follow these steps:

1. Start a new project.
2. Enter an appropriate title for the project.
3. Define the limits for the soil contour as
Excavation in sand
Define the soil stratigraphy

\[
\begin{align*}
\text{a. } x_{\min} &= 0.0 \text{ and } x_{\max} = 80.0, \\
\text{b. } y_{\min} &= 0.0 \text{ and } y_{\max} = 50.0.
\end{align*}
\]

3.2 Define the soil stratigraphy

In order to define the soil layers, a borehole needs to be added and material properties must be assigned. As all soil layers are horizontal, only a single borehole is needed.

1. Click the Create borehole button and create a borehole at (0 0 0). The Modify soil layers window pops up.
2. Add 4 layers with bottom levels at -1, -9.5, -11, -20.
3. Set the Head in the borehole column to -4 m.

3.3 Create and assign the material data sets

A number of materials need to be defined for the different soil layers. The material properties are shown in the table below Linear Elastic model.

Table 5: Material properties for the soil layers

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Fill</th>
<th>Sand</th>
<th>Soft Clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>General</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td></td>
<td></td>
<td></td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Hardening Soil</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Hardening Soil</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Hardening Soil</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td></td>
<td></td>
<td></td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Drained</td>
<td></td>
<td></td>
<td>Undrained A</td>
<td></td>
</tr>
<tr>
<td>Unit weight above phreatic level</td>
<td>( \gamma_{\text{unsat}} )</td>
<td>16.0</td>
<td>17.0</td>
<td>16.0</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Unit weight below phreatic level</td>
<td>( \gamma_{\text{sat}} )</td>
<td>20.0</td>
<td>20.0</td>
<td>17.0</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Parameters</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Secant stiffness for CD triaxial test</td>
<td>( E_{50}^{\text{ref}} )</td>
<td>( 2.2\cdot10^4 )</td>
<td>( 4.3\cdot10^4 )</td>
<td>( 2.0\cdot10^3 )</td>
<td>kN/m²</td>
</tr>
</tbody>
</table>
### Excavation in sand

Create and assign the material data sets

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Fill</th>
<th>Sand</th>
<th>Soft Clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tangent oedometer stiffness</td>
<td>$E_{oed}^{\text{ref}}$</td>
<td>2.2·10^4</td>
<td>2.2·10^4</td>
<td>2.0·10^3</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Unloading/reloading stiffness</td>
<td>$E_{ur}^{\text{ref}}$</td>
<td>6.6·10^4</td>
<td>1.29·10^5</td>
<td>1.0·10^4</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Power for stress level dependency of stiffness</td>
<td>m</td>
<td>0.5</td>
<td>0.5</td>
<td>1.0</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>$c_{\text{ref}}'$</td>
<td>1</td>
<td>1</td>
<td>5</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\varphi'$</td>
<td>30.0</td>
<td>34.0</td>
<td>25</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>0.0</td>
<td>4.0</td>
<td>0.0</td>
<td>°</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$\nu_{ur}'$</td>
<td>0.2</td>
<td>0.2</td>
<td>0.2</td>
<td>-</td>
</tr>
</tbody>
</table>

#### Drainage type

| Interface strength                             | -         | Manual | Manual | Manual | -      |
| Interface reduction factor                     | $R_{\text{inter}}$ | 0.65   | 0.7    | 0.5     | -      |

#### Parameters

| $K_0$ determination                          | -         | Automatic | Automatic | Automatic | -      |
| Lateral earth pressure coefficient           | $K_0$    | 0.5000    | 0.4408    | 0.7411    | -      |
| Over-consolidation ratio                     | OCR       | 1.0       | 1.0       | 1.5       | -      |
| Pre-overburden pressure                      | POP       | 0.0       | 0.0       | 0.0       | -      |

1. Click the **Materials** button 🔄 in the side toolbar.
   The **Material sets** window pops up.
2. Create a new data set under **Soil and interfaces** set type.
3. Identify the new data set as **Fill**.
4. From the Material model drop-down menu, select Hardening Soil model. In contrast with the Mohr-Coulomb model, the Hardening Soil model takes into account the difference in stiffness between virgin-loading and unloading-reloading. For a detailed description of the Hardening Soil model, see the Material Models Manual.

5. Define the saturated and unsaturated unit weights according to Table 5 (on page 38).

6. In the Parameters tabsheet, enter values for $E_{50 \text{ ref}}$, $E_{\text{oed} \text{ ref}}$, $E_{\text{ur} \text{ ref}}$, $m$, $c'_{\text{ref}}$, $\varphi'_{\text{ref}}$, $\psi$ and $\nu'_{\text{ur}}$. Note that Poisson's ratio is an advanced parameter.

7. As no consolidation will be considered in this exercise, the permeability of the soil will not influence the results. Therefore, the default values can be kept in the Flow parameters tabsheet.

8. In the Interfaces tabsheet, select Manual in the Strength box and enter a value of 0.65 for the parameter $R_{\text{inter}}$. This parameter relates the strength of the interfaces to the strength of the soil, according to the equations:

\[ c_i = R_{\text{inter}} c_{\text{soil}} \text{ and } \tan\varphi_i = R_{\text{inter}} \tan\varphi_{\text{soil}} \]

Hence, using the entered $R_{\text{inter}}$-value gives a reduced interface friction and interface cohesion (adhesion) compared to the friction angle and the cohesion in the adjacent soil.

Note:
- When the Rigid option is selected in the Strength drop-down, the interface has the same strength properties as the soil ($R_{\text{inter}} = 1.0$).
- Note that a value of $R_{\text{inter}} < 1.0$, reduces the strength as well as the stiffness of the interface (for more information see the Reference Manual).

9. In the Initial tabsheet, define the OCR-value according to Table 5 (on page 38).

10. Click OK to close the window.

11. After closing the Material sets window, click the OK button to close the Modify soil layers window.

12. In the Soil mode right-click the upper soil layer. Select Set material > Fill.

13. In the same way assign the Soft Clay material to the soil layer between $y = -9.5 \text{ m}$ and $y = -11.0 \text{ m}$.

14. Assign the Sand material to the remaining two soil layers.

Note: The Tension cut-off option is activated by default at a value of 0 kN/m$^2$. This option is found in the Advanced options on the Parameters tabsheet of the Soil window. Here the Tension cut-off value can be changed or the option can be deactivated entirely.

### 3.4 Define the structural elements

The creation of walings and struts, ground anchors, sheet pile walls and surface loads is described below.

#### 3.4.1 Walings and Struts

The material properties for the structural elements are shown in the table below. These are needed for defining the material in a later step.
**Excavation in sand**

Define the structural elements

Table 6: Material properties of waling and strut

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Strut</th>
<th>Waling</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material type</td>
<td>Type</td>
<td>Elastic</td>
<td>Elastic</td>
<td>-</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>2.1·10^8</td>
<td>2.1·10^8</td>
<td>kN/m^2</td>
</tr>
<tr>
<td>Unit weight</td>
<td>γ</td>
<td>78.5</td>
<td>78.5</td>
<td>kN/m^3</td>
</tr>
<tr>
<td>Cross section area</td>
<td>A</td>
<td>0.007367</td>
<td>0.008682</td>
<td>m^2</td>
</tr>
<tr>
<td>Moment of Inertia</td>
<td>I_2</td>
<td>5.073·10^{-5}</td>
<td>3.66·10^{-4}</td>
<td>m^4</td>
</tr>
<tr>
<td></td>
<td>I_3</td>
<td>5.073·10^{-5}</td>
<td>1.045·10^{-4}</td>
<td>m^4</td>
</tr>
</tbody>
</table>

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures** mode.
2. Create a surface between (30 20 0), (30 32 0), (50 32 0) and (50 20 0).
3. Extrude the surface to z = -1, z = -6.5 and z = -11.
4. Right-click on the deepest created volume (between z = 0 and z = -11) and select **Decompose into surfaces**.
5. Delete the top surfaces (2 surfaces). An extra surface is created as the volume is decomposed.
6. Hide the excavation volumes (do not delete). The eye button in the **Model explorer** and the **Selection explorer** trees can be used to hide parts of the model and simplify the view. A hidden project entity is indicated by a closed eye.
7. Click the **Create structure** button 🏗️.
8. Create beams (walings) around the excavation circumference at level z=-1 m. Press **<Shift>** and keep it pressed while moving the mouse cursor in the -z-direction. Stop moving the mouse as the z-coordinate of the mouse cursor is -1 in the cursor position indicator. Note that as you release **<Shift>**, the z-coordinate of the cursor location does not change. This is an indication that you can draw only on the xy-plane located at z = -1.
9. Click on (30 20 -1), (30 32 -1), (50 32 -1), (50 20 -1), (30 20 -1) to draw the walings. Click on the right mouse button to stop drawing walings.
10. Create a beam (strut) between (35 20 -1) and (35 32 -1). Press **<Esc>** to end defining the strut.
11. Create data sets for the walings and struts according to Table 6 (on page 41) and assign the materials accordingly.
12. Copy the strut into a total of three struts at x = 35 (existing), x = 40, and x = 45.

### 3.4.2 Ground anchors

The material properties for the structural elements are shown in the table below. These are needed for defining the material in a later step.
Excavation in sand

Define the structural elements

### Table 7: Material properties of the node-to-node anchors

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Node-to-node anchor</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material type</td>
<td>Type</td>
<td>Elastic</td>
<td>-</td>
</tr>
<tr>
<td>Axial stiffness</td>
<td>EA</td>
<td>6.5·10⁵</td>
<td>kN</td>
</tr>
</tbody>
</table>

### Table 8: Material properties of the embedded beams (grout body)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Grout</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>3·10⁷</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Unit weight</td>
<td>γ</td>
<td>24</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Beam type</td>
<td>-</td>
<td>Predefined</td>
<td>-</td>
</tr>
<tr>
<td>Predefined beam type</td>
<td>-</td>
<td>Massive circular beam</td>
<td>-</td>
</tr>
<tr>
<td>Diameter</td>
<td>Diameter</td>
<td>0.14</td>
<td>m</td>
</tr>
<tr>
<td>Axial skin resistance</td>
<td>Type</td>
<td>Linear</td>
<td>-</td>
</tr>
<tr>
<td>Skin resistance at the top of the embedded beam</td>
<td>T_{skin,start,max}</td>
<td>200</td>
<td>kN/m</td>
</tr>
<tr>
<td>Skin resistance at the bottom of the embedded beam</td>
<td>T_{skin,end,max}</td>
<td>0.0</td>
<td>kN/m</td>
</tr>
<tr>
<td>Base resistance</td>
<td>F_{max}</td>
<td>0.0</td>
<td>kN</td>
</tr>
</tbody>
</table>

In PLAXIS 3D ground anchors can be modelled using the Node-to-node anchor and the Embedded beam options as described in the following:

1. First the ungrouted part of the anchor is created using the Node-to-node anchor feature. Start creating a node-to-node anchor by selecting the corresponding button in the options displayed as you click on the Create structure button.
2. Click on the command line and type \(30 \ 24 \ -1 \ 21 \ 24 \ -7\). Press <Enter> and <Esc> to create the ungrouted part of the first ground anchor.
3. Create a node-to-node anchor between the points (50 24 -1) and (59 24 -7).
4. The grouted part of the anchor is created using the Embedded beam option. Create embedded beams between (21 24 -7) and (18 24 -9) and between (59 24 -7) and (62 24 -9). Set the Behaviour to Grout body (for more information see the Reference Manual).
5. Create a data set for the embedded beam and a data set for the node-to-node anchor according to Table 7 (on page 42) and Table 8 (on page 42) respectively. Assign the data sets to the node-to-node anchors and to the embedded beams.
**Excavation in sand**

Define the structural elements

---

**Note:** The colour indicating the material set assigned to the entities in project can be changed by clicking on the Colour box of the selected material set and selecting a colour from the **Colour** part of the window.

6. The remaining grouted anchors will be created by copying the defined grouted anchor. Click on the Select button and click on all the elements composing both of the ground anchors keeping <Ctrl> pressed.

7. Use the Create array function to copy both ground anchors (2 embedded beams + 2 node-to-node anchors) into a total of 4 complete ground anchors located at \( y = 24 \) and \( y = 28 \) by selection the 1D, in \( y \) direction option in the Shape drop-down menu and define the Distance between columns as 4 m.

8. Multi-select all parts of the ground anchors (8 entities in total). While all parts are selected and <Ctrl> is pressed, right-click and select **Group**.

9. In the **Model explorer** tree, expand the **Groups** subtree by clicking on the (+) in front of the groups.

10. Click the Group_1 and rename it to **GroundAnchors**.

---

**Note:** The name of the entities in the project should not contain any space or special character except _ (underscore).

---

### 3.4.3 Pile sheet walls and loads

The material properties for the structural elements are shown in the table below. These are needed for defining the material in a later step.

**Table 9: Material properties of pile sheet walls**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Sheet pile wall</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of behaviour</td>
<td>Type</td>
<td>Elastic, non-isotropic</td>
<td>-</td>
</tr>
<tr>
<td>Thickness</td>
<td>d</td>
<td>0.379</td>
<td>m</td>
</tr>
<tr>
<td>Weight</td>
<td>( \gamma )</td>
<td>2.55</td>
<td>kN/m³</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>( E_1 )</td>
<td>1.46·10⁷</td>
<td>kN/m²</td>
</tr>
<tr>
<td></td>
<td>( E_2 )</td>
<td>7.3·10⁵</td>
<td>kN/m²</td>
</tr>
<tr>
<td></td>
<td>( \nu_{12} )</td>
<td>0.0</td>
<td>-</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>( G_{12} )</td>
<td>7.3·10⁵</td>
<td>kN/m²</td>
</tr>
<tr>
<td></td>
<td>( G_{13} )</td>
<td>1.27·10⁶</td>
<td>kN/m²</td>
</tr>
<tr>
<td></td>
<td>( G_{23} )</td>
<td>3.82·10⁵</td>
<td>kN/m²</td>
</tr>
</tbody>
</table>

To define the sheet pile walls and the corresponding interfaces, follow these steps:
1. **Select all four vertical surfaces created as the volume was decomposed.** Keeping <Ctrl> pressed, right-click and select the **Create plate** option from the appearing menu.

2. **Create a data set for the sheet pile walls (plates) according to Table 9 (on page 43).** Assign the data sets to the four walls.

3. **As all the surfaces are selected, assign both positive and negative interfaces to them using the options in the right mouse button menu.**

   **Note:** The term 'positive' or 'negative' for interfaces has no physical meaning. It only enables distinguishing between interfaces at each side of a surface.

4. **In the Model explorer tree expand the Surfaces subtree, set AxisFunction to Manual and set Axis1 to -1.** Do this for all the pile wall surfaces.

   **Note:**
   - Non-isotropic (different stiffness in two directions) sheet pile walls are defined. The local axis should point in the correct direction (which defines which is the 'stiff' or the 'soft' direction). As the vertical direction is generally the stiffest direction in sheet pile walls, local axis 1 shall point in the z-direction.
   - The first local axis is indicated by a red arrow, the second local axis is indicated by a green arrow and the third axis is indicated by a blue arrow. More information related to the local axes of plates is given in the Reference Manual.

5. **Create a surface load defined by the points: (34 19 0), (41 19 0), (41 12 0), (34 12 0).** The geometry is now completely defined.

### 3.5 Generate the mesh

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

2. **Select the surface representing the excavation. In the Selection explorer set the value of Coarseness factor to 0.25.**

3. **Set the element distribution to Coarse.** Uncheck the box for Enhanced mesh refinements. Click the Generate mesh button to generate the mesh.

   **Note:** The Enhanced mesh refinements are automatically used in mesh generation. More information is available in the Reference Manual.

4. **Click the View mesh button to view the mesh.** Hide the soil in the model to view the embedded beams.

