PLAXIS

PLAXIS 3D 2024.2

Tutorial Manual 3D

Bentley®
Advancing Infrastructure

Last Updated: May 07, 2024

Table of Contents

Preface: Introduction 6

Chapter 1: Foundation in overconsolidated clay 7

1.1 Case A: Rigid foundation 8
1.1.1 Create a new project 9
1.1.2 Define the soil stratigraphy 11
1.1.3 Create and assign material data sets 12
1.1.4 Define the structural elements 18
1.1.5 Generate the mesh 19
1.1.6 Define and perform the calculation 21
1.1.7 View the calculation results 24
1.2 Case B: Raft foundation 25
1.2.1 Create a new project 27
1.2.2 Create and assign a material data set 27
1.2.3 Define the structural elements 28
1.2.4 Generate the mesh 32
1.2.5 Define and perform the calculation 32
1.2.6 View the calculation results 34
1.3 Case C: Pile-Raft foundation 35
1.3.1 Create a new project 36
1.3.2 Define the structural elements: Foundation piles 36
1.3.3 Generate the mesh 38
1.3.4 Define and perform the calculation 39
1.3.5 View the calculation results 39

Chapter 2: Excavation in sand 42

2.1 Create a new project 43
2.2 Define the soil stratigraphy 44
2.3 Create and assign the material data sets 44
2.4 Define the structural elements 46
2.4.1 Walings and Struts 46
2.4.2 Ground anchors 48
2.4.3 Pile sheet walls and loads 50
2.5 Generate the mesh 52
2.6 Define the calculation 52
2.6.1 Execute the calculation 54
2.7 Results 55

Chapter 3: Loading of a suction pile 60

3.1 Create a new project 61
3.2 Define the soil stratigraphy 61
3.3 Create and assign the material data sets 61
3.4 Define the structural elements 63

3.4.1 Create a suction pile 63
3.4.2 Create helper objects for local mesh refinements 68
3.5 Generate the mesh 70
3.6 Define the calculation 70
3.6.1 Initial phase: Initial conditions 70
3.6.2 Phase 1: Installation of suction pile 70
3.6.3 Phase 2: Load Pile 30 degrees 71
3.6.4 Phase 3,4,5,6: Load Pile with different direction angles 71
3.6.5 Execute the calculation 72
3.7 Results 72

Chapter 4: Stability of a diaphragm wall excavation 74

4.1 Create a new project 75
4.2 Define the soil stratigraphy 75
4.3 Create and assign the material data sets 75
4.4 Definition of the diaphragm wall 76
4.5 Generate the mesh 78
4.6 Define the calculation 79
4.6.8 Execute the calculation 81
4.7 Results 81

Chapter 5: Phased excavation of a shield tunnel [GSE] 84

5.1 Create a new project 85
5.2 Define the soil stratigraphy 85
5.3 Create and assign the material data sets 86
5.4 Definition of structural elements 87
5.4.1 Create tunnel cross section 88
5.4.2 Definition of Tunnel properties 91
5.4.3 Trajectory 95
5.4.4 Sequencing 97
5.5 Generate the mesh 107
5.6 Define and perform the calculation 108
5.6.1 Initial phase 108
5.6.2 Phase 1: Initial position of the TBM 109
5.6.3 Phase 2: TBM advancement 1 113
5.6.4 Phase 3: TBM advancement 2 114
5.6.5 Phase 4: TBM advancement 3 114
5.6.6 Phase 5: TBM advancement 4 114
5.7 Results 114

Chapter 6: Construction of a road embankment [ADV] 117

6.1 Create a new project 117
6.2 Define the soil stratigraphy 118
6.3 Create and assign the material data sets 119
6.4 Definition of embankment and drains 121
6.5 Generate the mesh 123
6.6 Define the calculation 123
6.6.1 Initial phase 123
6.6.2 Consolidation analysis 124
6.6.3 Execute the calculation 126
6.7 Results 127

6.8 Safety analysis 131
6.9 Using drains 134

Chapter 7: Rapid drawdown analysis [ULT] 136

7.1 Create a new project 136
7.2 Define the soil stratigraphy 137
7.3 Create and assign material data sets 137
7.4 Define the dam 139
7.5 Generate the mesh 139
7.6 Define and perform the calculation 140
7.6.1 Initial phase: High reservoir 141
7.6.2 Phase 1: Rapid drawdown 142
7.6.3 Phase 2: Slow drawdown 144
7.6.4 Phase 3: Low level 145
7.6.5 Phase 4 to 7 146
7.6.6 Execute the calculation 146
7.7 Results 147

Chapter 8: Dynamics analysis of a generator on an elastic foundation [ULT] 151

8.1 Create a new project 152
8.2 Define the soil stratigraphy 153
8.3 Create and assign material data sets 153
8.4 Definition of structural elements 154
8.5 Generate the mesh 156
8.6 Define and perform the calculation 156
8.6.1 Initial phase 157
8.6.2 Phase 1: Footing 157
8.6.3 Phase 2: Start generator 158
8.6.4 Phase 3: Stop generator 159
8.6.5 Execute the calculation 160
8.6.6 Additional calculation with damping 160
8.6.7 Results 161

Chapter 9: Free vibration and earthquake analysis of a building [ULT] 164

9.1 Define the geometry 164
9.2 Define the soil stratigraphy 165
9.3 Create and assign material data sets 165
9.4 Definition of structural elements 169
9.4.1 Create a building 169
9.4.2 Create the loads 171
9.4.3 Create interfaces on the boundary 173
9.5 Generate the mesh 173
9.6 Define and perform the calculation 174
9.6.1 Initial phase 174
9.6.2 Phase 1 - Building construction 174
9.6.3 Phase 2 -Excitation 174
9.6.4 Phase 3 - Free Vibration 175
9.6.5 Phase 4 - Earthquake 176
9.6.6 Execute the calculation 179
9.7 Results 179

Appendices 184
Appendix A: Calculation scheme for initial stresses due to soil weight 185

Introduction

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 maybe found, depending on hardware and software configuration.

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

Tutorials available and licencing levels:

Given PLAXIS 3D features and soil models are provided for separated licencing services, the present tutorial manuals are available with previous installation of a specific licence level.

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

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

- Tutorials with no identification - generally available for PLAXIS 3D licence.
- [ADV] - tutorials for users with PLAXIS 3D Advanced licence.
- [ULT] - tutorials for users with PLAXIS 3D Ultimate licence.
- [GSE] - tutorials for users with Geotechnical SELECT subscription (previous Basic, Advanced or Ultimate licence level required).

1 Foundation in overconsolidated clay

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.

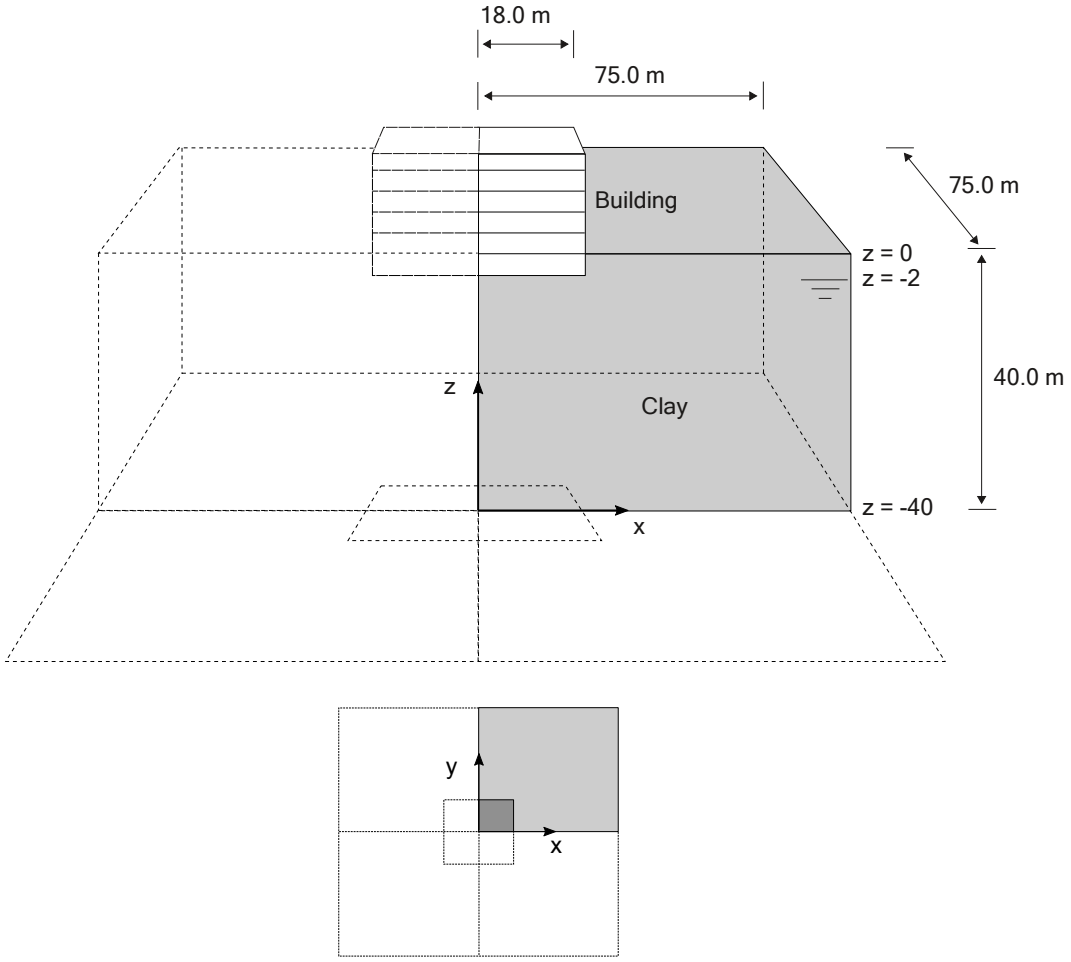


Figure 1: Geometry of a square building on a raft foundation

1.1 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.