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

### 3.6 Define the calculation
Excavation in sand
Define the calculation

The calculation consists of 6 phases. The initial phase consists of the generation of the initial stresses using the K0 procedure. The next phase consists of the installation of the sheet piles and a first excavation. Then the walings and struts will be installed. In phase 3, the ground anchors will be activated and prestressed. Further excavation will be performed in the phase after that. The last phase will be the application of the additional load next to the pit.

1. Click on the Staged construction tab to proceed with definition of the calculation phases.
2. The initial phase has already been introduced. Keep its calculation type as K0 procedure. Make sure all the soil volumes are active and all the structural elements are inactive.
3. Add a new phase (Phase_1). The default values of the parameters will be used for this calculation phase.
4. Deactivate the first excavation volume (from z=0 to z=-1).
5. In the Model explorer, activate all plates and interfaces by clicking on the checkbox in front of them. The active elements in the project are indicated by a green check mark in the Model explorer.
6. Add a new phase (Phase_2). The default values of the parameters will be used for this calculation phase.
7. In the Model explorer activate all the beams.
8. Add a new phase (Phase_3). The default values of the parameters will be used for this calculation phase.
9. In the Model explorer activate the GroundAnchors group.
10. Select one of the node-to-node anchors.
11. In the Selection explorer expand the node-to-node anchor features.
12. Click on the Adjust prestress checkbox. Enter a prestress force of 200 kN.

Figure 24: Node-to-node anchor in the Selection explorer

13. Do the same for all the other node-to-node anchors.
14. Add another phase (Phase_4). The default values of the parameters will be used for this calculation phase.
15. Select the soil volume to be excavated in this phase (between z=-1 and z=-6.5).
16. In the Selection explorer under WaterConditions feature, click on the Conditions and select the Dry option from the drop-down menu.
17. Deactivate the volume to be excavated (between z = -1 and z = -6.5).
18. Hide the soil and the plates around the excavation.
19. Select the soil volume below the excavation (between z = -6.5 and z = -9.5).
20. In Selection explorer under WaterConditions feature, click Conditions and select Head from the drop-down menu. Enter $z_{ref} = -6.5$ m.
21. Select the soft clay volume below the excavation.
22. Set the water conditions to Interpolate.
23. Preview this calculation phase.
24. Click the Vertical cross section button in the Preview window and define the cross section by drawing a line across the excavation.
   
   Note: Hold <Shift> when drawing to get a straight line.
25. Select the $p_{steady}$ option from the Stresses menu.
26. Display the contour lines for steady pore pressure distribution. Make sure that the Legend > View option is checked. The steady state pore pressure distribution is displayed in the figure below. Scroll the wheel button of the mouse to zoom in or out to get a better view.

Figure 25: Water conditions in the Selection explorer

Figure 26: Preview of the steady state pore pressures in Phase_4 in a cross section
Excavation in sand

Results

- Scaling: manual
- Maximum value: 0
- Number of intervals: 18

28. Click on the Close button to return to the Input program.

29. Add another phase (Phase_5). The default values of the parameters will be used for this calculation phase.

30. Activate the surface load and set $\sigma_z = -20$ kN/m$^2$.

3.6.1 Execute the calculation

Before starting the calculation process, some stress points next to the excavation pit and loading are selected to plot a stress strain curve later on.

1. Click the Select points for curves button.
   The model and Select points window will be displayed in the Output program.
2. Define (37.5 19 -1.5) as Point-of-interest coordinates.

   Note: The visualization settings can be changed from the menu. For more information refer to Reference Manual.

3. Click the Search closest button.
   The number of the closest node and stress point will be displayed.
4. Click the checkbox in front of the stress point to be selected.
   The selected stress point will be shown in the list.
5. Select also stress points near the coordinates (37.5 19 -5), (37.5 19 -6) and (37.5 19 -7) and close the Select points window.
6. Click the Update button to close the Output program.
7. Start the calculation process.
8. Save the project when the calculation is finished.

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 plot curves of structural forces, nodes can only be selected after the calculation.
- Nodes or stress points can be selected by just clicking them. When moving the mouse, the exact coordinates of the position are given in the cursor location indicator bar at the bottom of the window.

3.7 Results
After the calculations, the results of the excavation can be viewed by selecting a calculation phase from the Phases tree and pressing the View calculation results button.

1. Select the final calculation phase (Phase_5) and click the View calculation results button.

   The Output program will open and will show the deformed mesh at the end of the last phase.

2. The stresses, deformations and three-dimensional geometry can be viewed by selecting the desired output from the corresponding menus. For example, choose the menu Stresses > Plastic points to investigate the plastic points in the model.

3. In the Plastic points window, select all the options except the Elastic points and the Show only inaccurate points options.

![Figure 27: Plastic points window](image1)

The figure below shows the plastic points generated in the model at the end of the final calculation phase.

![Figure 28: Plastic points at the end of the final phase](image2)
The graph will now show the major principal strain against the major principal stress. Both values are zero at the beginning of the initial conditions. After generation of the initial conditions, the principal strain is still zero whereas the principal stress is not zero anymore. To plot the curves of all selected stress points in one graph, follow these steps:

1. Right-click and select **Add curve > From current project**.
2. Generate curves for the three remaining stress nodes in the same way.

![Figure 29: Curve generation window](image)

The graph will now show the stress-strain curves of all four stress points.

![Figure 30: Stress - Strain curve](image)
To see information about the markers, make sure the menu item **View > Value indication** is checked and hold the mouse on a marker for a while. Information about the coordinates in the graph, the number of the point in the graph, the number of the phase and the number of the step is given. Especially the lower stress points show a considerable increase in the stress when the load is applied in the last phase.

**Note:**

- To re-enter the **Curve generation** window (in the case of a mistake, a desired regeneration or a modification), select the menu item **Format > Curve settings**. As a result the **Curves settings** window appears, on which the **Regenerate** button should be clicked.
- The menu item **Format > Chart settings** menu may be used to modify the settings of the chart.

To create a stress path plot for stress node (37.5 19 -1.5) follow these steps:

1. Create a new chart.
2. In the **Curves generation** window, select node (37.5 19 -1.5) from the drop-down menu of the x-axis of the graph and $\sigma'_{yy}$ under **Cartesian effective stresses**.
3. Select node (37.5 19 -1.5) from the drop-down menu of the y-axis of the graph. Select $\sigma'_{zz}$ under **Cartesian effective stresses**.
4. Click **OK** to confirm the input.

![Figure 31: Vertical effective stress ($\sigma'_{yy}$) versus horizontal effective stress ($\sigma'_{yy}$) at stress node located near (37.5 19 -1.5)](image)
In this tutorial a suction pile in an offshore foundation will be considered. A suction pile is a hollow steel pile with a large diameter and a closed top, which is installed in the seabed by pumping water from the inside. The resulting pressure difference between the outside and the inside is the driving force behind this installation.

This exercise will investigate the displacement of the suction pile under working load conditions. Four different angles of the working load will be considered. The installation process itself will not be modelled.

**Objectives**
- Using shape designer
- Using rigid body objects
- Undrained effective stress analysis with undrained strength parameters
- Undrained shear strength increasing with depth
- Copying material data sets
- Changing settings in Output
- Helper objects for local mesh refinements

**Geometry**
In this exercise, the length of the suction pile is 10 m and the diameter is 5.0 m. An anchor chain is attached on the side of the pile, 7 m from the top. The water depth at the considered location is 50 m above the ground. Only one symmetric half will be modelled.

The soil consists of clay but because of the short duration of the load, an undrained stress analysis with undrained strength parameters will be performed. The geometry for the problem is shown below.

![Figure 32: Geometry of the suction pile](image-url)
4.1 Create a new project

An area of 30 m wide and 60 m long with half of the suction pile will be modelled in this example. With these dimensions the model is sufficiently large to avoid any influence from the model boundaries. To define the geometry for this exercise, follow these steps:

1. Start the Input program and select New project from the Create/Open project dialog box.
2. Enter an appropriate title for the project.
3. Keep the standard units and set the model dimensions to
   a. \( x_{\text{min}} = -30.0 \) and \( x_{\text{max}} = 30.0 \).
   b. \( y_{\text{min}} = 0.0 \) and \( y_{\text{max}} = 30.0 \).
4. Click OK.

4.2 Define the soil stratigraphy

In the current example only one horizontal soil layer is present. A single borehole is sufficient to define it.

1. Click the Create borehole button and create a borehole at (0 0 0).
   The Modify soil layers window pops up.
2. In the Modify soil layers window add a soil layer with top boundary at \( z = 0 \text{ m} \) and bottom boundary at \( z = -30 \text{ m} \).
3. The water depth at the considered location is 50 m. This would imply that the head is set to 50 m, but the results will be equal as long as the whole geometry is below the water level (the water above the ground is a total load and not an effective load). Hence, a head of 1.0 m is sufficient. Set the Head to 1.0 m.

4.3 Create and assign the material data sets

The material properties for the data sets are shown in the table below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Clay</th>
<th>Interface</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Mohr-Coulomb model</td>
<td>Mohr-Coulomb model</td>
<td>-</td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td>Undrained B</td>
<td>Undrained B</td>
<td>-</td>
</tr>
<tr>
<td>Soil weight</td>
<td>( \gamma_{\text{unsat}} )</td>
<td>20</td>
<td>( \gamma_{\text{sat}} )</td>
<td>20 kN/m³</td>
</tr>
</tbody>
</table>
### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Clay</th>
<th>Interface</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>E'</td>
<td>1000</td>
<td>1000</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>ν'</td>
<td>0.35</td>
<td>0.35</td>
<td>-</td>
</tr>
<tr>
<td>Undrained shear strength</td>
<td>s_u,ref</td>
<td>1.0</td>
<td>1.0</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Friction angle</td>
<td>φ_u</td>
<td>0.0</td>
<td>0.0</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>ψ</td>
<td>0.0</td>
<td>0.0</td>
<td>°</td>
</tr>
<tr>
<td>Increase in stiffness</td>
<td>E'_inc</td>
<td>1000</td>
<td>1000</td>
<td>kN/m²/m</td>
</tr>
<tr>
<td>Reference level</td>
<td>z_ref</td>
<td>0.0</td>
<td>0.0</td>
<td>m</td>
</tr>
<tr>
<td>Increase in undrained shear strength</td>
<td>s_u,inc</td>
<td>4.0</td>
<td>4.0</td>
<td>kN/m²/m</td>
</tr>
<tr>
<td>Reference level</td>
<td>z_ref</td>
<td>0.0</td>
<td>0.0</td>
<td>m</td>
</tr>
<tr>
<td>Tension cut-off</td>
<td>-</td>
<td>Inactive</td>
<td>Inactive</td>
<td>-</td>
</tr>
</tbody>
</table>

### Interfaces

| Interface strength            | -      | Manual | Rigid | -         |
| Interface strength reduction  | R_inter| 0.7    | 1.0   | -         |

### Initial

| K₀ determination              | -      | Manual | Manual | -         |
| Initial lateral earth pressure coeff. | K₀ₓ, K₀ᵧ | 0.5    | 0.5   | -         |

1. Click the **Materials** button.
2. Create the data sets given in Table 10 (on page 52). In the **Parameters** tabsheet deselect the **Tension cut-off** option in the advanced parameters for strength. In this exercise, the permeability of the soil will not influence the results. Instead of using effective strength properties, the cohesion parameter will be used in this example to model undrained shear strength. Advanced parameters can be entered after expanding the **Advanced** data tree in the **Parameters** tabsheet.

**Note:**
The **Interface** data set can be quickly created by copying the ‘Clay’ data set and changing the R_inter value.
3. Assign the 'Clay' material data set to the soil layer and close the Material sets window.

4.4 Define the structural elements

The suction pile is modelled in the Structures mode as half a cylindrical surface and this is then defined as a rigid body. Also, a helper object is defined for local mesh refinements.

4.4.1 Create a suction pile

In the Structures mode the suction pile as a rigid body will be defined. This is done by creating a polycurve at the soil surface and extruding it downward.

To create the suction pile, the Rigid body functionality is used. For more information on Rigid bodies, refer to the corresponding section in the Reference Manual.

1. Click the Structures tab to proceed to Structures mode.
2. Click the Create polycurve button in the side toolbar.
3. Click at (2.5 0 0) on the drawing area to define the insertion point.

Note: Select the menu Options > Visualization settings and set the Intervals to 2, while leaving the Spacing to 1 m. This allows to move the mouse with 0.5 m interval.

The Shape designer window pops up.

4. In the General tabsheet, the default option Free is valid for Shape.
5. The polycurve is drawn in the xy-plane (see the figure below). Hence the default orientation axes are valid for this example. For more information refer to "Definition of tunnel cross section geometry" in the Reference Manual.
Loading of a suction pile
Define the structural elements

6. In the **Segments** tabsheet, click on the **Add section** button in the top toolbar.
7. Set the **Segment type** to **Arc**, the **Relative start angle** to 90°, the **Radius** to 2.5 m and the **Segment angle** to 180°.
8. Click **OK** to add the polycurve to the geometry and to close the **Shape designer**.

9. Click on the created polycurve and select the **Extrude object** and set the z value to -10 m.

10. Right-click the created surface, and select **Create positive interface** to create a positive interface for the suction pile. Similarly create a negative interface for the surface.

11. Right-click the polycurve and select **Close** from the appearing menu. Further, right click the closed polycurve and select **Create surface**. This creates the top surface of the suction pile.

12. Right-click the top surface and create a negative interface.

13. Select the **Interfaces** in the **Model explorer**. In the **Selection explorer** tree, select **Custom** for the **Material mode** from the dropdown menu.

14. Select **Interface** for the **Material** from the dropdown menu.

15. Multi-select the top and the curved surface. Right-click on the selected surfaces and select the option **Create rigid body** from the appearing menu.
16. In the Selection explorer, set the reference point as \((2.5 \, 0 \, -7)\) for the rigid bodies by assigning the values to \(x_{\text{ref}}\), \(y_{\text{ref}}\) and \(z_{\text{ref}}\).

17. Set the Translation condition \(y\) to Displacement, the Rotation condition \(x\) and Rotation condition \(z\) to Rotation. Their corresponding values are \(u_y = \varphi_x = \varphi_z = 0\).
4.4.2 Create helper objects for local mesh refinements

A surface is created around the suction pile to achieve better mesh refinements. This is done by creating a circular surface around the suction pile using the *Shape designer*.

1. Click the *Create polycurve* button in the side toolbar and click on (7.5 0 0) in the drawing area.
2. In the *General* tabsheet the default option for the shape (Free) and the default orientation axes (x-axis, y-axis) are valid for this polycurve.
3. In the *Segments* tabsheet, click on the *Add segment* in the top toolbar. Set the *Segment type* to *Arc*, *Relative start angle* to 90°, *Radius* to 7.5 m and *Segment angle* to 180°.
4. Click *Close polycurve* from the top toolbar to close the polycurve.
5. Click *OK* to add the polycurve to the geometry and to close the *Shape desginer*.
6. Click on the created polycurve and select the *Extrude object* and set the z value to -15 m.
7. Multi select the two created polycruntves, right-click and select Delete from the appearing menu.

Figure 38: Deleting the two created polycruntves

The geometry of the project is defined. A screenshot of the geometry is shown below.
Generate the mesh

In order to generate the mesh:

1. Click on the Mesh tab to proceed to the Mesh mode.
2. Hide the soil volume around the suction pile. Multi-select the suction pile, the surface around the suction pile and the top surface of the suction pile.
3. In the Selection explorer set the value of Coarseness factor to 0.25.
4. The element distribution is Medium. Click the Generate mesh button to generate the mesh.
5. Proceed to the Staged construction mode.

Define the calculation

The calculation for this exercise will consist of 6 phases. These are the determination of initial conditions, the installation of the suction pile and four different load conditions. The effect of the change of the load direction while keeping the magnitude unchanged will be analysed.