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

1.1.1 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 (See Figure 2 (on page 9)).

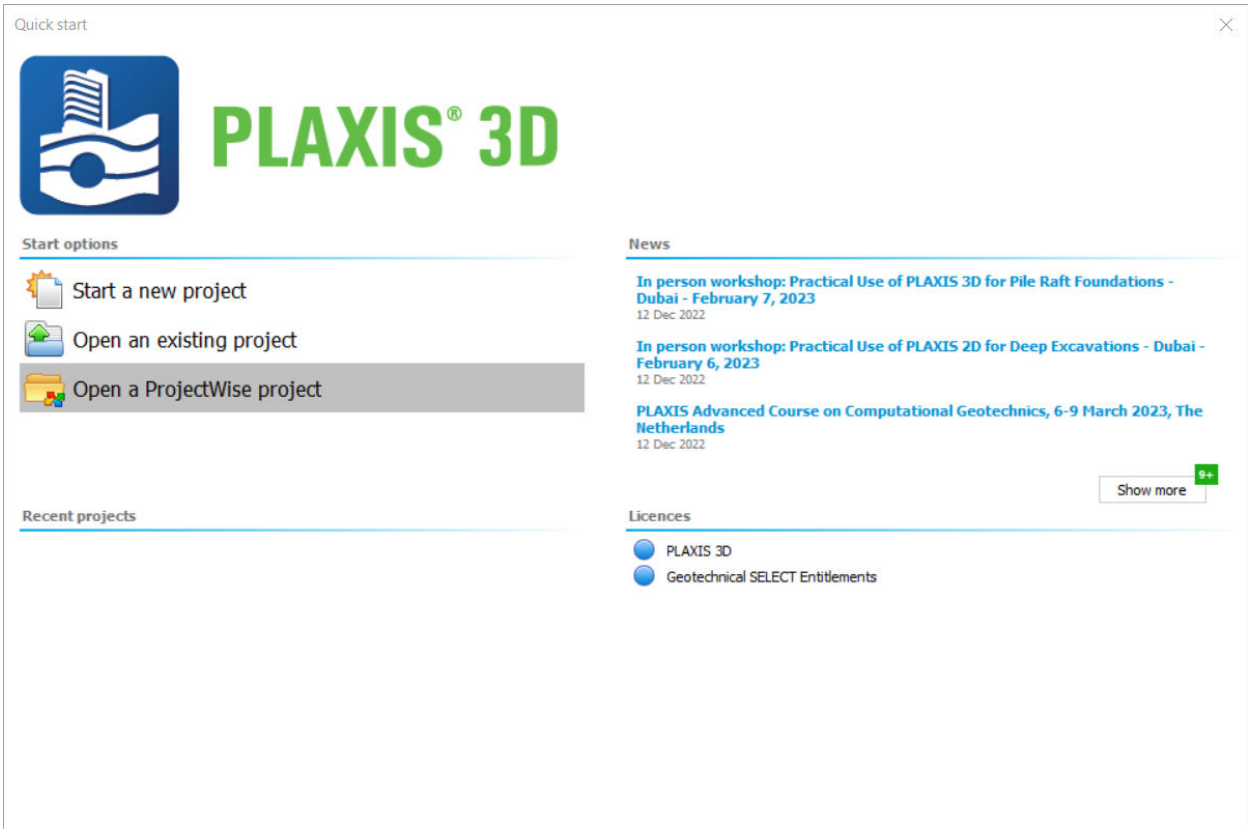


Figure 2: Quick Start window

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

The **Project properties** window (see Figure 3 (on page 10)) appears with the tabsheets: **Project**, **Model** and **Cloud services**.

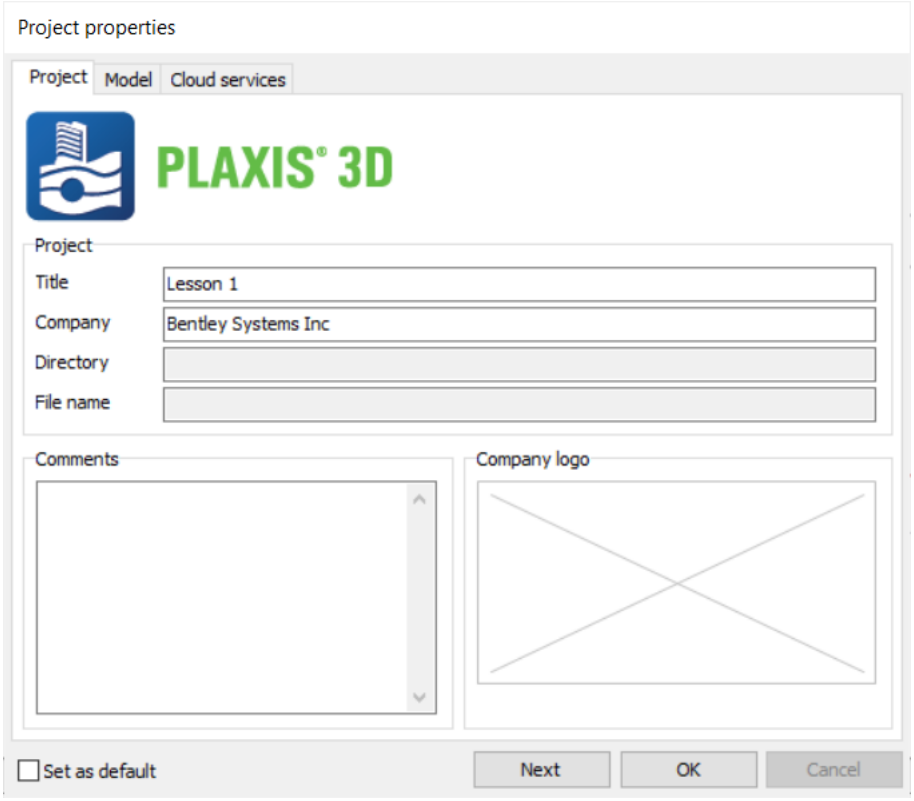


Figure 3: Project properties window

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 in Figure 4 (on page 11):

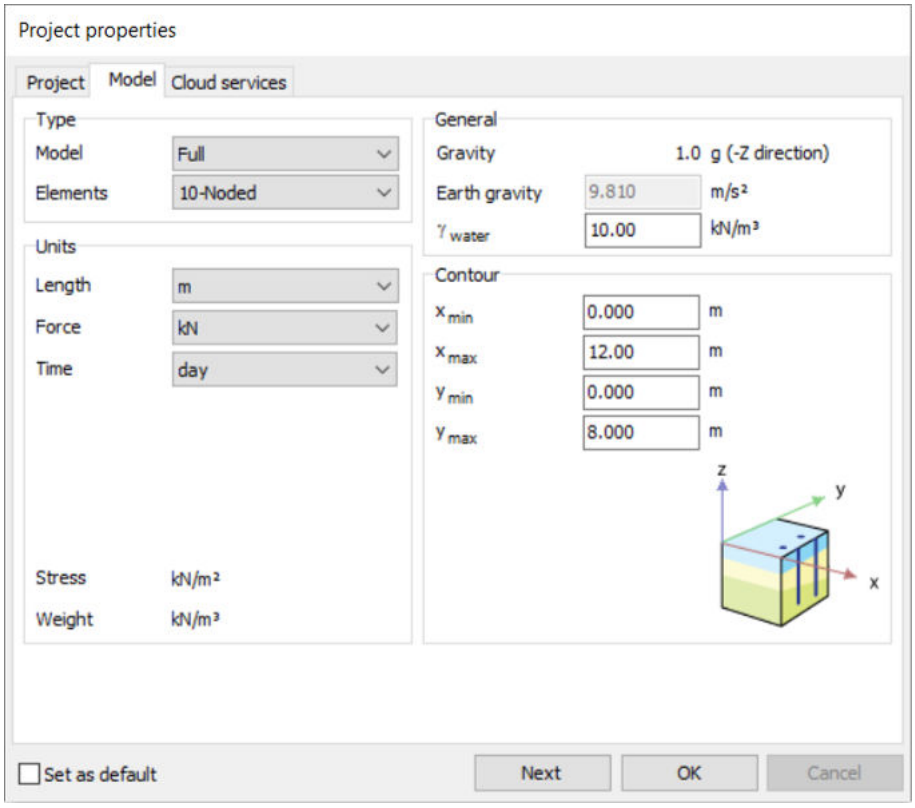


Figure 4: Project properties - Model tab

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_{water} box the unit weight of water can be defined. Keep this to the default value of 10 kN/m^3.
8. In the **Contour** group set the model dimensions to:
   a. x_{min} = 0.0 and x_{max} = 75.0,
   b. y_{min} = 0.0 and y_{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 ...**.