1. Click on the Staged construction tab to proceed with the definition of the calculation phases.
2. Keep the calculation type of the Initial phase to K0 procedure. Ensure that all the structures and interfaces are switched off.
3. Add a new calculation phase and rename it as Install pile.
4. For this phase, we use the option of Ignore undrained behaviour.
5. Activate all the rigid bodies and interfaces in the project.
6. Add a new phase and rename it as Load pile 30 degrees.
7. In the Phases window, check the Reset displacements to zero checkbox in the Deformation control parameters subtree.
8. Set the Solver type to Paradiso (multicore direct) to enable a faster calculation for this particular project.
9. In the Numerical control parameters subtree uncheck the Use default iter parameters checkbox, which allows you to change advanced settings.
10. Set the Max load fraction per step to 0.1.
11. Click on the Rigid bodies in the Model explorer.
12. In the Selection explorer tree, set $F_x = 1949$ kN and $F_z = 1125$ kN for the selected rigid bodies.
13. Define the remaining phases according to the information in Table 11 (on page 61). For each phase select the Reset displacements to zero option and set Solver type to Pardiso (multicore direct) and Max load fraction per step to 0.1.

Table 11: Load information at the chain attachment point

<table>
<thead>
<tr>
<th>Phase</th>
<th>Start from phase</th>
<th>$F_x$</th>
<th>$F_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load pile 30 degrees</td>
<td>Phase_1</td>
<td>1949 kN</td>
<td>1125 kN</td>
</tr>
<tr>
<td>[Phase_2]</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load pile 40 degrees</td>
<td>Phase_1</td>
<td>1724 kN</td>
<td>1447 kN</td>
</tr>
<tr>
<td>[Phase_3]</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load pile 50 degrees</td>
<td>Phase_1</td>
<td>1447 kN</td>
<td>1724 kN</td>
</tr>
<tr>
<td>[Phase_4]</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load pile 60 degrees</td>
<td>Phase_1</td>
<td>1125 kN</td>
<td>1949 kN</td>
</tr>
<tr>
<td>[Phase_5]</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The order of the phases is indicated in the Phases explorer. Calculation of Phase_1 starts after the calculation of Initial phase is completed. The calculation of the remaining phases starts after the calculation of the pile installation phase is completed.

![Phases explorer](image)
4.6.1 Execute the calculation

1. Start the calculation process.
2. Save the project when the calculation is finished.

4.7 Results

To view the results:

1. View the results of the last calculation phase. The deformed mesh of the whole geometry will be shown. In particular, the displacements of the suction pile itself are of interest.
2. Select the shadings representation and rotate the model such that the x-axis is perpendicular to the screen.
3. If the axes are not visible, select this option from the View menu. It is quite clear that the point force acting on the pile does not disturb the displacement field locally indicating that the pile is sufficiently thick here.
4. In the same manner, the total displacements of the suction pile under a different direction of the load can be inspected by selecting the appropriate phase from the drop-down menu. In particular, Phase_2 is of interest, as in this phase the horizontal part of the load will have the largest value.
Note: As an alternative for true 3D finite element calculations, the online tool SPCalc from XG-Geotools provides a quick solution for multiple bearing capacity calculations of suction piles. For more information see www.xg-geotools.com.
Construction of a road embankment

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 two new calculation options are introduced, namely a consolidation analysis and the calculation of a safety factor by means of a safety analysis (phi/c-reduction). It also involves the modelling of drains to speed up the consolidation process.

Objectives

- Modelling drains
- Consolidation analysis
- Change of permeability during consolidation
- Safety analysis (phi/c-reduction)

Geometry

The figure below shows a cross section of a road embankment.

![Figure 42: Situation of a road embankment on soft soil](image)

5.1 Create a new project

The embankment is 16 m wide. 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). A representative section of 2 m is considered in the project. The embankment itself is composed of loose sandy soil. The subsoil consists of 6 m of soft soil. The upper 3 m of this soft soil layer is modelled as a peat layer and the lower 3 m as 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.

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. Keep the default units and set the model dimensions to
   a. \( x_{\text{min}} = 0 \) and \( x_{\text{max}} = 60 \)
   b. \( y_{\text{min}} = 0 \) and \( y_{\text{max}} = 2 \).

### 5.2 Define the soil stratigraphy

The soil layers comprising the embankment foundation are defined using a borehole. The embankment layers are defined in the **Structures** mode.

1. Click the **Create borehole** button and create a borehole at (0 0 0). The **Modify soil layers** window pops up.
2. Define three soil layers as shown in **Figure 43** (on page 65).
3. The water level is located at \( z = -1 \) m. In the borehole column specify a value of -1 to **Head**.

![Figure 43: Soil layer distribution](image)
5.3 Create and assign the material data sets

The material properties for the data sets are shown in the table below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Embankment</th>
<th>Sand</th>
<th>Peat</th>
<th>Clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Material model</strong></td>
<td>Model</td>
<td>Hardening Soil model</td>
<td>Hardening Soil model</td>
<td>Soft Soil model</td>
<td>Soft Soil model</td>
<td>-</td>
</tr>
<tr>
<td><strong>Type of material behaviour</strong></td>
<td>Type</td>
<td>Drained</td>
<td>Drained</td>
<td>Undrained (A)</td>
<td>Undrained (A)</td>
<td>-</td>
</tr>
<tr>
<td><strong>Soil unit weight above phreatic level</strong></td>
<td>$\gamma_{\text{unsat}}$</td>
<td>16</td>
<td>17</td>
<td>8</td>
<td>15</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td><strong>Soil unit weight below phreatic level</strong></td>
<td>$\gamma_{\text{sat}}$</td>
<td>19</td>
<td>20</td>
<td>12</td>
<td>18</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td><strong>Initial void ratio</strong></td>
<td>$e_{\text{init}}$</td>
<td>0.5</td>
<td>0.5</td>
<td>2.0</td>
<td>1.0</td>
<td>-</td>
</tr>
<tr>
<td><strong>Parameters</strong></td>
<td>$\lambda^*$</td>
<td>-</td>
<td>-</td>
<td>0.15</td>
<td>0.05</td>
<td>-</td>
</tr>
<tr>
<td><strong>Modified swelling index</strong></td>
<td>$\kappa^*$</td>
<td>-</td>
<td>-</td>
<td>0.03</td>
<td>0.01</td>
<td>-</td>
</tr>
<tr>
<td><strong>Secant stiffness in standard drained triaxial test</strong></td>
<td>$E_{50}^{\text{ref}}$</td>
<td>$2.5 \cdot 10^4$</td>
<td>$3.5 \cdot 10^4$</td>
<td>-</td>
<td>-</td>
<td>kN/m$^2$</td>
</tr>
</tbody>
</table>
## Construction of a road embankment

Create and assign the material data sets

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Embankment</th>
<th>Sand</th>
<th>Peat</th>
<th>Clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tangent stiffness for primary oedometer loading</td>
<td>$E_{oed}^{ref}$</td>
<td>$2.5 \cdot 10^4$</td>
<td>$3.5 \cdot 10^4$</td>
<td>-</td>
<td>-</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Unloading / reloading stiffness</td>
<td>$E_{ur}^{ref}$</td>
<td>$7.5 \cdot 10^4$</td>
<td>$1.05 \cdot 10^5$</td>
<td>-</td>
<td>-</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Power for stress-level dependency of stiffness</td>
<td>$m$</td>
<td>0.5</td>
<td>0.5</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion (constant)</td>
<td>$c_{ref}'$</td>
<td>1.0</td>
<td>0.0</td>
<td>2.0</td>
<td>1.0</td>
<td>kN/m²</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\varphi'$</td>
<td>30</td>
<td>33</td>
<td>23</td>
<td>25</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>0.0</td>
<td>3.0</td>
<td>0</td>
<td>0</td>
<td>°</td>
</tr>
<tr>
<td>Advanced: Set to default</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>-</td>
</tr>
</tbody>
</table>

### Groundwater

<table>
<thead>
<tr>
<th>Data set</th>
<th>USDA</th>
<th>USDA</th>
<th>USDA</th>
<th>USDA</th>
<th>USDA</th>
<th>-</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>-</td>
</tr>
<tr>
<td>Soil type</td>
<td>Loamy sand</td>
<td>Sand</td>
<td>Clay</td>
<td>Clay</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>&gt; 2μm</td>
<td>6.0</td>
<td>4.0</td>
<td>70.0</td>
<td>70.0</td>
<td>%</td>
<td></td>
</tr>
<tr>
<td>2μm - 50μm</td>
<td>11.0</td>
<td>4.0</td>
<td>13.0</td>
<td>13.0</td>
<td>%</td>
<td></td>
</tr>
<tr>
<td>50μm - 2mm</td>
<td>83.0</td>
<td>92.0</td>
<td>17.0</td>
<td>17.0</td>
<td>%</td>
<td></td>
</tr>
<tr>
<td>Use defaults</td>
<td>From data set</td>
<td>From data set</td>
<td>None</td>
<td>From data set</td>
<td>-</td>
<td></td>
</tr>
</tbody>
</table>
### Construction of a road embankment

Create and assign the material data sets

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Embankment</th>
<th>Sand</th>
<th>Peat</th>
<th>Clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal permeability (x-direction)</td>
<td>k_x</td>
<td>3.499</td>
<td>7.128</td>
<td>0.1</td>
<td>0.04752</td>
<td>m/day</td>
</tr>
<tr>
<td>Horizontal permeability (y-direction)</td>
<td>k_y</td>
<td>3.499</td>
<td>7.128</td>
<td>0.1</td>
<td>0.04752</td>
<td>m/day</td>
</tr>
<tr>
<td>Vertical permeability</td>
<td>k_z</td>
<td>3.499</td>
<td>7.128</td>
<td>0.05</td>
<td>0.04752</td>
<td>m/day</td>
</tr>
<tr>
<td>Change in permeability</td>
<td>c_k</td>
<td>1 · 10^{15}</td>
<td>1 · 10^{15}</td>
<td>1.0</td>
<td>0.2</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Interfaces</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Interface strength</td>
<td>-</td>
<td>Rigid</td>
<td>Rigid</td>
<td>Rigid</td>
<td>Rigid</td>
<td>-</td>
</tr>
<tr>
<td>Strength reduction factor inter.</td>
<td>R_{inter}</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Initial</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>K_0 determination</td>
<td>-</td>
<td>Automatic</td>
<td>Automatic</td>
<td>Automatic</td>
<td>Automatic</td>
<td>-</td>
</tr>
<tr>
<td>Over-consolidation ratio</td>
<td>OCR</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>-</td>
</tr>
<tr>
<td>Pre-overburden pressure</td>
<td>POP</td>
<td>0.0</td>
<td>0.0</td>
<td>5.0</td>
<td>0.0</td>
<td>kN/m^2</td>
</tr>
</tbody>
</table>

**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 due to compression of the soil. This option is recommended when using advanced models.

1. Click the **Materials** button 📊.
2. Create soil material data sets according to Table 12 (on page 66) and assign them to the corresponding layers in the borehole (Figure 43 (on page 65)).

3. Close the Modify soil layers window and proceed to the Structures mode to define the structural elements.

5.4 Definition of embankment and drains

The embankment and the drains are defined in the Structures mode. To define the embankment layers:

1. Reorientate the model such that the front view is displayed by clicking the corresponding button in the toolbar.
2. Create a surface by defining points at (0 0 0), (0 0 4), (8 0 4) and (20 0 0).
3. Create a line passing through (0 0 2) and (14 0 2) to define the embankment layers.
4. Select both the created line and surface by keeping the <Ctrl> key pressed while clicking them in the model.
5. Click the Extrude object button. Assign a value of 2 to the y-component of the extrusion vector as shown in the figure below and click Apply.

![Figure 44: Extrusion window](image)

6. Delete the surface and the line with its corresponding points that were created before the extrusion.
7. Right-click the volume created by extrusion and select the menu item Soil_4 > Set material > Embankment.

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.

Drains are arranged in a square pattern, having a distance of 2 m between two consecutive drains in a row (or column). Only one row of drains will be considered in this tutorial. To create the drain pattern:

1. Click the Create hydraulic conditions button in the side toolbar.
2. Click the Create line drain button in the appearing menu. Define a line drain in the model between points (1 1 0) and (1 1 -6).
3. Click the Create array button to define the drain pattern.
4. In the Create array window select the menu item *Shape > 1D, in x direction* in the drop-down menu and specify the pattern as shown in the figure below.

![Create array window](image)

*Figure 45: Settings of the drain pattern*

The model geometry is shown in the following figure:

![Model geometry](image)

*Figure 46: Model geometry*

5.5 Generate the mesh

1. Proceed to the *Mesh* mode.
2. Select all volumes, including the embankment and in the *Selection explorer* set the *Coarseness factor* to 0.3.
3. Click the *Generate mesh* button. Set the element distribution to *Coarse*. 
4. View the generated mesh.

The resulting mesh is shown below.

![Figure 47: The generated mesh](image)

5.6 Define the calculation

The embankment construction process will be considered twice. In the first calculation the drains will not be considered.

5.6.1 Initial phase

In the initial situation the embankment is not present. Therefore, the corresponding soil volumes are deactivated in the initial phase. The **K0 procedure** can be used to calculate the initial stresses. The initial water pressures are fully hydrostatic and based on a general phreatic level defined by the **Head** value assigned to the boreholes. For the **Initial phase**, the **Phreatic** option is selected for the pore pressure calculation type and the **Global water level** is set to **BoreholeWaterlevel_1** corresponding to the water level defined by the heads specified for the boreholes.

The boundary conditions for flow can be specified in the **Model conditions** subtree in the **Model explorer**. In the current situation the left vertical boundary (Xmin) must be closed because of symmetry, so horizontal flow should not occur. The bottom is open because the excess pore pressures can freely flow into the deep and permeable sand layer. The upper boundary is obviously open as well. The view of the **GroundwaterFlow** subtree after the definition is given in the figure below.
5.6.2 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 3D allows for a fully automatic time stepping procedure that takes this critical time step into account. Within this procedure there are three main possibilities for the *Loading type* parameter:

- 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 the soil has reached a specified degree of consolidation (*Degree of consolidation*).

**Consolidation process - No drains**

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.

To define the calculation phases, follow these steps:

![Model explorer (InitialPhase)](image)

*Figure 48: Boundary conditions for groundwater flow*
Construction of a road embankment

Define the calculation

Phase 1
1. Click the Add phase button to introduce the first construction phase.
2. In the General subtree select the Consolidation option in the Calculation type drop-down menu.
3. The Loading type is by default set to Staged construction. This option will be used for this phase.
4. The Phreatic option is automatically selected for the pore pressure calculation type. Note that the global water level for a calculation phase can be defined in the Water subtree available under the Model conditions in the Model explorer.
5. Specify a value of 2 days to the Time interval and click OK to close the Phases window.
6. In the Staged construction mode activate the first part of the embankment.

Phase 2
The second phase is also a Consolidation analysis. 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 introduce the next calculation phase.
2. Define the calculation type as Consolidation.
3. Specify a value of 30 days to the Time interval. The default values of the other parameters are used for this phase.

Phase 3

1. Click the Add phase button to introduce the next calculation phase.
2. Define the calculation type as Consolidation.
3. Specify a value of 1 day to the Time interval. The default values of the other parameters are used.
4. In the Staged construction mode activate the second part of the embankment.

Phase 4
The fourth phase is a Consolidation analysis to a minimum excess pore pressure.

1. Click the Add phase button to introduce the next calculation phase.
2. Define the calculation type as Consolidation.
3. Select the Minimum excess pore pressure option in the Loading type drop-down menu. The default value for the minimum pressure (\(|P\text{-stop}| = 1.0 \text{ kN/m}^2\)) as well as the default values for other parameters are used.

The definition of the calculation phases is complete.

5.6.3 Execute the calculation
1. Before starting the calculation, click the **Select points for curves** button and select the following points:
   As the first node, select the toe of the embankment at (20 0 0). The second node 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 is needed, close to (but not actually on) the left boundary (e.g. (0.7 0 -3)).

2. Start the calculation.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info
window. In addition to the multipliers, a parameter $P_{\text{max}}$ 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.

![Figure 49: Calculation progress displayed in the Active tasks window](image)

### 5.7 Results

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 shows the uplift of the embankment toe and hinterland due to the undrained behaviour.
Construction of a road embankment

Results

**Figure 50: Deformed mesh after undrained construction of embankment (Phase 3)**

1. Select the menu item **Deformations > Incremental displacements > |Δu|**.
2.  
   Select the menu item **View > Arrows** 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:

**Figure 51: Displacement increments after undrained construction of embankment**

1. Click `<Ctrl> + <7>` to display the developed excess pore pressures (see Appendix F of the PLAXIS 3D Reference Manual for more shortcuts). They can be displayed by selecting the corresponding option in the side menu displayed as the **Pore pressures** option is selected in the **Stresses** menu.
2.  
   Click the **Center principal directions**. The principal directions of excess pressures are displayed at the center of each soil element. The results are displayed in the figure below. It is clear that the highest excess pore pressure occurs under the embankment centre.
1. Select Phase 4 in the drop down menu.
2. Define a vertical cross section passing through (0 1) and (60 1).
3. Click the Contour lines button in the toolbar to display the results as contours.
4. Select the menu item View > Viewpoint. The corresponding window pops up.
5. In the Viewpoint window select the Front view option as shown:

   ![Figure 53: Viewpoint window](image)

6. Use the Draw scanline button or the corresponding option in the View menu to define the position of the contour line labels.

It can be seen that the settlement of the original soil surface and the embankment increases considerably during the fourth phase. This is due to the dissipation of the excess pore pressures (= consolidation), which causes further settlement of the soil. The figure below shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below 1.0 kN/m².
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. Click the *Curves manager* button in the toolbar. The corresponding window pops up.
2. In the *Charts* tabsheet click *New*. The *Curve generation* window pops up.
3. For the x-axis, select the *Project* option from the drop-down menu and select *Time* in the tree.
4. For the y-axis, select the point in the middle of the soft soil layers (the second node (0.7 0.0 -3)) from the drop-down menu. In the tree select *Stresses > Pore pressure > p excess*.
5. Select the *Invert sign* option for the y-axis.
6. Click *Ok* to generate the curve.
7. Click the *Settings* button in the toolbar. The *Settings* window will appear displaying the tabsheet of the created curve.
8. Click the *Phases* button and select the phases 1 to 4 in the appearing window.
9. Rename the curve by typing *Phases 1 - 4* in the *Curve title* cell.
10. Click *Apply* to update the plot.
11. Save the chart in Output and save the project in Input.

**Note:** To display the legend inside the chart area right-click on the name of the chart and select the menu item *View > Legend in chart*.

The figure below 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.
5.8 Safety analysis

5.8.1 General considerations

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.

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:

\[
S_{\text{global}} = \frac{S_{\text{max}} \text{imum available}}{S_{\text{needed forequilibrium}}}
\]

Where \( S \) represents the shear strength. The ratio of the true strength to the computed minimum strength required for equilibrium is the safety factor that is conventionally used in soil mechanics. By introducing the standard Coulomb condition, the safety factor is obtained:

\[
S_{\text{global}} = \frac{c - \sigma_n \tan(\phi)}{c' - \sigma_n \tan(\phi'_r)}
\]

Where \( c \) and \( \phi \) are the input strength parameters and \( \sigma_n \) is the actual normal stress component. The parameters \( c' \) and \( \phi'_r \) are reduced strength parameters that are just large enough to maintain equilibrium. The principle described above is the basis of a Safety analysis that can be used in PLAXIS 3D to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:
The reduction of strength parameters is controlled by the total multiplier $\Sigma M_{sf}$. This parameter is increased in a step-by-step procedure until failure occurs. The safety factor is then defined as the value of $\Sigma M_{sf}$ at failure, provided that at failure a more or less constant value is obtained for a number of successive load steps.

The Safety calculation option is available in the Calculation type drop-down menu in the Phases window.

5.8.2 Define the calculation

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

1. In the Phases window and select Phase 1 in the Start from phase drop-down menu.
2. Add a new calculation phase.
3. In the General subtree, select Safety as calculation type.
4. The Loading type is automatically changed to Incremental multipliers.
5. The first increment of the multiplier that controls the strength reduction process, $M_{sf}$, is set automatically to 0.1. This value will be used in this tutorial.
6. Note that the Use pressures from the previous phase option in the Pore pressure calculation type drop-down menu is automatically selected and grayed out indicating that this option cannot be changed.
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. The default values of all the remaining parameters will be used. The first safety calculation has now been defined.
8. Follow the same steps to create new calculation phases that analyse the stability at the end of each consolidation phase. In addition to selecting Safety as calculation type, select the corresponding consolidation phase as the Start from phase parameter. The Phases explorer displaying the Safety calculation phases is shown below.
9. Calculate the safety phases.

**Note:**
- The default value of *Max steps* in a Safety calculation is 100. In contrast to an *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 10000.
- 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, Σ Msf, is automatically controlled by the load advancement procedure.

### 5.8.3 Evaluation of the results - Safety

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 the last *Safety* phase (Phase_8) and click the *View calculation results* button.
2. Select the menu item *Deformations* > *Incremental displacements* > **|Δu|**.
3. Change the presentation from *Arrows* to *Shadings*. The resulting plots give a good impression of the failure mechanisms. The magnitude of the displacement increments is not relevant.
The safety factor can be obtained from the Calculation info option of the Project menu. The value of $\Sigma \text{Ms f}$ 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 $\Sigma \text{Ms f}$ 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 (the first node) for the x-axis. Select the menu item Deformations > Total displacements > $|u|$.  
4. For the y-axis, select Project and then select Multiplier > $\Sigma \text{Ms f}$. The Safety phases are considered in the chart. As a result, the curve of appears.
5. Set x-axis interval maximum to 1 in Chart tab.
The maximum displacements plotted are not relevant. It can be seen that for all curves a more or less constant value of $\Sigma M_{sf}$ is obtained. Hovering the mouse cursor over a point on the curves, a box showing the exact value of $\Sigma M_{sf}$ can be obtained.

5.9 Using drains

In this section the effect of the drains in the project will be investigated. The embankment constructions will be redefined by introducing four new phases having the same properties as the first four consolidation phases. The differences in the new phases are:

- The drains should be active in all the new phases.
- The **Time interval** in the first three of the consolidation phases (1 to 3) 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 excess pressure ($|P_{-\text{stop}}|$).

1. After the calculation is finished, 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 node (0.7 0 -3) can be used.

2. Open the **Curves manager**

3. In the **Chart** tabsheet double click Chart 1 ($p_{\text{excess}}$ of node (0.7 0 -3) versus time). Select the displaying phases to Phases 1-4. Close the **Curves manager**.

4. Click the **Settings** button in the toolbar. The **Settings** window pops up.

5. Click the **Add curve** button and select the **Add from current project** option in the appearing menu. The **Curve generation** window pops up.

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

6. Select the **Invert sign** option for $y$-axis.

7. Click **OK** to accept the selected options and close the **Curve generation** window.

8. In the chart a new curve is added and a new tabsheet corresponding to it is opened in the **Settings** window.

9. Click the **Phases** button. From the displayed window select the **Initial phase** and the last four phases (drains) and click **OK**.

10. In the **Settings** click **Apply** to preview the generated curve.

11. 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.
Construction of a road embankment
Using drains

Figure 59: Effect of drains
Stability of a diaphragm wall excavation

This lesson is concerned with a diaphragm wall that is constructed in a stiff sandy clay layer with a groundwater level at 1.0 m below the surface. The excavation process of a diaphragm wall is executed in a specific sequence to obtain the maximum support from the surrounding soil and to prevent soil collapse. A diaphragm wall consists of a number of individually constructed sections. The construction of one such section is modelled in this exercise.

**Objectives**

- Defining user-defined water conditions
- Modelling of diaphragm walls installation

**Geometry**

A single diaphragm section is excavated in three parts, and the construction can be modelled in five phases. In the first three phases, the wall is excavated part by part in the sequence as shown in the figure below. During the excavation, fluid bentonite with a unit weight of 11 kN/m$^3$ is simultaneously pumped in the trench so that the bentonite pressure and the arching in the soil prevents the surrounding soil from collapse. After digging of the trench has been completed, in the fourth phase, fluid concrete is poured in the trench replacing the bentonite. In the fifth phase the concrete hardens, and the diaphragm wall section is complete. The stability of the excavation is lowest in the third phase, when the section is entirely excavated and filled with bentonite. A safety factor is calculated through a phi-c reduction procedure after each phase to observe the stability of the excavation.

![Geometry of the diaphragm wall](image)

*Figure 60: Geometry of the diaphragm wall*
6.1 Create a new project

The diaphragm wall considered in this exercise is 30 m deep and 1.2 m thick. One section is 7.0 m wide and consists of three excavation parts; part I and II are 2.5 m wide and part III is 2.0 m wide. The wall is symmetric about its central plane, so only one half of the thickness needs to be modelled. The interaction between the wall and the soil is considered to be fully rough, therefore interfaces are not required.

To define the geometry for this exercise, follow these steps:

1. Start the Input program and select New project from the Create/Open project dialog box.
2. Enter an appropriate title for the project.
3. Keep the standard units and set the model dimensions to
   a. \( x_{\text{min}} = 0 \) and \( x_{\text{max}} = 20 \),
   b. \( y_{\text{min}} = 0 \) and \( y_{\text{max}} = 10 \).
4. Click OK.

6.2 Define the soil stratigraphy

In the current example only one horizontal soil layer is present. A single borehole is sufficient to define it.

1. Click the Create borehole button and create a borehole at \((0 \ 0 \ 0)\).
   The Modify soil layers window pops up.
2. In the Modify soil layers window add a soil layer with top boundary at \( z = 40 \) m and bottom boundary at \( z = 0 \) m.
3. Set the Head to 39 m.

6.3 Create and assign the material data sets

The material properties for the data sets are shown in the table below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Stiff sandy clay</th>
<th>Concrete</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Mohr-Coulomb</td>
<td>Linear Elastic</td>
<td>-</td>
</tr>
</tbody>
</table>
### Stability of a diaphragm wall excavation

#### Definition of the diaphragm wall

The diaphragm wall is modelled in the Structures mode. The volume elements composing the diaphragm wall are generated by extruding rectangular surfaces.

The coordinates for the surfaces are given in the following table:

<table>
<thead>
<tr>
<th>Segment</th>
<th>Point coordinates</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(6.5 0 40) (9 0 40) (9 0.6 40) (6.5 0.6 40)</td>
</tr>
</tbody>
</table>

1. Click the **Materials** button.
2. Create the data sets for the soil layer and the concrete as specified in Table 13 (on page 85).
3. Assign the 'Stiff sandy clay' material data set to the soil layer and close the Material sets window.
1. Click the **Create surface** button in the side toolbar and create three surfaces accordingly to Table 14 (on page 86).

2. Select the created surfaces by keeping the `<Ctrl>` key pressed while clicking them in the model.

3. Click the **Extrude object** button in the side toolbar. Set the extrusion vector to (0 0 -30) and the extrusion vector length to 30.

4. Delete the surfaces.

---

**6.5 Generate the mesh**

In order to generate the mesh:
1. Click on the **Mesh** tab to proceed to the **Mesh** mode.
2. Multi-select all the volume elements of the diaphragm wall.
3. In the **Selection explorer** set the value of **Coarseness factor** to 0.50.
4. Click the **Generate mesh** button. The default option (**Medium**) is used to generate the mesh.
5. Click the **View mesh** button to inspect the generated mesh.

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

### 6.6 Define the calculation

The calculation consists of five phases. In the first phase, part I of the excavation is removed and simultaneously filled with bentonite. The bentonite, with a unit weight of 11 kN/m$^3$, is simulated employing an artificial 'water' pressure that increases linearly with depth. This pressure replaces the original water pressure inside the excavation. In the second and third phases of the excavation parts, II and III are removed and filled with bentonite. In the fourth phase, the entire excavated trench is filled with fluid concrete. The fluid concrete with a
unit weight of 24 kN/m³ is simulated by a change in the artificial 'water' pressure. In phase 5, the hardening of the concrete is simulated by removing the artificial pressures, reactivating the excavated clusters and assigning the concrete material set to these clusters.

6.6.1 Initial phase

The initial phase consists of the generation of the initial stresses using the K0 procedure. The default settings for the initial phase are valid.

6.6.2 Phase 1 - Excavation of part I

1. Add the first calculation phase.
2. Select the first excavation volume (part I).
3. In the selection explorer, deactivate the soil volume. Set the water condition to User-defined and enter $z_{\text{ref}} = 40$ m, $p_{\text{ref}} = 0.0$ kN/m² and $p_{\text{inc}} = -11$ kN/m²/m.

![Selection explorer image](image)

Figure 63: User-defined water condition in part I

A bentonite pressure is now defined in part I of the excavation, starting at 0 kN/m² at the reference level of 40 m and increasing at 11 kN/m² per m depth, resulting in 330 kN/m² at the bottom of the excavation.

4. Click the Preview phase button to check the settings for the current phase.
6.6.3 Phase 2 - Excavation of part II

1. Add a new phase.
2. Select the second excavation volume (part II).
3. In the selection explorer, deactivate the soil volume. Set the water condition to *User-defined* and enter \( z_{ref} = 40 \text{ m}, p_{ref} = 0.0 \text{ kN/m}^2 \) and \( p_{inc} = -11 \text{ kN/m}^2/\text{m} \).

6.6.4 Phase 3 - Excavation of part III

1. Add a new phase.
2. Select the third excavation volume (part III).
3. In the selection explorer, deactivate the soil volume. Set the water condition to *User-defined* and enter \( z_{ref} = 40 \text{ m}, p_{ref} = 0.0 \text{ kN/m}^2 \) and \( p_{inc} = -11 \text{ kN/m}^2/\text{m} \).

6.6.5 Phase 4 - Fluid concrete

The bentonite in the excavation is now replaced by fluid concrete with a weight of 24.0 kN/m\(^3\).

1. Add a new phase.
2. Select the three excavation volumes.
3. In the selection explorer, change the *User-defined* water conditions and enter \( p_{inc} = -24 \text{ kN/m}^2/\text{m} \). The other parameters must be kept at their original values (\( z_{ref} = 40 \text{ m}, p_{ref} = 0.0 \text{ kN/m}^2 \)).

6.6.6 Phase 5 - Cured concrete

1. Add a new phase.
2. Select the three excavation volumes.
3. In the selection explorer, reactivate the soil volumes and set the material to concrete.
4. Set the water condition to *Dry*.

**Note:**
Although the concrete is non-porous and the calculation program will automatically assume zero pore pressures in these elements, it is a good practise to regenerate the water pressures such that the generated pore pressures correspond to those used in the calculation program.

6.6.7 Phase 6 to 9 - Safety analysis

In Phases 6 to 9, stability calculations are defined for the previous phases respectively except for the fluid concrete phase (less critical than the bentonite phase thanks to the higher unit weight). Phase 3 should be the most critical because the support pressure from the bentonite is low. Also, the excavation is at its full width,
which reduces the possibility for lateral arching. A check on whether Phase 3 is the most critical stage can be carried out by calculating the safety factors for the first three phases through a Safety analysis.

1. Select Phase_1 in the Phases explorer.
2. Add a new calculation phase and proceed to the Phases window.
3. Set Calculation type to Safety. The Incremental multipliers option is valid as Loading type.
4. Select the Reset displacements to zero option in the Deformation control subtree.
5. In the Numerical control parameters subtree set the Max steps parameter to 40.
6. Follow the same procedure to add Safety analysis phases following phases 2, 3 and 5.

6.6.8 Execute the calculation

1. In the Staged construction mode select some nodes near (10 1 40) and (10 4.5 40) for curves.
2. Start the calculation process.
3. Save the project when the calculation is finished.