1.1.2 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 (see Figure 5 (on page 12)).
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.

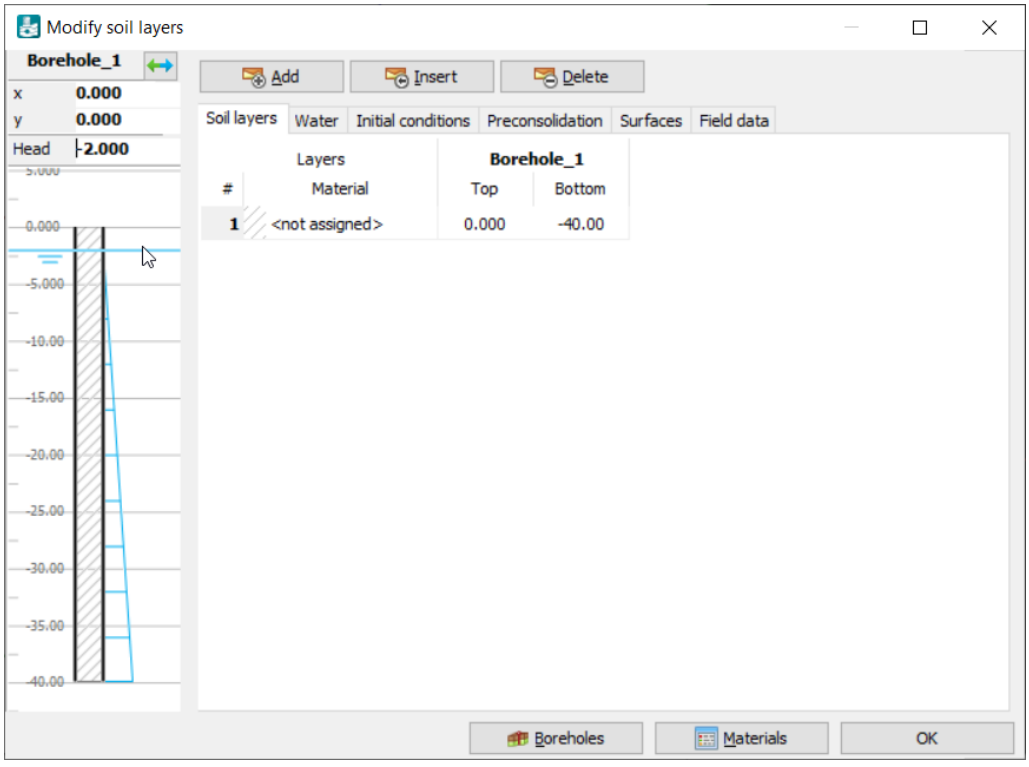


Figure 5: Modify soil layers window

Next, the material data sets are defined and assigned to the soil layers.

1.1.3 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**, **Discontinuities**, **Plates**, **Geogrids**, **Beams**, **Embedded beams** and **Anchors**.

The materials used in this tutorial are displayed in Table 1 (on page 13), and they are used as material data sets for **Soils and interfaces**.

Table 1: Material properties

Property	Name	Lacustrine clay	Building	Unit
General				
Soil Model	Model	Mohr-Coulomb	Linear Elastic model	-
Drainage type	Type	Drained	Non-porous	-
Unsaturated unit weight	γ_{unsat}	17.0	50	kN/m^3
Saturated unit weight	γ_{sat}	18.0	-	kN/m^3
Mechanical				
Young's modulus	E'_{ref}	$1{\cdot}10^4$	$3{\cdot}10^7$	kN/m^2
Poisson's ratio	$\nu(nu)$	0.3	0.15	-
Cohesion	c'_{ref}	10	-	kN/m^2
Friction angle	φ' *(phi)*	30.0	-	°
Dilatancy angle	ψ *(psi)*	0.0	-	°
Initial				
K_0	-	Automatic	Automatic	-
Lateral earth pressure coefficient	K_0	0.5000	0.5000	-

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 as displayed in Figure 6 (on page 14).

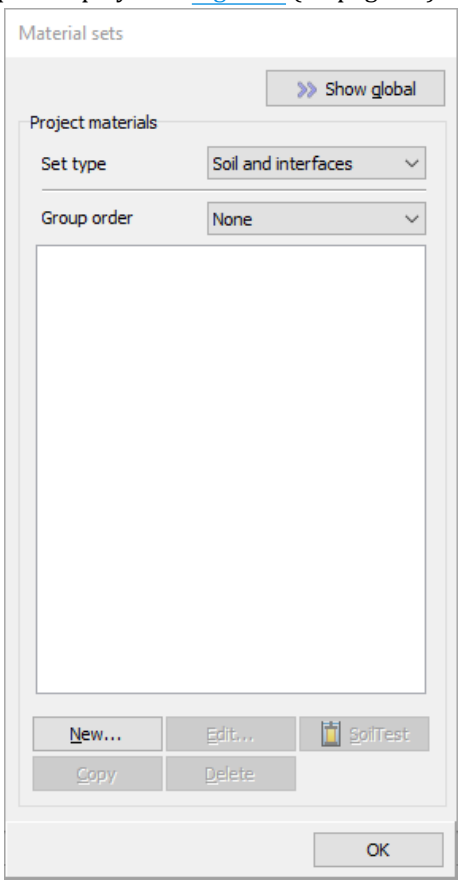


Figure 6: Material sets window

Note: Alternatively 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 (see Figure 7 (on page 15). It contains five tabsheets: **General**, **Mechanical**, **Groundwater**, **Interfaces** and **Initial**.
2. In the **General** tabsheet, **Material set** section, **Identification** field, type `Lacustrine Clay`.
3. Select Mohr-Coulomb model from the **Soil model** drop-down menu and **`Drained`** from the **Drainage type** drop-down menu.

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

4. Enter the unit weights in the **Unit weights** box according to the material data as listed in Table 1 (on page 13). Keep the unmentioned **Void ratio** parameters as their default values.