6.7 Results

The stability of the excavation can be evaluated from the calculated safety factor after each excavation stage. Use the Curves program to plot Σ Msf (the safety factor) as a function of the displacements |u| (see Figure 64 (on page 92)). In Phase 3, the stability is the lowest. However, Σ Msf remains greater than 1 and so collapse would not be expected.

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 tab sheet.
3. In the Curve generation window, select one of the two nodes for the x-axis. Select Deformations > Total displacements > |u|.
4. For the y-axis, select Project and then select Multiplier > Σ Msf. The Safety phases are considered in the chart. As a result, the curve of Figure 64 (on page 92) appears.
5. Set x-axis interval maximum to 0.1 in Chart tab.
An important phenomenon that keeps the excavation stable is arching in the soil. This phenomenon is shown in the three figures below. To see the principal stresses directions at a chosen depth, make a horizontal cross section by clicking the \textit{Horizontal cross section} button.

1. To create such plots, make a horizontal cross-section by clicking the \textit{Horizontal cross section} button in the side bar.
2. In the window that appears fill in a cross section height of 25 m (at the mid-height of the diaphragm wall).
3. Select the menu item \textbf{Stresses > Principal total stresses > Total principal stresses}.
4. Select the top view in \textbf{View > Viewpoint} to reorientate the model in order to obtain a clearer view of the arch effect.

\textbf{Figure 64: }$\Sigma\text{Msf}$ (safety factor) as a function of the total displacement

\textbf{Figure 65: }Principal stresses directions at $z = 25$ m at the end of Phase 1
Stability of a diaphragm wall excavation

Results

Figure 66: Principal stresses directions at z = 25 m at the end of Phase_2

Figure 67: Principal stresses directions at z = 25 m at the end of Phase_4
The lining of a shield tunnel is often constructed using prefabricated concrete ring segments, which are bolted together within the tunnel boring machine to form the tunnel lining. During the erection of the tunnel lining the tunnel boring machine (TBM) remains stationary. Once a tunnel lining ring has been fully erected, excavation is resumed, until enough soil has been excavated to erect the next lining ring. As a result, the construction process can be divided into construction stages with a length of a tunnel ring, often about 1.5 m long. In each of these stages, the same steps are repeated over and over again.

In order to model this, a geometry consisting of slices each 1.5 m long can be used. The calculation consists of a number of Plastic phases, each of which models the same parts of the excavation process: the support pressure at the tunnel face needed to prevent active failure at the face, the conical shape of the TBM shield, the excavation of the soil and pore water within the TBM, the installation of the tunnel lining and the grouting of the gap between the soil and the newly installed lining. In each phase the input for the calculation phase is identical, except for its location, which will be shifted by 1.5 m each phase.

Objectives

• Modelling of the tunnel boring process with a TBM
• Modelling of the cone shape of the TBM
• Using Tunnel designer to define geometry, trajectory and sequencing of the tunnel

Figure 68: Construction stages of a shield tunnel model

7.1 Define the geometry
In the model, only one symmetric half is included. The model is 20 m wide, it extends 80 m in the y-direction and it is 20 m deep. These dimensions are sufficient to allow for any possible collapse mechanism to develop and to avoid any influence from the model boundaries.

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. Keep the default units and set the model dimensions to
   
   a. $x_{\text{min}} = -20$ and $x_{\text{max}} = 0$,

   b. $y_{\text{min}} = 0$ and $y_{\text{max}} = 80$.

### 7.2 Define the soil stratigraphy

The subsoil consists of three layers. The soft upper sand layer is 2 m deep and extends from the ground surface to Mean Sea Level (MSL). Below the upper sand layer, there is a clay layer of 12 m thickness and this layer is underlain by a stiff sand layer that extends to a large depth. Only 6 m of the stiff sand layer is included in the model. Hence, the bottom of the model is 18 m below MSL. Soil layer is assumed to be horizontal throughout the model and so just one borehole is sufficient to describe the soil layers. The present groundwater head corresponds to the MSL.

1. Press the **Create borehole** button and click at the origin of the system of axis to create a borehole at (0 0 0). The **Modify soil layers** window will open.
2. Define 3 layers: Upper sand with the top at 2 m and the bottom at 0 m, Clay with the bottom at -12 m and Stiff sand with the bottom at -18 m.

![Figure 69: Soil layer distribution](image-url)
### 7.3 Create and assign the material data sets

The material properties for the data sets are shown in the table below.

**Table 15: Material properties for the soil layers**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Upper sand</th>
<th>Clay</th>
<th>Stiff sand</th>
<th>Concrete</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>General</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Mohr-Coulomb</td>
<td>Mohr-Coulomb</td>
<td>Mohr-Coulomb</td>
<td>Linear elastic</td>
<td>-</td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td>Drained</td>
<td>Drained</td>
<td>Drained</td>
<td>Non porous</td>
<td>-</td>
</tr>
<tr>
<td>Unit weight above phreatic level</td>
<td>( \gamma_{\text{unsat}} )</td>
<td>17.0</td>
<td>16.0</td>
<td>17.0</td>
<td>27.0</td>
<td>kN/m(^3)</td>
</tr>
<tr>
<td>Unit weight below phreatic level</td>
<td>( \gamma_{\text{sat}} )</td>
<td>20.0</td>
<td>18.0</td>
<td>20.0</td>
<td>-</td>
<td>kN/m(^3)</td>
</tr>
<tr>
<td><strong>Parameters</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Young's modulus</td>
<td>( E' )</td>
<td>( 1.3 \cdot 10^4 )</td>
<td>( 1.0 \cdot 10^4 )</td>
<td>( 7.5 \cdot 10^4 )</td>
<td>( 3.1 \cdot 10^7 )</td>
<td>kN/m(^2)</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>( \nu' )</td>
<td>0.3</td>
<td>0.35</td>
<td>0.3</td>
<td>0.1</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>( c'_{\text{ref}} )</td>
<td>1.0</td>
<td>5.0</td>
<td>1.0</td>
<td>-</td>
<td>kN/m(^2)</td>
</tr>
<tr>
<td>Friction angle</td>
<td>( \phi' )</td>
<td>31</td>
<td>25</td>
<td>31</td>
<td>-</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>( \psi )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-</td>
<td>°</td>
</tr>
<tr>
<td><strong>Interfaces</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interface strength</td>
<td>-</td>
<td>Rigid</td>
<td>Rigid</td>
<td>Rigid</td>
<td>Rigid</td>
<td>-</td>
</tr>
</tbody>
</table>
1. Open the materials database by clicking the Materials button and create the data sets for the soil layers and the final concrete lining in the tunnel as specified in the Table above (on page 96).

2. Assign the material data sets to the corresponding soil layers (Figure 69 (on page 95)) and close the Modify soil layers window. The concrete data set will be assigned later.

### 7.4 Definition of structural elements

The tunnel excavation is carried out by a tunnel boring machine (TBM) which is 9.0 m long and 8.5 m in diameter. The TBM already advanced 25 m into the soil. Subsequent phases will model an advancement by 1.5 m each.

**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 Staged construction mode, Reference Manual - Chapter 7 - Definition of connections.)
- The tunnel shape is composed of arcs with different radii (e.g. NATM tunnels).

#### Table 16: Material properties of the plate representing the TBM

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>TBM</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of behaviour</td>
<td>-</td>
<td>Elastic; Isotropic</td>
<td>-</td>
</tr>
<tr>
<td>Thickness</td>
<td>d</td>
<td>0.17</td>
<td>m</td>
</tr>
<tr>
<td>Material weight</td>
<td>γ</td>
<td>247</td>
<td>kN / m³</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>E₁</td>
<td>200·10⁶</td>
<td>kN / m²</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>v₁₂</td>
<td>0</td>
<td>-</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>G₁₂</td>
<td>100·10⁶</td>
<td>kN / m²</td>
</tr>
</tbody>
</table>
**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.

### 7.4.1 Create tunnel

In **Structures** mode both the geometry of the tunnel and the TBM will be defined.

1. Click the **Start designer** button in the side toolbar.
2. Click the **Create tunnel** button from the list.
3. Click anywhere on the drawing area to define the insertion point. The **Tunnel designer** window pops up.
4. In the **Selection explorer** set the insertion point of the tunnel to (0 0 -13.25)

![Figure 70: Insertion point of the tunnel](image)

5. In the **General** tabsheet select the **Circular** option in the drop-down menu for the **Shape type**.
6. The left half of the tunnel is generated in this example. Select the **Define left half** option in the drop-down menu for the **Whole or half tunnel**. A screenshot of the **General** tabsheet after the proper assignment is given in **Figure 71** (on page 99).
7. Click the **Segments** tabsheet 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.
8. In the **Segment** box set **Radius** to 4 m. This is the inner radius of the tunnel.
9. Proceed to the **Subsections** tabsheet.
10. Click the **Generate thick lining** button in the side toolbar. The **Generate thick lining** window pops up.
11. Assign a value of 0.25 m and click **OK**. A screenshot of the **Cross section** tabsheet after the proper assignment is given in **Figure 72** (on page 99).
12. Proceed to the **Properties** tabsheet. Here we define the properties for the tunnel such as grout pressure, surface contraction, jack forces and the tunnel face pressure.
13. In the **Slice** tabsheet, right-click the outer surface and select **Create plate** from the appearing menu (**Figure 73** (on page 100)).
14. Click on the **Material** in the lower part of the explorer. Create a new material dataset. Specify the material parameters for the TBM according to **Table 16** (on page 97).
Figure 71: General tabsheet of the Tunnel designer

Figure 72: The Cross section tabsheet of the Tunnel designer
7.4.2 Surface contraction

A soil-structure interaction has to be added on the outside of the tunnel due to the slight cone shape of the TBM. Typically, the cross-sectional area at the tail of the TBM is about 0.5% smaller than the front of the TBM. The reduction of the diameter is realised over the first 7.5 m length of the TBM while the last 1.5 m to the tail has a constant diameter. This means that the section tail has a uniform contraction of 0.5% and the remaining 5 sections have a linear contraction with a reference value \( C_{\text{ref}} = 0.5\% \) and an increment \( C_{\text{inc,axial}} = -0.0667\% \). The reference is set on the front surface of the excavated slice in the tunnel during the tunnel construction. This is done while setting the Sequencing steps. The \( C_{\text{inc,axial}} = -0.0667\%/\text{m} \) and remains the same in every step (1_1 to 1_5). For further information on Surface contraction refer to the Reference Manual.

1. Right-click the same outer surface and select Create negative interface from the appearing menu to create a negative surface around the entire tunnel.
2. Next step is to create Surface contraction for the tunnel. Right-click the outer surface and select Create surface contraction.
3. In the properties box, select the Axial increment option for the contraction distribution and define \( C_{\text{ref}} = 0.5\% \) and \( C_{\text{inc,axial}} = -0.0667\%/\text{m} \). The increment must be a negative number because the contraction decreases in the direction of the positive local 1-axis.

Note:

- A surface contraction of the tunnel contour of 0.5% corresponds approximately to a volume loss of 0.5% of the tunnel volume (applicable only for small values of surface contractions).
The entered value of contraction is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

7.4.3 Grout pressure

The surface load representing the grout pressure is constant during the building process. In the specifications of the tunnel boring process, it is given that the grout pressure should be -100 kN/m$^2$ at the top of the tunnel ($z = -4.75\text{m}$) and should increase with -20 kN/m$^2$/m depth. To define the grout pressure:

1. Right-click the outer surface and select Create surface load from the appearing menu to create a surface load around the entire tunnel.
2. In the properties box, select Perpendicular, vertical increment from the drop-down menu for Distribution.
3. Set the $\sigma_{n,\text{ref}}$ to -100 and $\sigma_{n,\text{inc}}$ to -20 and define (0 0 -4.75) as the reference point for the load by assigning the values to $x_{\text{ref}}, y_{\text{ref}}$ and $z_{\text{ref}}$ (Figure 74 (on page 101))

![Figure 74: Slice tabsheet in the Tunnel designer](image)

7.4.4 Tunnel face pressures

The tunnel face pressure is a bentonite pressure (Bentonite Slurry, BS) or an earth pressure (Earth Pressure Balance, EPB) that increases linearly with depth. For the initial position of the TBM and the successive four positions when simulating the advancement of the TBM, a tunnel face pressure has to be defined.

1. Select the Plane tabsheet above the displayed tunnel cross section.
2. Multi-click both the surfaces, right-click and select **Create surface load** from the appearing menu to create a surface load around the entire tunnel.

3. In the properties box, select **Perpendicular, vertical increment** from the drop-down menu for **Distribution**.

4. Set the $\sigma_{n,\text{ref}}$ to -90 and $\sigma_{n,\text{inc}}$ to -14 and define (0 0 -4.75) as the reference point for the load by assigning the values to $x_{\text{ref}}, y_{\text{ref}}$ and $z_{\text{ref}}$ (Figure 75 (on page 102))

![Figure 75: Plane tabsheet in the Tunnel designer](image)

### 7.4.5 Jack forces

In order to move forward during the boring process, the TBM has to push itself against the existing tunnel lining. This is done by hydraulic jacks. The force applied by the jacks on the final tunnel lining has to be taken into account. This will be assigned to the tunnel lining in **Sequencing** tab.

### 7.4.6 Trajectory

The next step is to create the path of the boring process. The TBM already advanced 25m into the soil and then proceeds from 25m to 41.5m excavating slices of 1.5m each:

1. Click the **Trajectory** tab to proceed to the corresponding tabsheet.

2. In the **Segments** tabsheet, click on the **Add segment** on the left toolbar.

3. In the properties box set the length to 25.

4. Add the next segment and set the length to 16.5.

5. To create the slices, proceed to the **Slices** tabsheet.
6. Click on the second created segment. In the properties box, select *Length* as the **Slicing method** and set the **Slice length** as 1.5m (Figure 76 (on page 103)).

![Figure 76: Trajectory tabsheet in the Tunnel designer](image)

### 7.4.7 Sequencing

In order to simplify the definition of the phases in **Staged construction** mode, the sequencing of the tunnel is defined. The soil in front of the TBM will be excavated, a support pressure will be applied to the tunnel face, the TBM shield will be activated and the conicity of the shield will be modelled, at the back of the TBM the pressure due to the back fill grouting will be modelled as well as the force the hydraulic jacks driving the TBM exert on the already installed lining, and a new lining ring will be installed.

1. Click the **Sequencing** tab to proceed to the corresponding tabsheet.
2. In the **Sequencing** tabsheet, set the **Excavation method** as **TBM**.
   
   a. **<Step_1_1, face excavation>**

   Select the **Slice** tabsheet above the displayed tunnel cross section and select the volumes inside the tunnel. In the **Selection explorer**, deactivate the soil and set the **WaterConditions** to **Dry**.

   In the **Slice** tabsheet as well, select the outer surface. In the **Selection explorer**, activate the negative interface, the plate and the surface contraction (Figure 77 (on page 104)).

   Set $C_{reg} = 0\%$ for the surface contraction (since this is on the front of the excavation).

   Go to the **Plane Front** tabsheet and select all the surfaces. Activate the surface load corresponding to the face pressure (Figure 78 (on page 104))
Phased excavation of a shield tunnel
Definition of structural elements

Figure 77: Slice tabsheet in the **Tunnel designer** for Step_1_1

Figure 78: Plane Front tabsheet in the **Tunnel designer** for Step_1_1

b. **<Step_1_2, TBM with conicity>**

Click on the **Add Step** to add a new step. The difference with the front of the TBM is only the face pressure.

or
Go to the **Plane Front** tabsheet and select all the surfaces. In the **Selection explorer**, the surface load corresponding to the face pressure is deactivated by default.

or

Go to the **Slice** tabsheet, select the outer surface and set \( C_{\text{ref}} = 0.1\% \), see Figure 79 (on page 105).