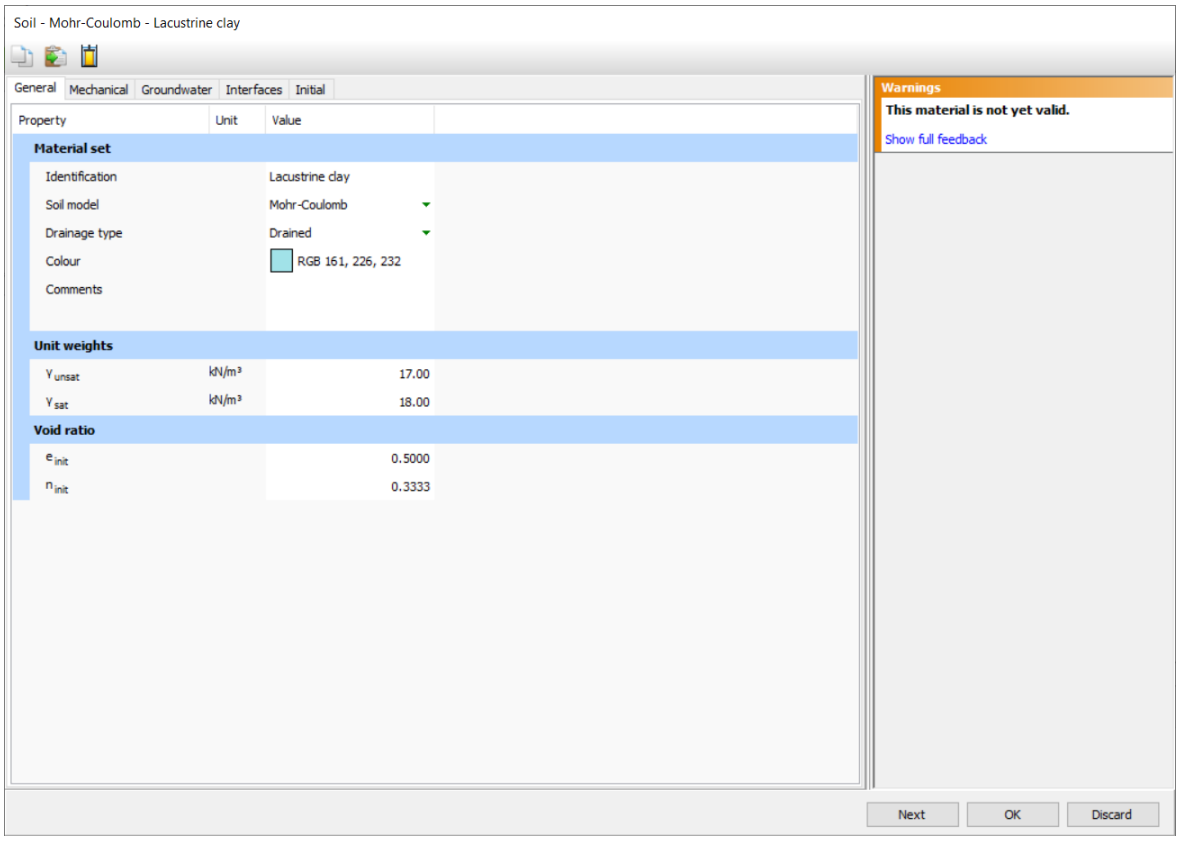


Figure 7: General tabsheet of the Soil and interfaces data set window

Note:

1. As displayed in Figure 7 (on page 15) a **Feedback side panel** is included in the **Soil** window. This panel prevents the definition of an invalid material data set. To display the list of detailed messages please select *Show full feedback*. Three types of messages are possible:
   - **Errors:** the parameter value or combination of parameter values must be changed, otherwise the material set could be invalid and calculation of the project will be blocked.
   - **Warnings:** the parameter value seems to deviate from a recommended parameter value or parameter range. Generally the material set will not be considered invalid and calculating the project will not be blocked. The chosen parameter could however cause unexpected results.
   - **Hints:** the entered parameter can be defined under certain circumstances or options.
2. The **Feedback side panel** is displayed at the moment of defining materials and structures. For the sake of simplicity, this panel will only be shown in some tutorial examples..

5. Click the **Next** button or click the **Mechanical** tab to proceed with the input of mechanical parameters. The parameters appearing on the **Mechanical** tabsheet depend on the selected material model (in this case the Mohr-Coulomb model). The Mohr-Coulomb model involves only five basic parameters (E'_{ref}, ν, c'_{ref}, φ', ψ).

Note: Consult the Material Models Manual for a detailed description of the different soil models and their corresponding parameters.

6. Enter the model parameters E'_{ref}, ν, c'_{ref}, φ' and ψ of **Lacustrine clay** according to Table 1 (on page 13) in the corresponding boxes of the **Mechanical** tabsheet.

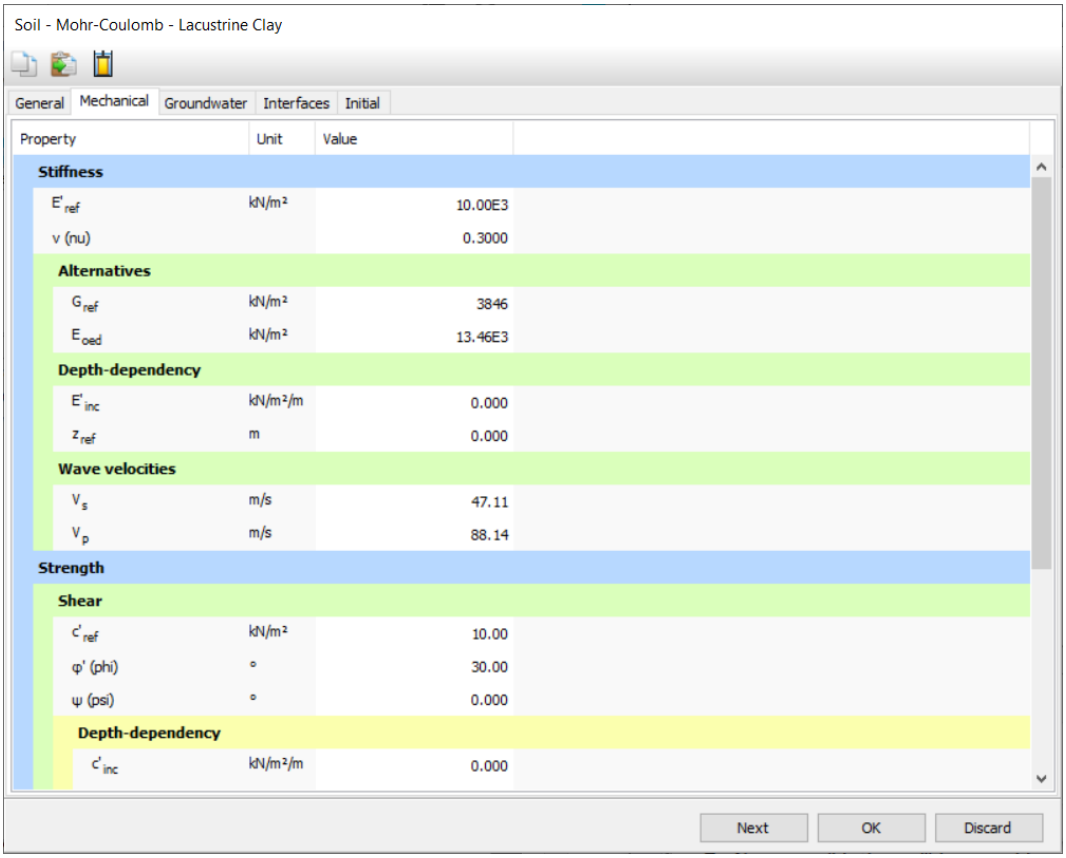


Figure 8: Mechanical tabsheet of the Soil and interfaces data set window

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 **K0 determination** is set to **`Automatic`**. In that case K_0 is determined from Jaky's formula: $K_0 = 1 - \sin\varphi$.
10. Click the **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 **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).

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 **New** button in the **Material sets** window.
2. In the **General** tabsheet, **Material set** section, **Identification** field, type `Building`.
3. Select Linear Elastic model from the **Material model** drop-down menu and **`Non-porous`** from the **Drainage type** drop-down menu as displayed in Figure 9 (on page 17).

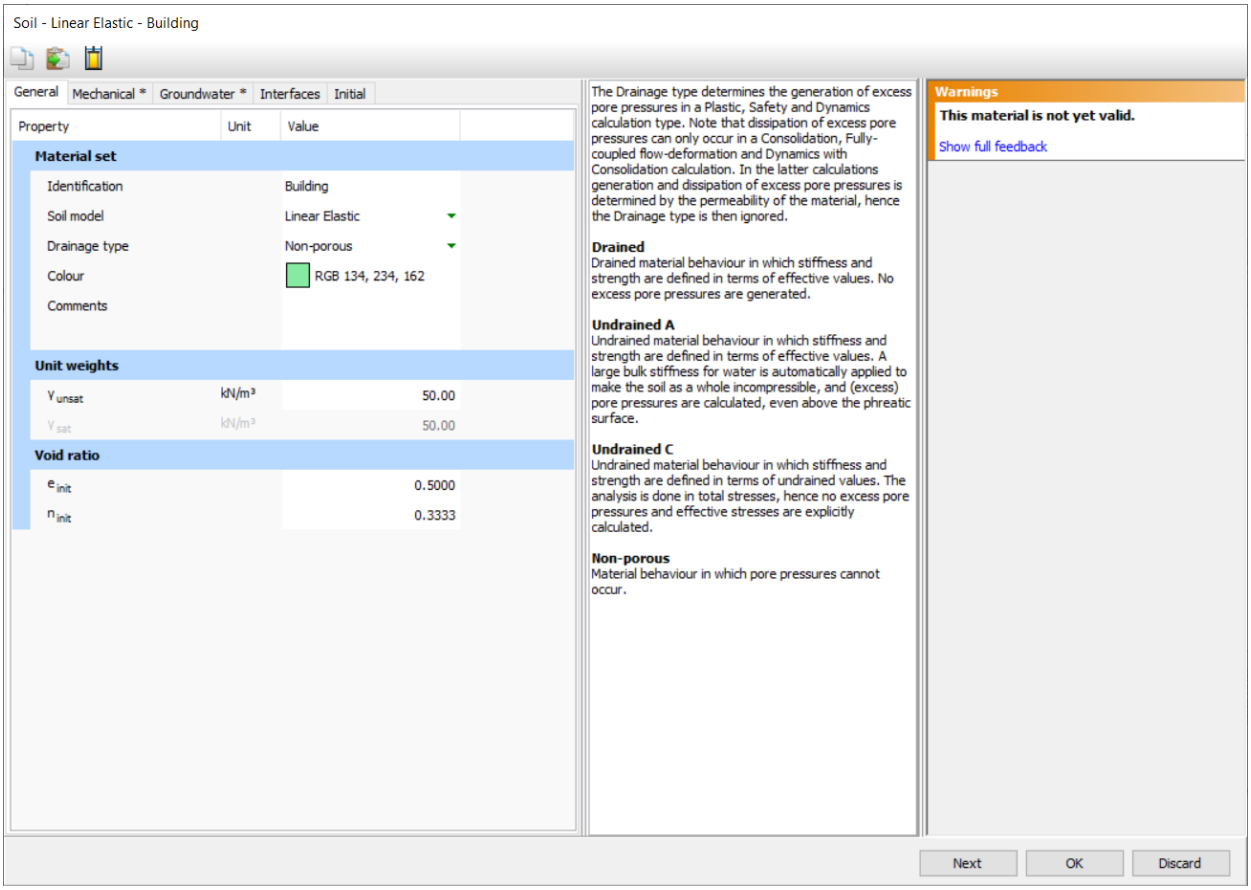


Figure 9: Drainage type - General tabsheet of the Soil and interfaces data set window