![Figure 79: Slice tabsheet in the Tunnel designer for Step_1_2](image)

c.  🌐 <Step_1_3 to Step_1_5, TBM with conicity>

Click on the **Add Step** button three times to add three new steps. These steps are necessary to define the remaining cone part of the TBM shield (Figure 80 (on page 106)).

or

For each step go to the **Slice** tabsheet, select the outer surface and set the following values for the surface contraction in the **Selection explorer**:

Step_1_3: \( C_{\text{ref}} = 0.2\% \)

Step_1_4: \( C_{\text{ref}} = 0.3\% \)

Step_1_5: \( C_{\text{ref}} = 0.4\% \), see Figure 80 (on page 106)
d. <Step_1_6, tail of the shield>

The last slice of the shield has a constant diameter. From the Slice tabsheet select the outer surface and select the surface contraction.

or

In the Selection explorer select the Uniform option with $C_{ref} = 0.5\%$ (Figure 81 (on page 106)).
e. **<Step_1_7, grouting and jack thrusting>**

Select the **Slice** tabsheet and select the outer surface.

or

Deactivate the negative interface, the plate and the surface contraction.

or

In the **Selection explorer**, activate the surface load corresponding to the grout pressure (Figure 82 (on page 107)).

Select the **Plane rear** tabsheet and select the outer surface to define the jack thrusting against the final lining.

or

In the **Selection explorer**, activate the surface load and select the **Perpendicular** option for the distribution with $\sigma_{n,\text{ref}} = 635.4 \text{kN/m}^2$ (Figure 83 (on page 108))

---

**Figure 82: Slice tabsheet in the Tunnel designer Step_1_7**
f. **Step 1.8, final lining**

Select the Slice tabsheet and select the outer surface.

or

In the Selection explorer, deactivate the grout pressure and activate the negative interface.

or

In the Slice tabsheet again, select the outer volume. Activate it, click the material and select the Concrete option from the drop-down menu (Figure 84 (on page 109)).

Select the Plane rear tabsheet and select the outer surface.

or

In the Selection explorer, deactivate the surface load corresponding to the thrusting jacks (Figure 84 (on page 109))
Figure 84: Slice tabsheet in the **Tunnel designer** Step_1_8

Figure 85: Plane rear tabsheet in the **Tunnel designer** Step_1_8

**Note:** For the steps Step1_1 to Step1_5 keep in mind that the contraction increment $C_{inc,axial}$ has to be -0.0667%.
3. Click on Generate to include the defined tunnel in the model.
4. To create the slices, proceed to the Slices tabsheet.
5. Close the Tunnel designer window.

This concludes the model creation in Structures mode (Figure 86 (on page 110)).

7.5 Generate the mesh

In the Mesh mode it is possible to specify global and local refinements and generate the mesh. The default local refinements are valid for this example.

1. Click the Generate mesh button in order to generate the mesh. The Mesh options window appears.

   The default option (Medium) will be used to generate the mesh.

2. Click the View mesh button to inspect the generated mesh(Figure 87 (on page 111)).
Define and perform the calculation

The excavation of the soil and the construction of the tunnel lining will be modelled in the Staged construction mode. Since water levels will remain constant the Flow conditions mode can be skipped. It should be noted that due to the mesh generation the tunnel effectively has been split into an upper part, located in the clay, and a lower part located in the stiff sand. As a result, both the lower and the upper part of the tunnel should be considered.

The first phase differs from the following phases, as in this phase the tunnel is activated for the first time. This phase will model a tunnel that has already advanced 25 m into the soil. Subsequent phases will model an advancement by 1.5 m each.

Initial phase

The initial phase consists of the generation of the initial stresses using the $K_0$ procedure. The default settings for the initial phase are valid.
7.6.2 Phase 1: Initial position of the TBM

In the first phase, it is assumed that the TBM has already advanced 25 m. The section next to the first 25 m (section 25 m - 26.5 m), will represent the area directly behind the TBM were grout is injected in the tail void. In the next 6 sections (26.5 m - 35.5 m) the TBM will be modelled.

1. Add the first calculation phase

2. In the Model explorer expand Tunnels and then expand Tunnel_1. Scroll down the Model explorer until the option Advancement step and set it to 7 in order to simulate the advancement of the first 25 m. The final lining will be activated in the following phase. In order to consider the conicity of the TBM in the first 25 m, the clusters representing final lining need to be deactivated, the plates representing the TBM are activated and 0.5% contraction is applied.

3. Select the right view to reorientate the model in order to obtain a clearer view of the inside of the tunnel.

4. In the drawing area select the soil volumes corresponding to the lining in the first 25 m (Figure 88 (on page 112))

![Figure 88: Selection of soil volumes (0 m - 25 m)](image)

5. In the Selection explorer deactivate the soil. The soil is switched off, but the wireframe representing the deactivated soil is still coloured red as the deactivated soil is still selected.

   **Note:** An object that is deactivated will automatically be hidden as a volume or surface, but a wireframe representing the hidden object will remain. The visibility of the object not active in a calculation phase can be defined in the corresponding tabsheet of the Visualization settings window (SECTION of the REFERENCE MANUAL)

6. The interface is already activated. To activate the plate and the contraction in the first 25 m of the tunnel:
   a. Select the Select plates option in the appearing menu. Select the surfaces between 0 m and 25 m in the model to which plates are assigned (Figure 89 (on page 113))
   b. In the Selection explorer activate plate and uniform contraction by checking the corresponding boxes.
   c. In the drawing area select the lateral surfaces of the outer volume, corresponding to the last slice of the TBM (grout and jack thrusting) at 25.0 m (Figure 90 (on page 113)). In the Selection explorer, deactivate the surface load corresponding to the jack thrusting, because the TBM is only placed in this phase and it’s not moving.
7. 

Click the **Preview** button to get a preview of everything that has been defined (Figure 91 (on page 113)). Make sure that both grout pressure and tunnel face pressure are applied and that both increase from top to bottom.
7.6.3 Phase 2: TBM advancement 1

In this phase, the advancement of the TBM by 1.5 m (from $y = 35.5$ to $y = 37$) will be modelled.

![Model explorer (Phase 5)](image)

*Figure 92: The Advance to next tunnel step option from Model explorer*

1. Add a new phase
2. In the **Model explorer** expand **Tunnels** and then expand **Tunnel_1**. Scroll down the **Model explorer** until the option **Advancement step** and set it to 8 in order to simulate the advancement of the first 26.5 m.

7.6.4 Phase 3: TBM advancement 2

In this phase, the TBM advances from $y = 37$ to $y = 38.5$.

1. Add a new phase
2. In the **Model explorer** expand **Tunnels** and right-click **Tunnel_1**. Then click on **Advance to next tunnel step**.

7.6.5 Phase 4: TBM advancement 3
In this phase, the TBM advances from \( y = 38.5 \) to \( y = 40 \).

1. Add a new phase
2. In the Model explorer expand Tunnels and right-click Tunnel_1. Then click on Advance to next tunnel step.

### 7.6.6 Phase 5: TBM advancement 4

In this phase, the final advancement of the TBM is modelled (from \( y = 40 \) to \( y = 41.5 \)).

1. Add a new phase
2. In the Model explorer expand Tunnels and right-click Tunnel_1. Then click on Advance to next tunnel step.
3. Press the Calculate button to start the calculation. Ignore the message "No nodes or stress points selected for curves" as any load-displacement curves are drawn in this example, and start the calculation.

### 7.7 Results

Once the calculation has been completed, the results can be evaluated in the Output program. In the Output program the displacement and stresses are shown in the full 3D model, but the computational results are also available in tabular form. To view the results for the current analysis, follow these steps:

1. Select the last calculation phase (Phase 5) in the Phases explorer
2. Click the View calculation results button in the side toolbar to open the Output program. The Output program will by default show the 3D deformed mesh at the end of the selected calculation phase.
3. From the Deformations menu, select Total displacements and then \( u_z \) in order to see the total vertical displacements in the model as a shaded plot (Figure 93 (on page 116))

In order to see the settlements at ground level make a horizontal cross section by choosing the Horizontal cross section button. In the window that appears fill in a cross section height of 1.95 m. The window with the cross section opens (Figure 94 (on page 116)). The maximum settlement at ground level is about 1.9 cm.
Phased excavation of a shield tunnel

Results

Figure 93: Total vertical displacements after the final phase $u \approx 3.1\text{cm}$

Figure 94: Settlement trough at ground level $|u| \approx 1.9\text{cm}$
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. The dam consists of a clay core with a well graded fill at both sides. The subsoil consists of overconsolidated silty sand.

**Objectives**
- Performing fully coupled flow deformation analysis
- Defining time-dependent hydraulic conditions
- Using unsaturated flow parameters

**Geometry**

The dam to be considered is 30 m high. The top width and the base width of the dam are 5 m and 172.5 m respectively. The geometry of the dam is depicted below. The normal water level behind the dam is 25 m high. A situation is considered where the water level drops 20 m. The normal phreatic level at the right hand side of the dam is 10 m below ground surface.

8.1 Define the geometry

Assuming the dam is located in a wide valley, a representative length of 50 m is considered in the model in order to decrease the model size.

To create the geometry model, follow these steps:
1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.
2. In the **Project properties** window, enter an appropriate title.
3. Keep the default units and set the model dimensions to:
   a. \( x_{\text{min}} = -130.0 \text{ m}, x_{\text{max}} = 130.0 \text{ m} \)
   b. \( y_{\text{min}} = 0 \text{ m} \) and \( y_{\text{max}} = 50.0 \text{ m} \)

![Figure 96: The geometry of the model](image)

### 8.2 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. Create a borehole at \((0.0 \ 0.0)\).
   The **Modify soil layers** window pops up.
2. Add a soil layer extending from ground surface \((z = 0)\) to a depth of 30 m \((z = -30)\).
3. Set the **Head** in the borehole to \(-10 \text{ m}\). A horizontal water level will be automatically generated. This water level in combination with surface groundwater flow boundary conditions will be used in the **Fully coupled flow deformation** analyses.

### 8.3 Create and assign material data sets

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

The layers have the following properties:

**Table 17: Material properties of the dam and subsoil**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Core</th>
<th>Fill</th>
<th>Subsoil</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Mohr-Coulomb</td>
<td>Mohr-Coulomb</td>
<td>Mohr-Coulomb</td>
<td>-</td>
</tr>
</tbody>
</table>
## Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Core</th>
<th>Fill</th>
<th>Subsoil</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draining type</td>
<td>Type</td>
<td>Undrained (B)</td>
<td>Drained</td>
<td>Drained</td>
<td>-</td>
</tr>
<tr>
<td>Soil unit weight above phreatic level</td>
<td>$\gamma_{\text{unsat}}$</td>
<td>16.0</td>
<td>16.0</td>
<td>17.0</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>Soil unit weight below phreatic level</td>
<td>$\gamma_{\text{sat}}$</td>
<td>18.0</td>
<td>20.0</td>
<td>21.0</td>
<td>kN/m$^3$</td>
</tr>
</tbody>
</table>

### Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Core</th>
<th>Fill</th>
<th>Subsoil</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>E'</td>
<td>1.5·10$^3$</td>
<td>2.0·10$^4$</td>
<td>5.0·10$^4$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu_{ur}$</td>
<td>0.35</td>
<td>0.33</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>$c_{\text{ref}}'$</td>
<td>-</td>
<td>5.0</td>
<td>1.0</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Undrained shear strength</td>
<td>$s_{u,\text{ref}}$</td>
<td>5.0</td>
<td>-</td>
<td>-</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\phi'$</td>
<td>-</td>
<td>31</td>
<td>35.0</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>-</td>
<td>1.0</td>
<td>5.0</td>
<td>°</td>
</tr>
<tr>
<td>Young's modulus inc.</td>
<td>$E_{\text{inc}}'$</td>
<td>300</td>
<td>-</td>
<td>-</td>
<td>kN/m$^2$/m</td>
</tr>
<tr>
<td>Reference level</td>
<td>$z_{\text{ref}}$</td>
<td>30</td>
<td>-</td>
<td>-</td>
<td>m</td>
</tr>
<tr>
<td>Undrained shear strength inc.</td>
<td>$s_{u,\text{inc}}$</td>
<td>3.0</td>
<td>-</td>
<td>-</td>
<td>kN/m$^2$</td>
</tr>
</tbody>
</table>

### Groundwater

<table>
<thead>
<tr>
<th>Flow data set</th>
<th>Model</th>
<th>Hypres</th>
<th>Hypres</th>
<th>Hypres</th>
<th>-</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>-</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>Van Genuchten</td>
<td>-</td>
</tr>
<tr>
<td>Soil</td>
<td>-</td>
<td>Subsoil</td>
<td>Subsoil</td>
<td>Subsoil</td>
<td>-</td>
</tr>
<tr>
<td>Soil coarseness</td>
<td>-</td>
<td>Very fine</td>
<td>Coarse</td>
<td>Coarse</td>
<td>-</td>
</tr>
<tr>
<td>Permeability in horizontal direction</td>
<td>$k_x$</td>
<td>1.0·10$^{-4}$</td>
<td>0.25</td>
<td>0.01</td>
<td>m/day</td>
</tr>
<tr>
<td>Permeability in horizontal direction</td>
<td>$k_y$</td>
<td>1.0·10$^{-4}$</td>
<td>0.25</td>
<td>0.01</td>
<td>m/day</td>
</tr>
</tbody>
</table>
Rapid drawdown analysis

Define the dam

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Core</th>
<th>Fill</th>
<th>Subsoil</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Permeability in vertical direction</td>
<td>$k_z$</td>
<td>$1.0 \cdot 10^{-4}$</td>
<td>0.25</td>
<td>0.01</td>
<td>m/day</td>
</tr>
</tbody>
</table>

To create the material sets, follow these steps:

1. Open the **Material sets** window.
2. Create data sets under **Soil and interfaces** set type according to the information given in Table 17 (on page 118). Note that the **Interfaces** and **Initial** tabsheets are not relevant (no interfaces or **K0 procedure** used).
3. Assign the **Subsoil** material dataset to the soil layer in the borehole.

8.4 Define the dam

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

The surface groundwater boundary conditions are defined in the table below:

**Table 18: Surface groundwater flow boundary conditions**

<table>
<thead>
<tr>
<th>Surface</th>
<th>Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(-130 0 0), (-80 0 0), (-80 50 0), (-130 50 0)</td>
</tr>
<tr>
<td>2</td>
<td>(-80 0 0), (-2.5 0 30), (-2.5 50 30), (-80 50 0)</td>
</tr>
<tr>
<td>3</td>
<td>(-130 0 0), (-130 0 -30), (-130 50 -30), (-130 50 0)</td>
</tr>
</tbody>
</table>

1. Define a surface by specifying points located at (-80 0 0), (92.5 0 0), (2.5 0 30) and (-2.5 0 30).
2. Define a surface by specifying points located at (-10 0 0), (10 0 0), (2.5 0 30) and (-2.5 0 30).
3. Multi-select the created surfaces and right-click on the drawing area. Select the **Intersect and recluster** menu item.
4. Multi-select the surfaces and extrude along (0 50.0 0) The volumes representing the dam are generated.
5. Delete the surfaces used to create the soil volumes.
6. Assign the corresponding material data sets to the soil volumes.
7. Time dependent conditions can be assigned to surface groundwater flow boundary conditions. Define surface groundwater flow boundary conditions (under the **Create hydraulic conditions** tool) according to the information in Table 18 (on page 120).

8.5 Generate the mesh
For the generation of the mesh it is advisable to set the Element distribution parameter to Fine.

To modify the global coarseness:

1. Click the Generate mesh button in the side toolbar. The Mesh options window is displayed.
2. Select the Fine option form the Element distribution drop-down:

![Mesh options window](image)

*Figure 97: Modification of the Global coarseness*

3. Click OK to close the Mesh options window and to generate the mesh.
4. Click the View mesh button in the side toolbar to preview the mesh. The resulting mesh is displayed:

![Generated mesh](image)

*Figure 98: The generated mesh*

### 8.6 Define and perform the calculation

In the calculation process the initial state (high reservoir), the rapid drawdown case, the slow drawdown case and finally the low water level case will be considered. A safety analysis will be performed for each of the cases.
Table 19: Water levels

<table>
<thead>
<tr>
<th>Level</th>
<th>Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>High reservoir</td>
<td>(-130 0 25), (-10 0 25), (93 0 -10), (130 0 -10), (130 50 -10), (93 50 -10), (-10 50 25), (-130 50 25)</td>
</tr>
<tr>
<td>Low reservoir</td>
<td>(-130 0 5), (-10 0 5), (93 0 -10), (130 0 -10), (130 50 -10), (93 50 -10), (-10 50 5), (-130 50 5)</td>
</tr>
</tbody>
</table>

1. Proceed to the **Flow conditions** mode.
2. Create water levels corresponding to the full reservoir and the low water level cases according to the information given in Table 19 (on page 122).
3. In the **Attributes library** of the **Model explorer** rename the created user water levels as **High_Reservoir** and **Low_Reservoir**.

**Note:**
No modifications, such as **Time dependency** is possible for **Borehole water levels** and non-horizontal **User water levels**.

### 8.6.1 Initial phase: High reservoir

1. Proceed to the **Staged construction** mode.
2. Double-click the initial phase in the **Phases explorer**.
3. In the **General** subtree of the **Phases** window rename the phase as **High reservoir**.
4. Select the **Gravity loading** option as **Calculation type**.
   Note that **Staged construction** is the only option available for **Loading type**.
5. Select the **Steady state groundwater flow** option as pore pressure calculation type.
6. Note that the options **Ignore undr. behaviour (A,B)** and **Ignore suction** are by default selected in the **Deformation control parameters** subtree. The default values will be used for the parameters in the **Numerical control parameters** and **Flow control parameters** subtrees.
7. Click **OK** to close the **Phases** window.
8. In the **Staged construction** mode activate the soil clusters representing the embankment.
9. In the **Model explorer** expand the **Model conditions** subtree.
10. In the **GroundwaterFlow** subtree set **BoundaryYMin**, **BoundaryYMax** and **BoundaryZMin** to **Closed**. The remaining boundaries should be **Open**.
11. In the Water subtree select the high reservoir water level (High_Reservoir) as GlobalWaterLevel.

8.6.2 Phase 1: Rapid drawdown

In the rapid drawdown phase the water level in the reservoir will be lowered from \( z = 25 \text{ m} \) to \( z = 5 \text{ m} \) in a period of 5 days. To define the function describing the fluctuation of the water level:

1. Add a new calculation 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).
4. Set the Calculation type to Fully coupled flow-deformation.
5. Set the Time interval to 5 days.
6. The Reset displacements to zero option is automatically selected in the Deformation control parameters subtree.
7. Click OK to close the Phases window.
8. Expand the Attributes library in the Model explorer.
9. Right-click on Flow functions and select the Edit option in the appearing menu.
Rapid drawdown analysis

The **Flow functions** window is displayed.

10. 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.
   a. Specify a proper name to the function for the rapid drawdown (e.g. *Rapid*).
   b. Select the **Linear** option from the **Signal** drop-down menu.
   c. Assign a value of -20 m to **Δ Head**, representing the amount of the head decrease.
   d. Specify a time interval of 5 days. A graph is displayed showing the defined function.

![Figure 100: The flow function for the rapid drawdown case](image)

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

11. Activate all the surface groundwater flow boundary conditions.

12. Multi-select the surface groundwater flow BCs in the drawing area.

13. In the **Selection explorer** select the **Head** option as behaviour. The distribution of the head is **Uniform**.
   Assign a value of 25 m to \( h_{\text{ref}} \).

14. Set the time dependency to **Time dependent** and select the **Rapid** option as **Head function**.
   Information related to the head function is displayed in the **Object explorers** as well.

15. In the **Water** subtree in the **Model explorer** select the BoreholeWaterLevel_1 option as **GlobalWaterLevel**.

8.6.3 Phase 2: Slow drawdown

In the slow drawdown phase the water level in the reservoir will be lowered from \( z = 25 \) m to \( z = 5 \) m in a period of 50 days. To define the function describing the fluctuation of the water level:

1. Select the initial phase (High reservoir) in the **Phases explorer**.
2. Add a new calculation phase.
3. In the 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).
5. Set the Calculation type to Fully coupled flow-deformation.
6. Set the Time interval option to 50 days.
7. The Reset displacements to zero option is automatically selected in the Deformation control parameters subtree.
8. Click OK to close the Phases window.
9. Create a new flow function following the steps previously described.
   a. Specify a proper name to the function for the slow drawdown (e.g. Slow).
   b. Select the Linear option from the Signal drop-down menu.
   c. Assign a value of -20 m to Δ Head, representing the amount of the head decrease.
   d. Specify a time interval of 50 days.

![Flow functions window](Figure 101: The flow function for the slow drawdown case)

10. Activate all the surface groundwater flow boundary conditions and multi-select them in the drawing area.
11. In the Selection explorer select the Head option as behaviour. The distribution of the head is Uniform. Assign a value of 25 m to \( h_{\text{ref}} \).
12. Set the time dependency to Time dependent and select the Slow option as Head function.
13. In the Water subtree in the Model explorer select the BoreholeWaterLevel_1 option as GlobalWaterLevel.
8.6.4 Phase 3: Low level

This phase considers the steady-state situation of a low reservoir level.

1. Select the initial phase (High reservoir) in the Phases explorer.
2. Add a new calculation phase.
3. In the Phases explorer double-click the newly added phase. The Phases window is displayed.
4. In the General subtree specify the name of the phase (ex: Low level).
5. The default calculation type (Plastic) is valid for this phase.
6. The default Pore pressure calculation type (Steady state groundwater flow) is valid for this phase.
7. In the Deformation control subtree, select Ignore und. behaviour (A,B) and make sure that the Reset displacements to zero is selected as well.
8. Click OK to close the Phases window.
9. The surface groundwater flow BCs should be deactivated in the Model explorer.
10. In the Water subtree select the low reservoir water level (Low_Reservoir) as GlobalWaterLevel.

8.6.5 Phase 4 to 7

In Phases 4 to 7, stability calculations are defined for the previous phases respectively.

1. Select Phase_1 in the Phases explorer.
2. Add a new calculation phase and proceed to the Phases window.
3. In the General subtree specify the name of the phase (ex: Rapid drawdown - Safety).
4. Set Calculation type to Safety. The Incremental multipliers option is valid as Loading type.
5. Select the Reset displacements to zero option in the Deformation control subtree.
6. In the Numerical control parameters subtree set the Max steps parameter to 50 for Phase 4.
7. Follow the same procedure for Phases 5 to 7.
8.6.6 Execute the calculation

1. In the Staged construction mode select a node at the crest (-2.5 25.0 30.0).
2. Start the calculation process. Ignore the calculation warnings.
3. Save the project when the calculation has finished.

8.7 Results

1. After the calculation is finished click the View the calculation results button.

   The Output window now shows the deformed mesh for the selected phase.
2. Select the menu item Stresses > Pore pressures > $P_{\text{water}}$.
3. Define a vertical cross section passing through (-130 15) and (130 15)

   **Note:**
   
   - Note that by default the legend is locked in cross section plots, meaning that the same layer distribution will be used if the cross section is relocated in the model or if the results are displayed for other phases.
   - The legend can be unlocked by clicking on the Lock icon under the legend. A 'free' legend is indicated by the Open lock icon.

The results of the four groundwater flow calculations in terms of pore pressure distribution are shown in the series of figures below. Four different situations were considered:

- The situation with a high (standard) reservoir level
- The situation after rapid drawdown of the reservoir level
- The situation after slow drawdown of the reservoir level
- The situation with a low reservoir level

**Figure 103:** Pore water pressure distribution for high reservoir level (Initial phase)

**Figure 104:** Pore water pressure distribution after rapid drawdown (Phase 1)
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 phases 1 to 4.

For the phases 1 and 2 change the legend settings to **Manual** and set the values as follows:

- Minimum value: -480
- Maximum value: 200
- Number of intervals: 18
In this tutorial attention is focused on the variation of the safety factor of the dam for the different situations. Therefore, the development of $\Sigma M_{sf}$ is plotted for the phases 4 to 7 as a function of the displacement of the dam crest point:

![Figure 107: Safety factors for different situations](image)

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 3D to effectively analyse such situations.
Dynamic analysis of a generator on an elastic foundation

In this tutorial the influence of a vibrating source on its surrounding soil is studied. To reduce the calculation time, only one-quarter of the overall geometry is modelled, using symmetry boundary conditions along the lines of symmetry. The physical damping due to the viscous effects is taken into consideration via Rayleigh damping. Also, due to radial wave propagation, 'geometric damping' can be significant in attenuating the vibration.

The modelling of the boundaries is one of the key points in the dynamic calculation. 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

- Performing a Dynamic calculation
- Defining dynamic boundary conditions (viscous)
- Defining dynamic loads by means of load multipliers
- 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. Oscillations caused by the generator are transmitted through the footing into the subsoil. These oscillations are simulated as a uniform harmonic loading, with a frequency of 10 Hz and amplitude of 10 kN/m². In addition to the weight of the footing, the weight of the generator is modelled as a uniformly distributed load of 8 kN/m².
9.1 Define the geometry

To create the geometry model, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.
2. In the **Project properties** window, enter an appropriate title.
3. Keep the default units and set the model dimensions to:
   a. $x_{\text{min}} = 0$ and $x_{\text{max}} = 20$
   b. $y_{\text{min}} = 0$ and $y_{\text{max}} = 20$

The model boundaries should be sufficiently far from the region of interest, to avoid disturbances due to possible reflections. Although special measures (absorbent boundaries) are adopted in order to avoid spurious reflections, there is always a small influence and it is still a good habit to put boundaries far away. In a dynamic analysis, model boundaries are generally taken further away than in a static analysis.
9.2 Define the soil stratigraphy

1. The subsoil consists of one layer with a depth of 10 m. The ground level is defined at \( z = 0 \).
2. Note that water conditions are not considered in this example and the hydraulic head is set at \( z = -10 \).

9.3 Create and assign material data sets

Create the material data set according to the table below and assign it to the soil layer.

**Table 20: Material properties**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Sandy clay</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>Linear elastic</td>
<td>-</td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td>Drained</td>
<td></td>
</tr>
<tr>
<td>Unit weight above phreatic level</td>
<td>( \gamma_{\text{unsat}} )</td>
<td>20.0</td>
<td>kN/m(^3)</td>
</tr>
<tr>
<td>Unit weight below phreatic level</td>
<td>( \gamma_{\text{sat}} )</td>
<td>20.0</td>
<td>kN/m(^3)</td>
</tr>
</tbody>
</table>
### 9.4 Definition of structural elements

The generator is defined in the **Structures** mode. The **Polycurve** feature is used to define the geometry.

1. Click the **Create polycurve** button in the side toolbar and click on (0 0 0) in the drawing area.
2. In the **General** tabsheet the default option for shape (**Free**) and the default orientation axes (x-axis, y-axis) are valid for this polycurve.
3. In the **Segments** tabsheet three segments are defined as given in the table below. The insertion point is located at (0 0 0).

<table>
<thead>
<tr>
<th>Segment</th>
<th>Segment 1</th>
<th>Segment 2</th>
<th>Segment 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segment type</td>
<td>Line</td>
<td>Arc</td>
<td>Line</td>
</tr>
<tr>
<td>Segment properties</td>
<td>Relative start angle = 0° Length = 0.5 m</td>
<td>Relative start angle = 90° Radius = 0.5 m Segment angle = 90° Length = 0.5 m</td>
<td></td>
</tr>
</tbody>
</table>

4. Right-click the polycurve and select the **Create surface** option from the appearing menu.
5. Right-click the created surface and select the **Create surface load** option in the appearing menu.
6. In the **Selection explorer**, the **Uniform** distribution is valid for the surface load. Assign (0 0 -8) to the pressure components.
9.4.1 Definition of dynamic multipliers

Dynamic loads are defined on the basis of input values of loads or prescribed displacements and corresponding time-dependent multipliers.

To create the multipliers of the dynamic load:

1. In the Model explorer expand the Attributes library subtree.
2. Right-click the Dynamic multipliers subtree and select the Edit option from the appearing menu.
   The Multipliers window pops up.
3. Click the Load multipliers tab.
4. Click Add button to introduce a multiplier for the loads.
5. Define a Harmonic signal with an Amplitude of 10, a Frequency of 10 Hz and a Phase of 0° as shown below:

![Image of Multipliers window]

Figure 110: Definition of a Harmonic multiplier

6. In the Selection explorer, under DynSurfaceLoad_1 specify the components of the load as (0 0 -1).
7. Click Multiplier \(_z\) in the dynamic load subtree and select the LoadMultiplier_1 option from the appearing menu.

Note:
The dynamic multipliers can be defined in the Geometry modes as well as in the Calculation modes.

9.5 Generate the mesh
1. Proceed to the **Mesh** mode.
2. Refine the surface corresponding to the generator by assigning a **Coarseness factor** of 0.125.
3. Click the **Generate mesh** button. The **Medium** option will be used for **Element distribution**.
4. View the generated mesh.

![Figure 111: The generated mesh](image)

**Note:** In all dynamic calculations, the user should pay special attention to the element size to decrease numerical dispersion of waves. It should be noted that large elements are not able to transmit high frequencies. The transmission of waves is governed by both wave speed and wave length. If dynamic input contains high frequencies, either high frequencies should be filtered out or a finer mesh should be used.

9.6 Define and perform the calculation

The calculation consists of 4 phases. The initial phase consists of the generation of the initial stresses using the **K0 procedure**. The second phase is a **Plastic** calculation where the static load is activated. The third phase is a **Dynamic** calculation where the effect of the functioning generator is considered. The fourth and final phase is a **Dynamic** calculation as well where the generator is turned off and the soil will vibrate freely.

9.6.1 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.

9.6.2 Phase 1

1. Add a new calculation phase (Phase_1). The default settings of the added phase will be used for this calculation phase.
2. In the Staged construction mode activate the static component of the surface load. Do not activate the dynamic load.

![Figure 112: Applied load in the Phase_1](image)

9.6.3 Phase 2

1. Add a new calculation phase (Phase_2).
2. In the General subtree in the Phases window, select the Dynamic option as calculation type.
3. Set the Time interval parameter to 0.5 s.
4. In the Deformation control parameters subtree in the Phases window select the Reset displacement to zero parameter. The default values of the remaining parameters will be used for this calculation phase.
5. In the **Numerical control parameters** subtree uncheck the **Use default iter parameters** checkbox, which allows you to change advanced settings and set the **Time step determination** to **Manual**.

6. Set the **Max steps** to 250.

7. In the **Staged construction** mode activate the dynamic component of the surface load. Note that the static component of the load is still active.

![Screen capture of the numerical control parameters](image)

*Figure 113: Applied load in the Phase 2*

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, Ymax and Zmin. The dynamic boundaries can be specified in the **Dynamics** subtree located under the **Model conditions** in the **Model explorer**.
9.6.4 Phase 3

1. Add a new calculation phase (Phase_3).
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 Numerical control parameters subtree uncheck the Use default iter parameters checkbox, which allows you to change advanced settings and set the Time step determination to Manual.
5. Set the Max steps to 250.
6. 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.

The following figure shows the Phases explorer of this tutorial.
9.6.5 Execute the calculation

1. 🔐 Select nodes located at the ground surface (ex: (1.4 0 0), (1.9 0 0), (3.6 0 0)) to consider in curves.
2. 🚀 Execute the calculation.
3. 📁 Save the project.

9.6.6 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 tab sheet click the box next to the Rayleigh α parameter.
   Note that the display of the General tab sheet has changed displaying the Single DOF equivalence box.
4. Set the value of the ξ parameter to 5% for both targets.
5. Set the frequency values to 9 and 11 for the Target 1 and Target 2 respectively.
6. Click on one of the definition cells of the Rayleigh parameters. The values of α and β are automatically calculated by the program.
7. Click OK to close the data base.
8. Check whether the phases are properly defined (according to the information given before) and start the calculation.