4. Enter the unit weight in the **General properties** box according to the material data as listed in Table 1 (on page 13). This unit weight corresponds to the total permanent and variable load of the building.
5. Click the **Next** button or click the **Mechanical** tab to proceed with the input of model parameters.
   The linear elastic model involves only two basic parameters (E_{ref}, ν) enter them in the corresponding boxes following Table 1 (on page 13).
6. Click the **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.
7. Click the **OK** button to close the **Material sets** window.
8. Click the **OK** button to close the **Modify soil layers** window.

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 **Show global** button. .

1.1.4 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 finalise 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 (see Figure 10 (on page 18)).

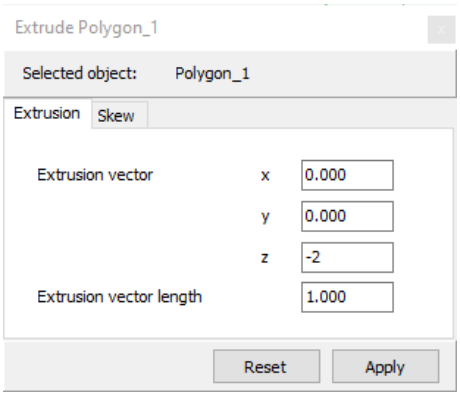


Figure 10: Extrude window

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.

1.1.5 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.
   The created building volume will appear green in colour after refinement. (see Figure 11 (on page 19)).

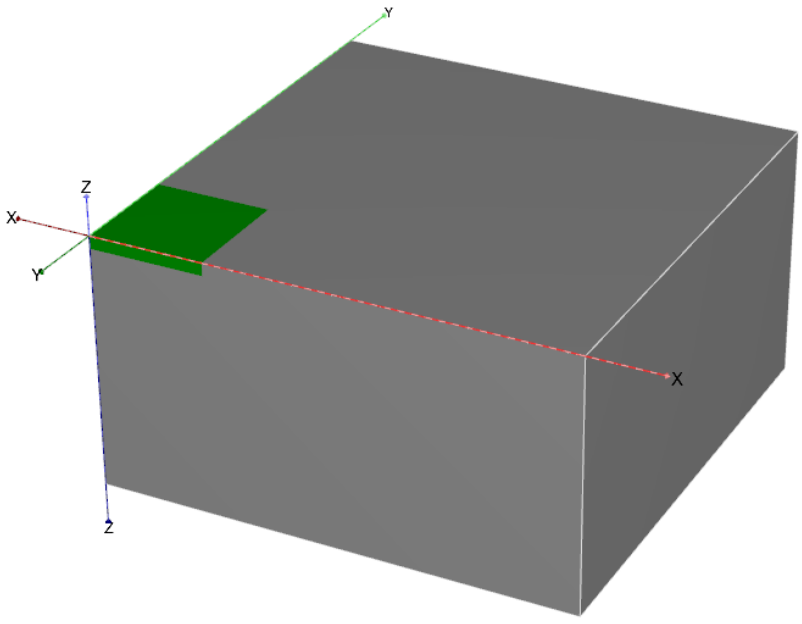


Figure 11: The indication of the local refinement in the model

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 as displayed on Figure 12 (on page 20).

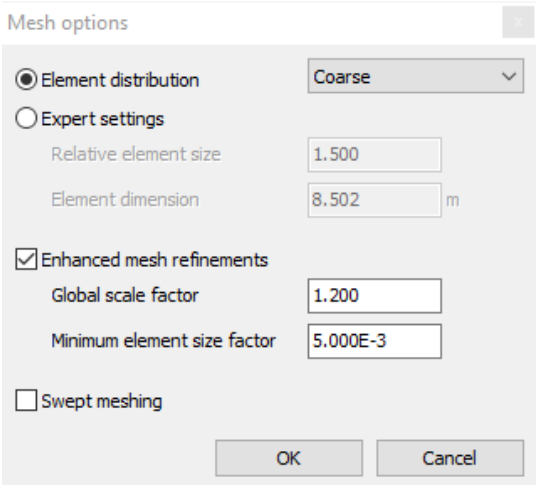


Figure 12: Mesh options window

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 (see Figure 13 (on page 20)).

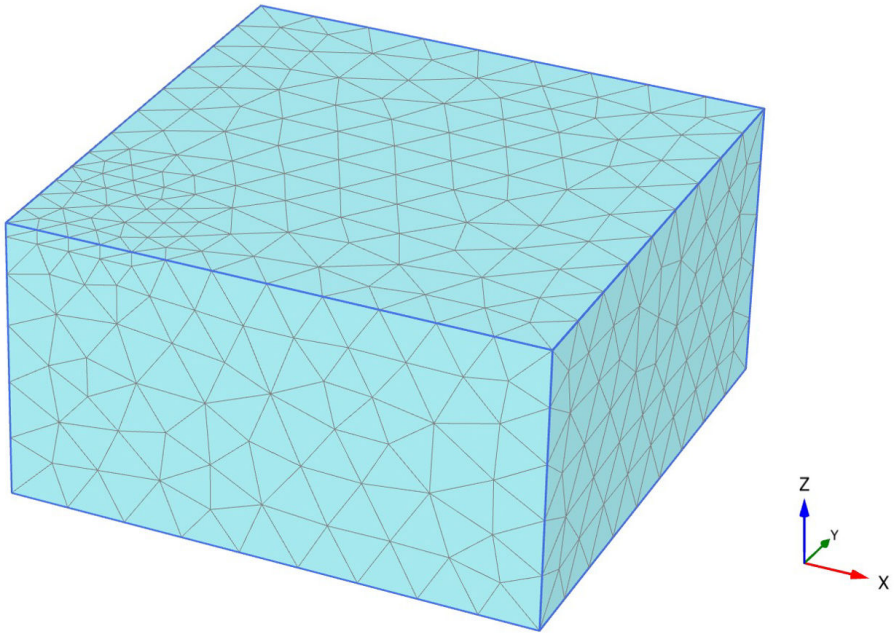


Figure 13: Generated mesh in the Output window

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

1.1.6 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 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 mode** to proceed with the definition of calculation phases.

   When a new project has been defined, a first calculation phase named 'Initial phase' (see Figure 14 (on page 21)), is automatically created and selected in the **Phases explorer**:

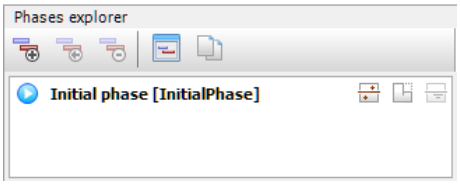


Figure 14: Phases explorer

   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 as in Figure 15 (on page 22).

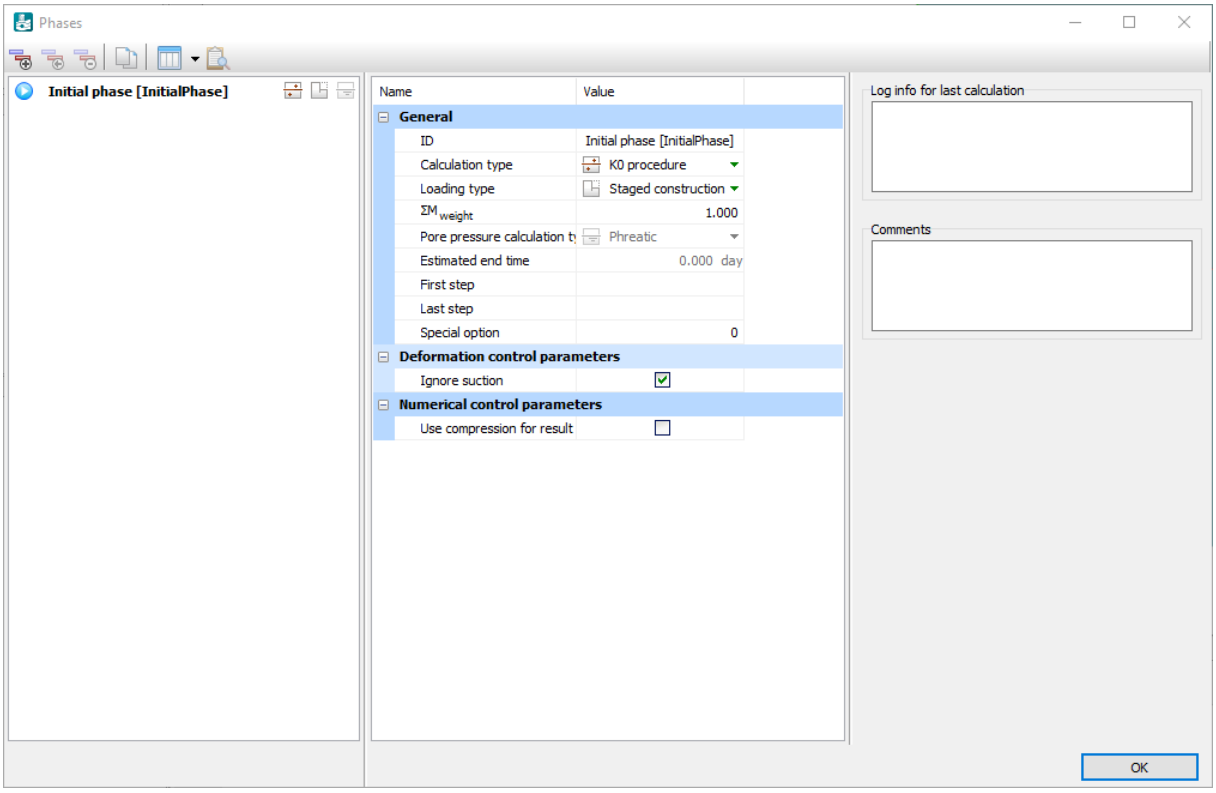


Figure 15: The Phases window for Initial phase

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

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

Note: The **K0 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 **K0 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 is automatically assigned to **GlobalWaterLevel: BoreholeWaterLevel_1** generated according to the **Head** value assigned to boreholes in the **Modify soil layers** window. .
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** (see Figure 16 (on page 23)).

   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.

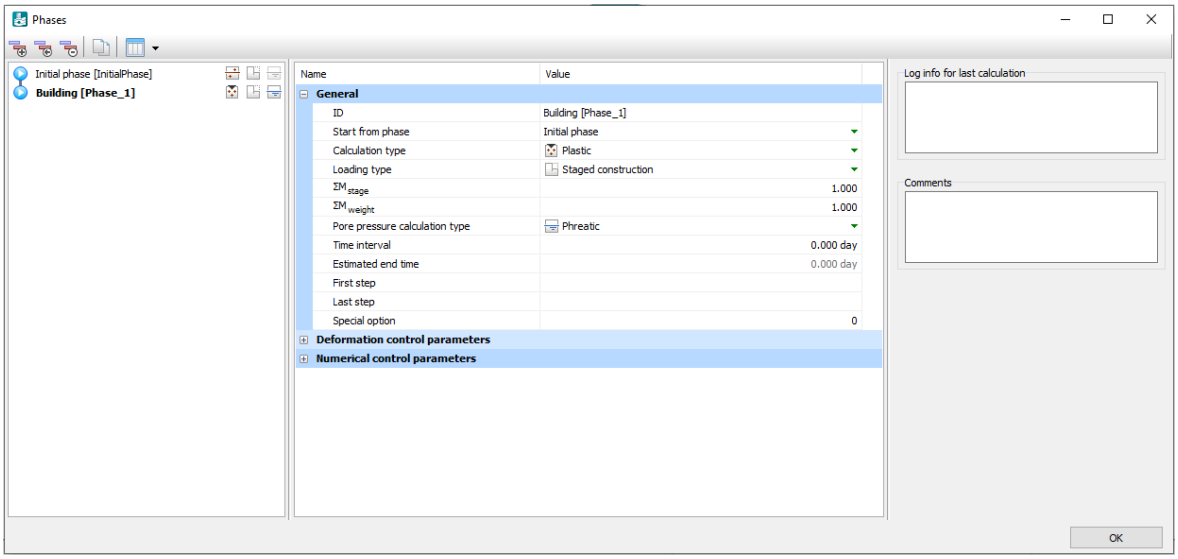


Figure 16: The Phases window for Building phase

5. Click **OK** to close the **Phases** window.
6. Right-click the building volume that was created earlier. Select the menu **Soil_1_Soil_2_1** > **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 (see Figure 17 (on page 24)) which gives information about the progress of the actual calculation phase.

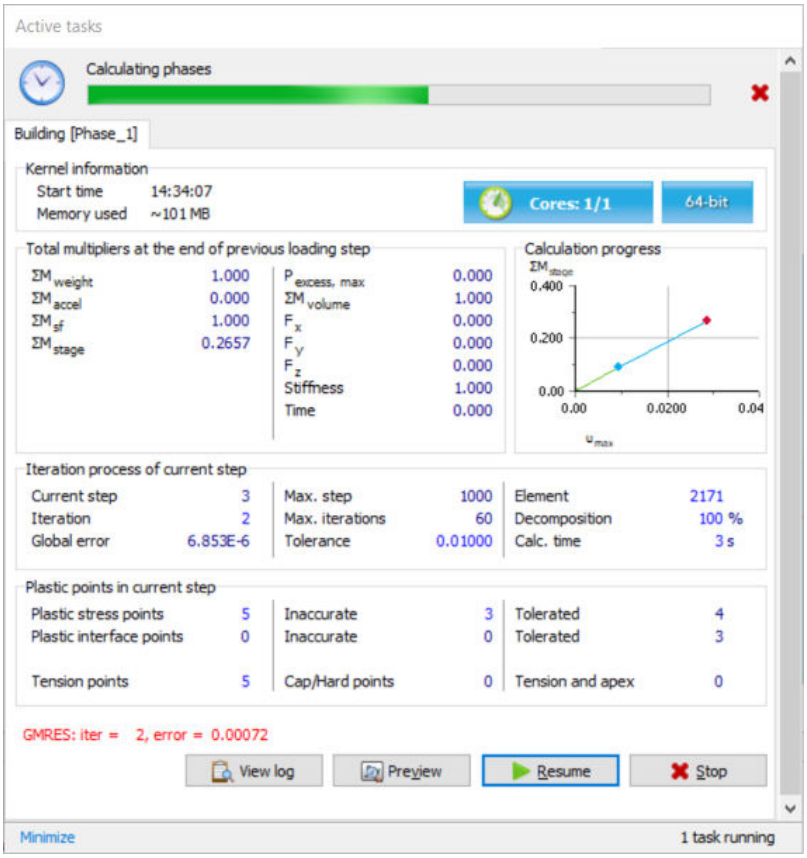


Figure 17: Active task window displaying the calculation progress

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.

1.1.7 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|**.
   Figure 18 (on page 25) 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.

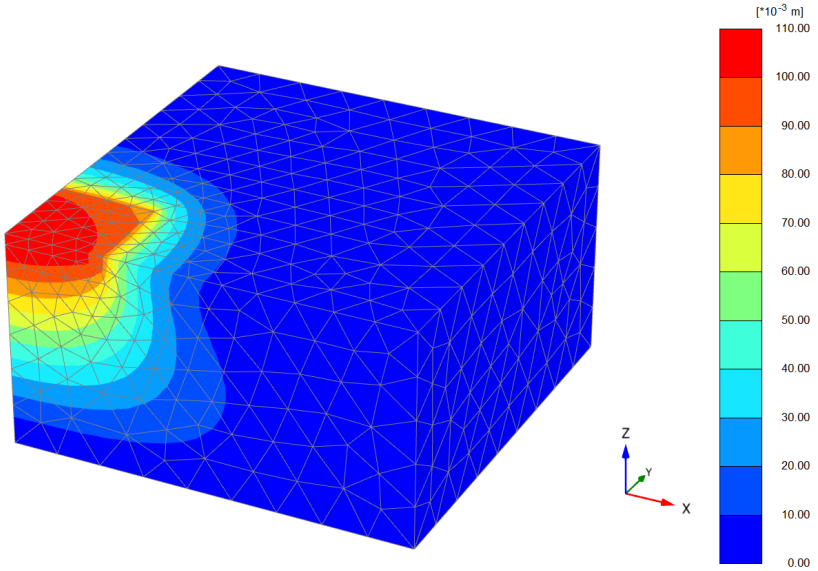


Figure 18: Shadings of Total displacements at the end of the last phase

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.

1.2 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 19 (on page 26).

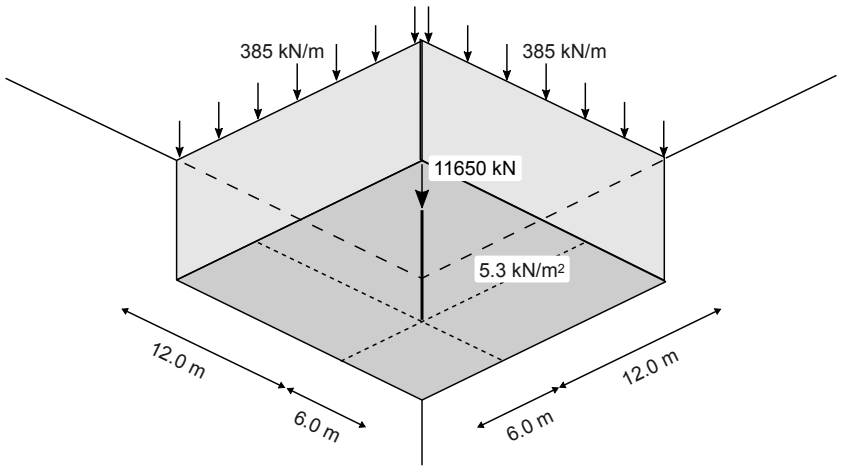


Figure 19: Geometry of the basement

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

1.2.1 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`).

1.2.2 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 **Mechanical** tabsheet, change the stiffness of the soil E'_{ref} to 5000 kN/m^2.
5. In the stiffness parameters for **Depth-dependency** enter a value of 500 in the E'_{inc} box. 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 (See Figure 20 (on page 28)).

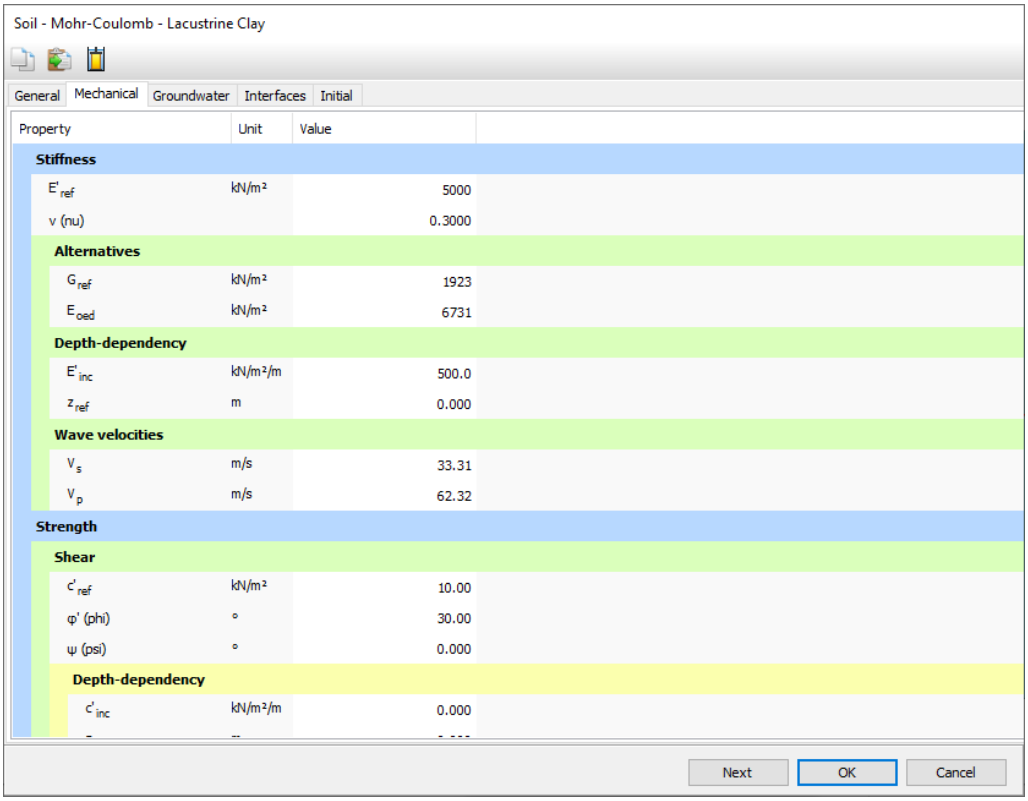


Figure 20: Stiffness Depth-dependency parameters

6. Click **OK** to close the **Soil** window.
7. Click **OK** to close the **Material sets** window.

1.2.3 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

Parameter	Name	Basement floor	Basement wall	Unit
General				
Type of behaviour	Type	Elastic	Elastic	-
Unit weight	γ	15	15.5	kN/m^3

Mechanical				
Isotropic	-	Yes	Yes	-
Young's modulus	E_1	$3 \cdot 10^7$	$3 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{12}	0.15	0.15	-
Thickness	d	0.5	0.3	m

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

Parameter	Name	Basement column	Basement beam	Unit
General				
Type of behaviour	Type	Elastic	Elastic	-
Unit weight	γ	24	6.0	kN/m^3
Mechanical				
Cross section type	Type	User-defined	User-defined	-
Cross section area	A	0.49	0.7	m^2
Moment of Inertia	I_2	0.020	0.029	m^4
	I_3	0.020	0.058	m^4
Young's modulus	E	$3 \cdot 10^7$	$3 \cdot 10^7$	kN/m^2

Creation of basement floor and basement walls Creation basement columns and basement beams

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. Once this is done, the internal surfaces should be visible.

6. Right-click the bottom surface of the building. From the appearing menu select the **Create** > **Create plate** option .
7. Delete the two vertical surfaces at the model boundaries. Subsequently, assign plates to the two vertical basement surfaces inside the model. Figure 21 (on page 30) shows a view of the basement and wall plates.

   - 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.

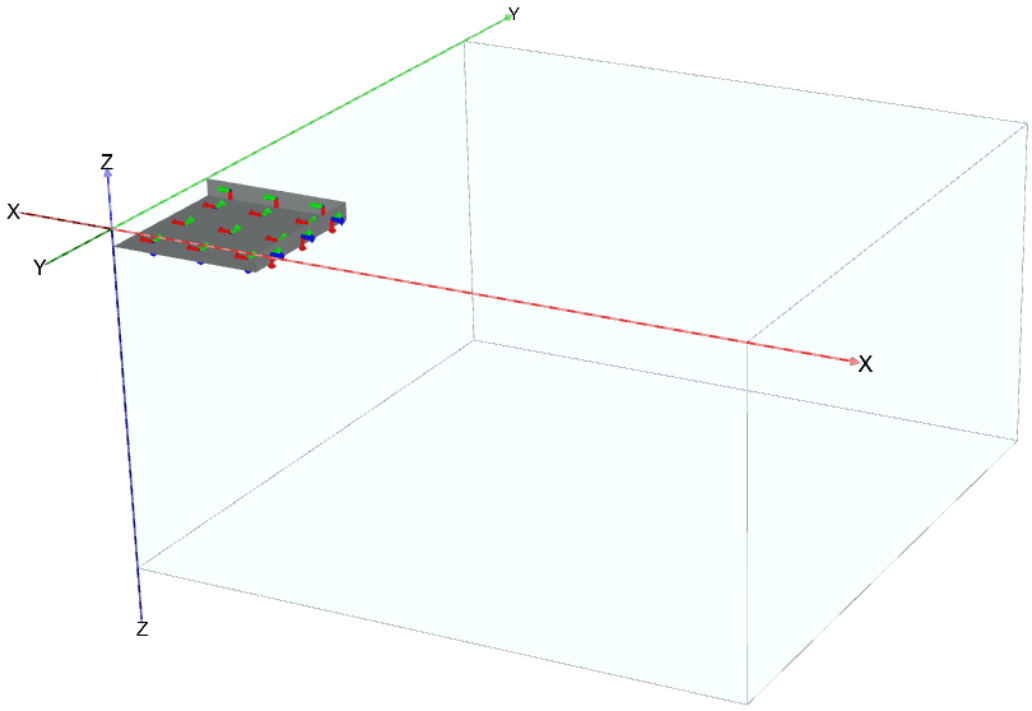


Figure 21: Location of plates in the project

Note: To visualise the local axes displayed in Figure 21 (on page 30) in the menu bar select **Options** > **Show local axes** > **Show local axes on surfaces with structures**.

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 28).
   At the moment of defining the Mechanical properties of a plate, a local axes and loading direction conventions window appears as displayed in Figure 22 (on page 31).

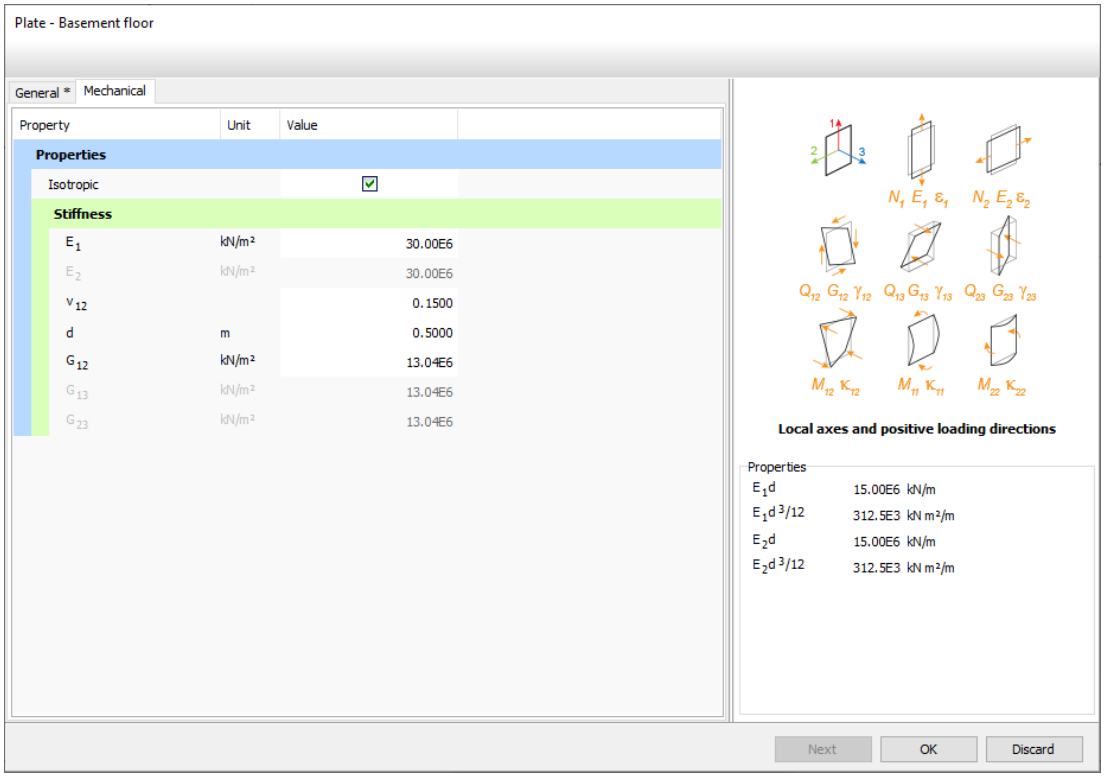


Figure 22: Mechanical properties of basement floor plate

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. If the materials are correctly assigned the plates get the respective material color.
11. Click the **OK** button to close the **Material sets** window.
12. Right-click on the basement floor and select the **Create** > **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. In this example, the value will be assigned in the **Phase definition** inside the (**Stage construction mode**).
13. Click the **Create line** button in the side toolbar and select the **Create line load** option from the additional tools displayed.
14. Click the command input area, type `0 18 0 18 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.
15. Click the **Create line** button in the side toolbar and select the **Create beam** option from the additional tools displayed.
16. To create the basement column a vertical beam is used, thenClick on (6 6 0) which is first point of the 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.
17. For the basement beams, 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.

18. Click the **Materials** button to open the material data base and set the **Set type** to **Beams**.
19. Create data sets for the horizontal beams according to Table 3 (on page 29).
20. Assign the data set to the corresponding beam elements by drag and drop.
21. Click the **Create load** button in the side toolbar.
22. 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.

1.2.4 Generate the mesh

To generate the mesh, follow these steps:

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

1.2.5 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 K_0 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**).

The calculation type for the phases representing the construction stages is set by default to **Plastic** .

Phase 1: Excavation

1. In the **Phases** window rename Phase_1 to `Excavation`.
2. 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** > **BoreholeVolume_1_1**.
3. In the **Model explorer** click the checkbox in front of the plates corresponding to the basement walls to activate them.

Phase 2: Construction

1. In the **Phases explorer** click the **Add phase** button . A new phase (Phase_2) is added. Double-click Phase_2.
   The **Phases** window pops up.
2. Rename the phase by defining its **ID** as `Construction`. Keep the default settings of the phase and close the **Phases** window.
3. In the **Model explorer** click the checkbox in front of the plate corresponding to the basement floor to activate it.
4. In the **Model explorer** click the checkbox in front of the beams to activate all the beams in the project.

Phase 3: Loading

1. Add a new phase following the **Construction** phase. Rename it to `Loading`.
2. 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/m^2, acting in the negative z-direction.
3. 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.
4. 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.

1.2.6 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 values of the vertical displacements are shown in Figure 23 (on page 34). If the title is not visible, select this option from the **View** menu.

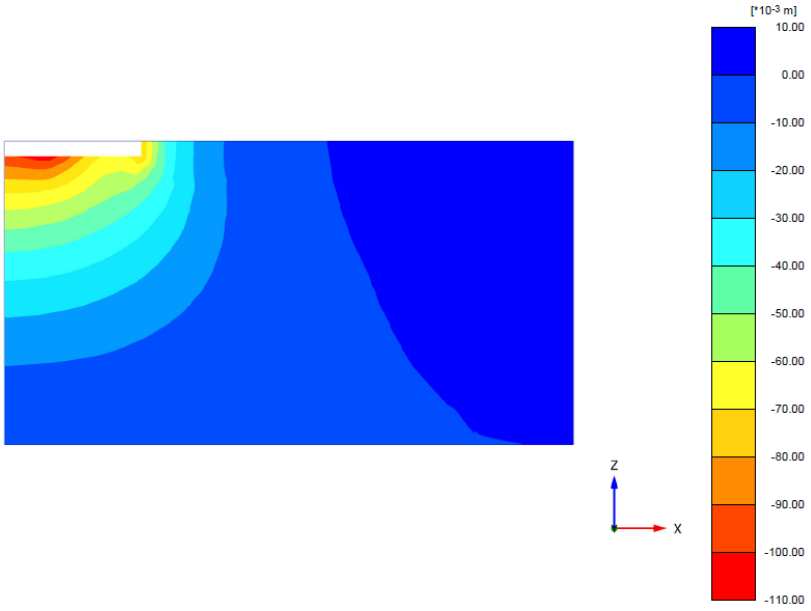


Figure 23: Cross section showing the total vertical displacement

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 bending moments are displayed in Figure 24 (on page 35).

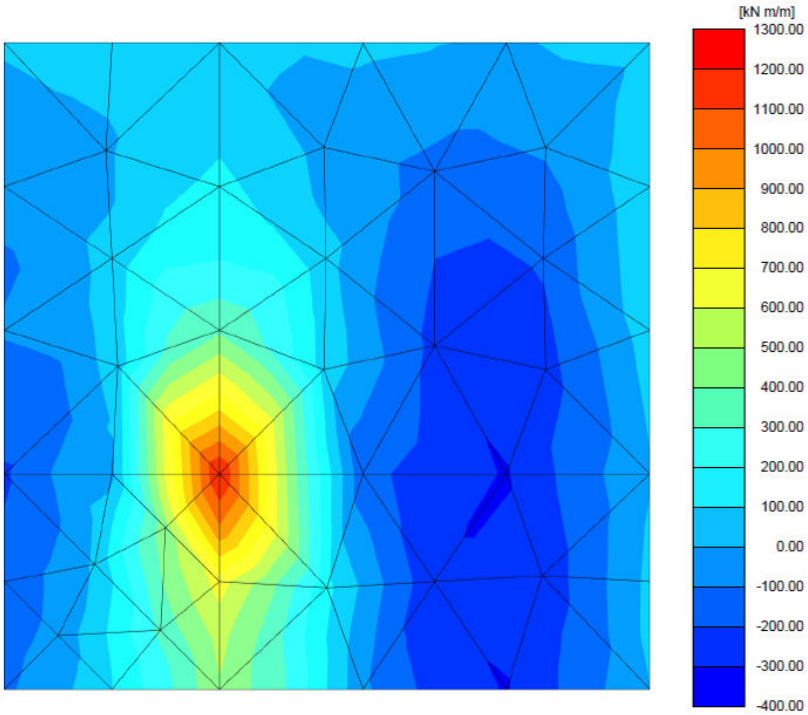


Figure 24: Bending moments in the basement floor

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.

1.3 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 20m and a diameter of 1.5m.

Objectives

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

1.3.1 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`).

1.3.2 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

Parameter	Name	Pile foundation	Unit
General			
Material type	Type	Elastic	-
Unit weight	γ	6.0	kN/m^3
Mechanical			
Cross section type	-	Predefined	-
Predefined beam type	-	Solid circular beam	-
Diameter	-	1.5	m
Young's modulus	E	$3 \cdot 10^7$	kN/m^2
Axial skin resistance	Type	Linear	-
Skin resistance at the top of the embedded beam	$T_{skin,start,max}$	200	kN/m

Mechanical			
Skin resistance at the bottom of the embedded beam	$T_{skin,end,max}$	500	kN/m
Base resistance	F_{max}	$1 \cdot 10^4$	kN

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 right click on the line then select the **Create** > **Create embedded beam** option 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 36). 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.