9.6.7 Results

The Curve generator feature is particularly useful for dynamic 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 x-axis and $u_x$ to the y-axis. Figure 117 (on page 142) shows the response of the pre-selected points at the surface of the structure. Even with no damping, the waves are dissipated thanks to the geometric damping.
Define and perform the calculation

Figure 117: Vertical displ.- time on the surface at different distances to the vibrating source (without damping)

Figure 118 (on page 142) shows the response of the pre-selected points at the surface of the structure with material damping. The vibration is seized when some time is elapsed after the removal of the dynamic component of the surface load (at t = 0.5 s). Comparing the results without damping with the results with damping, also the displacement amplitudes are lower.

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 119 (on page 143) shows the total accelerations in the soil at the end of phase 2 (t = 0.5 s).
Dynamic analysis of a generator on an elastic foundation

Define and perform the calculation

Figure 119: Total accelerations in the soil at the end of Phase 2 (with damping)
10 Free vibration and earthquake analysis of a building

This example demonstrates the natural frequency of a long 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 along the model boundary.

**Objectives**

- Performing a **Dynamic** calculation
- Defining dynamic boundary conditions (free-field and compliant base)
- Defining earthquakes by means of displacement multipliers
- Modelling of free vibration of structures
- Modelling of hysteretic behaviour by means of Hardening Soil model with small-strain stiffness
- Calculating the natural frequency by means of Fourier spectrum

**Geometry**

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including the basement. The total height from the ground level is $5 \times 3 \, \text{m} = 15 \, \text{m}$ 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 clay layer of 15 m depth underlayered by a deep sand layer. In the model, 25 m of the sand layer will be considered.

10.1 Define the geometry

The length of the building is much larger than its width and the earthquake is supposed to have a dominant effect across the width of the building. Taking these facts into consideration, a representative section of 3 m will be considered in the model in order to decrease the model size. To create the geometry follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.
2. In the **Project properties** window, enter an appropriate title.
3. Keep the default units and set the model dimensions to:
   - $x_{\text{min}} = -80$ and $x_{\text{max}} = 80$
   - $y_{\text{min}} = 0$ and $y_{\text{max}} = 3$
10.2 Define the soil stratigraphy

1. The subsoil consists of two layers. The **Upper clayey layer** lies between the ground level \((z = 0)\) and \(z = -15\).
2. The underlying **Lower sandy layer** lies to \(z = -40\).
3. Define the phreatic level by assigning a value of -15 to the **Head** in the borehole.

10.3 Create and assign material data sets

Two material data sets are needed for this tutorial. The properties and some details of the material model are explained here.

**Table 22: Material properties**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Upper clayey layer</th>
<th>Lower sandy layer</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>Model</td>
<td>HS small</td>
<td>HS small</td>
<td>-</td>
</tr>
<tr>
<td>Drainage type</td>
<td>Type</td>
<td>Drained</td>
<td>Drained</td>
<td>-</td>
</tr>
<tr>
<td>Soil unit weight above phreatic level</td>
<td>(\gamma_{\text{unsat}})</td>
<td>16</td>
<td>20</td>
<td>kN/m(^3)</td>
</tr>
<tr>
<td>Parameter</td>
<td>Name</td>
<td>Upper clayey layer</td>
<td>Lower sandy layer</td>
<td>Unit</td>
</tr>
<tr>
<td>----------------------------------------------------</td>
<td>------------</td>
<td>--------------------</td>
<td>-------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Soil unit weight above phreatic level</td>
<td>$\gamma_{\text{sat}}$</td>
<td>20</td>
<td>20</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>Parameters</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Secant stiffness in standard drained triaxial test</td>
<td>$E_{50}^{\text{ref}}$</td>
<td>$2.0 \cdot 10^4$</td>
<td>$3.0 \cdot 10^4$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Tangent stiffness for primary oedometer loading</td>
<td>$E_{oed}^{\text{ref}}$</td>
<td>$2.561 \cdot 10^4$</td>
<td>$3.601 \cdot 10^4$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Unloading / reloading stiffness</td>
<td>$E_{uur}^{\text{ref}}$</td>
<td>$9.484 \cdot 10^4$</td>
<td>$1.108 \cdot 10^5$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Power for stress-level dependency of stiffness</td>
<td>$m$</td>
<td>0.5</td>
<td>0.5</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>$c'_{\text{ref}}$</td>
<td>10</td>
<td>5</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\varphi'$</td>
<td>18.0</td>
<td>28.0</td>
<td>$^\circ$</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>0.0</td>
<td>0.0</td>
<td>$^\circ$</td>
</tr>
<tr>
<td>Shear strain at which $G_s = 0.722 G_0$</td>
<td>$\gamma_{0.7}$</td>
<td>$1.2 \cdot 10^{-4}$</td>
<td>$1.5 \cdot 10^{-4}$</td>
<td>-</td>
</tr>
<tr>
<td>Shear modulus at very small strains</td>
<td>$G_0^{\text{ref}}$</td>
<td>$2.7 \cdot 10^5$</td>
<td>$1.0 \cdot 10^5$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu'_{ur}$</td>
<td>0.2</td>
<td>0.2</td>
<td>-</td>
</tr>
</tbody>
</table>

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, $G_0^{\text{ref}}$, 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.
In the Hardening Soil model with small-strain stiffness, the tangent shear modulus is bounded by a lower limit, $G_{ur}$.

$$G_{ur} = \frac{E_{ur}}{2(1 + v_{ur})}$$

The values of $G_{ur}^{ref}$ for the **Upper clayey layer** and **Lower sandy layer** and the ratio to $G_0^{ref}$ are shown in Table 23 (on page 148). This ratio determines the maximum damping ratio that can be obtained.
Table 23: $G_u$ values and ratio to $G_0^{\text{ref}}$

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Upper clayey layer</th>
<th>Lower clayey layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G_u$</td>
<td>kN/m$^2$</td>
<td>39517</td>
<td>41167</td>
</tr>
<tr>
<td>$G_0^{\text{ref}}/G_u$</td>
<td>-</td>
<td>6.83</td>
<td>2.43</td>
</tr>
</tbody>
</table>

The two figures below show the damping ratio as a function of the shear strain for the material used in the model. For a more detailed description and elaboration from the modulus reduction curve to the damping curve can be found in the literature. See 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.

1. Create the material data set according to the table above. and
2. Assign it to the corresponding soil layers. The upper layer consists of mostly clayey soil and the lower one consists of sandy soil.
10.4 Definition of structural elements

The structural elements of the model are defined in the Structures mode.

10.4.1 Create a building

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including the basement. The total height from the ground level is $5 \times 3 \, \text{m} = 15 \, \text{m}$ and the basement is 2 m deep. A value of $5 \, \text{kN/m}^2$ is taken as the weight of the floors and the walls.

For the building two material data sets are needed, with the following material properties:

**Table 24: Material properties of the building (plate properties)**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Rest of building</th>
<th>Basement</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of behaviour</td>
<td>Type</td>
<td>Elastic; Isotropic</td>
<td>Elastic; Isotropic</td>
<td>-</td>
</tr>
<tr>
<td>Thickness</td>
<td>$d$</td>
<td>0.3</td>
<td>0.3</td>
<td>m</td>
</tr>
<tr>
<td>Material weight</td>
<td>$\gamma$</td>
<td>33.33</td>
<td>50</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>$E_1$</td>
<td>$3 \times 10^7$</td>
<td>$3 \times 10^7$</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu_{12}$</td>
<td>0</td>
<td>0</td>
<td>-</td>
</tr>
<tr>
<td>Rayleigh damping</td>
<td>$\alpha$</td>
<td>0.2320</td>
<td>0.2320</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$\beta$</td>
<td>$8 \times 10^{-3}$</td>
<td>$8 \times 10^{-3}$</td>
<td>-</td>
</tr>
</tbody>
</table>

To define the structure:

The central column of the structure is modelled using the Node-to-node anchor feature. To create the central column of the structure:

1. Define a surface passing through the points (-5 0 -2), (5 0 -2), (5 3 -2) and (-5 3 -2).
2. Create a copy of the surface by defining an 1D array in z-direction. Set the number of the columns to 2 and the distance between them to 2 m.
3. Select the created surface at $z = 0$ and define a 1D array in the z-direction. Set the number of the columns to 6 and the distance between consecutive columns to 3 m.
4. Define a surface passing through the points (5 0 -2), (5 3 -2), (5 3 15) and (5 0 15).

5. Create a copy of the vertical surface by defining an 1D array in x-direction. Set the number of the columns to 2 and the distance between them to -10 m.

6. Multiselect the vertical surfaces and the horizontal surface located at z = 0.

7. Right-click on the selection and select the **Intersect and recluster** option from the appearing menu. It is important to do the intersection in the **Structures mode** as different material data sets are to be assigned to the basement and the rest of the building.

8. Select all the created surfaces representing the building (basement, floors and walls), right-click and select the **Create plate** option from the appearing menu.

9. Define the material data set for the plates representing the structure according to **Table 24** on page 149.

10. Assign the **Basement** material data set to the horizontal plate located at z = -2 and the vertical plates located under the ground level.

11. Assign the corresponding material data set to the rest of the plates in the model.

12. In order to model the soil-structure intersection at the basement of the building assign interfaces to the outer side of the basement. Note that depending on the local coordinate system of the surfaces an interface either positive or negative is assigned.

13. Create a **Line** through points (0 1.5 -2) and (0 1.5 0) corresponding to the column in the basement floor.

14. Create a **Line** through points (0 1.5 0) and (0 1.5 3) corresponding to the column in the first floor.

15. Create a copy of the last defined line by defining an 1D array in z-direction. Set the number of the columns to 5 and the distance between them to 3 m.

16. Select the created lines, right-click and select the **Create node-to-node anchor** option from the appearing menu.

17. Create the material data set according to the table below and assign it to the anchors.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Column</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material type</td>
<td>Type</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness</td>
<td>EA</td>
<td>2.5·10^6</td>
<td>kN</td>
</tr>
</tbody>
</table>

**10.4.2 Create the loads**

A static lateral force of 10 kN/m is applied laterally at the top left corner of the building. To create the load:

The earthquake is modelled by imposing a prescribed displacement at the bottom boundary. To define the prescribed displacement:To define the dynamic multipliers for the prescribed displacement:
1. Create a line load passing through (-5 0 15) and (-5 3 15).
2. Specify the components of the load as (10 0 0).
3. Create a surface prescribed displacement passing through (-80 0 -40), (80 0 -40), (80 3 -40) and (-80 3 -40).
4. Specify the x-component of the prescribed displacement as Prescribed and assign a value of 1.0. The y and z components of the prescribed displacement are Fixed. The default distribution (Uniform) is valid.
5. In the Model explorer expand the Attributes library subtree. Right-click on Dynamic multipliers and select the Edit option from the appearing menu. The Multipliers window pops up displaying the Displacement multipliers tabsheet.
6. To add a multiplier click the corresponding button in the Multipliers window.
7. From the Signal drop-down menu select the Table option.
8. The file containing the earthquake data is available in the PLAXIS 3D Knowledge Base (tutorial Free vibration and earthquake analysis of a building).
9. Open the page in a web browser, copy all the data to a text editor (e.g. Notepad) and save the file in your computer with the extension *.smc. Alternatively this file can also be found in the Importables folder in the PLAXIS 3D directory.
10. In the Multipliers window click the Open button and select the saved file. In the Import data window select the Strong motion CD-ROM files option from the Parsing method drop-down menu and press OK to close the window.
11. Select the Acceleration option in the Data type drop-down menu.
12. Select the Drift correction options and click OK to finalize the definition of the multiplier. In the Dynamic multipliers window the table and the plot of the data is displayed.
13. In the **Model explorer** expand the **Surface displacements** subtree and assign the Multiplier $x$ to the $x$-component by selecting the option in the drop-down menu.

### 10.4.3 Create interfaces on the boundary

**Free-field** and **Compliant base** require the manual 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. Create a surface passing through $(-80 \ 3 \ 0)$, $(-80 \ 0 \ 0)$, $(-80 \ 0 \ -40)$ and $(-80 \ 3 \ -40)$. Right-click the created surface and and click **Create positive interface** to add an interface inside the model.

2. Create a surface passing through $(80 \ 3 \ 0)$, $(80 \ 0 \ 0)$, $(80 \ 0 \ -40)$ and $(80 \ 3 \ -40)$. Right-click the created surface and and click **Create negative interface** to add an interface inside the model.

3. The surface at the bottom of the model is already created by the prescribed displacement. Right-click the surface at the bottom of the model and click **Create positive interface** to add an interface inside the model.
10.5 Generate the mesh

1. Proceed to the **Mesh** mode.
2. Click the **Generate mesh** button. Set the element distribution to **Fine**.
3. View the generated mesh.

*Figure 126: The generated mesh*

10.6 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.

10.6.1 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. 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.
10.6.2 Phase 1

1. Add a new calculation phase (Phase_1). 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.

![Figure 127: Construction of the building](image)

10.6.3 Phase 2

1. Add a new calculation 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 line load. The value of the load is already defined in the Structures mode.

10.6.4 Phase 3

1. Add a new calculation phase (Phase_3).
2. In the Phases window select the Dynamic option as Calculation type.
3. Set the Time interval parameter to 5 sec.
4. In the Staged construction mode deactivate the line load.
5. In the Model explorer expand the Model conditions subtree.
6. Expand the **Dynamics** subtree. By default the boundary conditions in the x and y directions are set to viscous. Select the **None** option for the boundaries in the y direction. Set the boundary Zmin to viscous.

![Model explorer (Phase_3)](image)

**Figure 128: Boundary conditions for Dynamic calculations (Phase_3)**

**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.

### 10.6.5 Phase 4

1. ![Add a new phase](image)
   
   Add 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](image)
   
   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. The default values of the remaining parameters will be used in this calculation phase.

6. In the **Numerical control parameters** subtree uncheck the **Use default iter parameters** checkbox, which allows you to change advanced settings and set the **Time step determination** to **Manual**.

7. Set the **Max steps** to 1000 and the **Max number of sub steps** to 4.

8. In the **Model explorer** expand the **Model conditions** subtree.

9. Expand the **Dynamics** subtree. Set the **Free-field** option for the boundaries in the x direction. The boundaries in the y direction are already set to **None**. Set the boundary Zmin to **Compliant base**.

10. Make sure that the interfaces on the boundary of the model are not activated in the **Model explorer**.

11. In the **Model explorer** activate the **Surface displacement** and its dynamic component. Set the value of \( u_x \) 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 outcropping motion.

---

*Figure 129: Boundary conditions for Dynamic calculations (Phase 4)*
10.6.6 Execute the calculation

1. Select points for load displacement curves at (0 1.5 15), (0 1.5 6), (0 1.5 3) and (0 1.5 -2).
2. Execute the calculation.

10.7 Results

Figure 130 (on page 157) shows the deformed structure at the end of the Phase 2 (application of horizontal load).

Figure 130: Deformed mesh of the system at the end of Phase 2

Figure 131 (on page 158) shows the time history of displacements of the selected points A (0 1.5 15), B (0 1.5 6), C (0 1.5 3) and D (0 1.5 -2) 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.
In the **Chart** tabsheet of the **Settings** window select the **Use frequency representation (spectrum)** and **Use standard frequency (Hz)** options in the **Dynamics** box. The plot is shown in the figure below. From this figure it can be evaluated that the dominant building frequency is around 1 Hz. For a better visualisation of the results animations of the free vibration and earthquake can be created.

*Figure 131: Time history of displacements (Free vibration)*
The figure below shows the time history of displacements of the point A (0 1.5 15) for the earthquake 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.

The time history signature of the point A (0 1.5 15) of the earthquake phase has been transformed to normalised power spectra through Fast Fourier transform for Phase 4 and is plotted in the figure below.
Figure 134: Acceleration power spectra at (0 1.5 15)
Calculation scheme for initial stresses due to soil weight

A

Examples of non-horizontal surfaces, and non-horizontal weight stratifications are: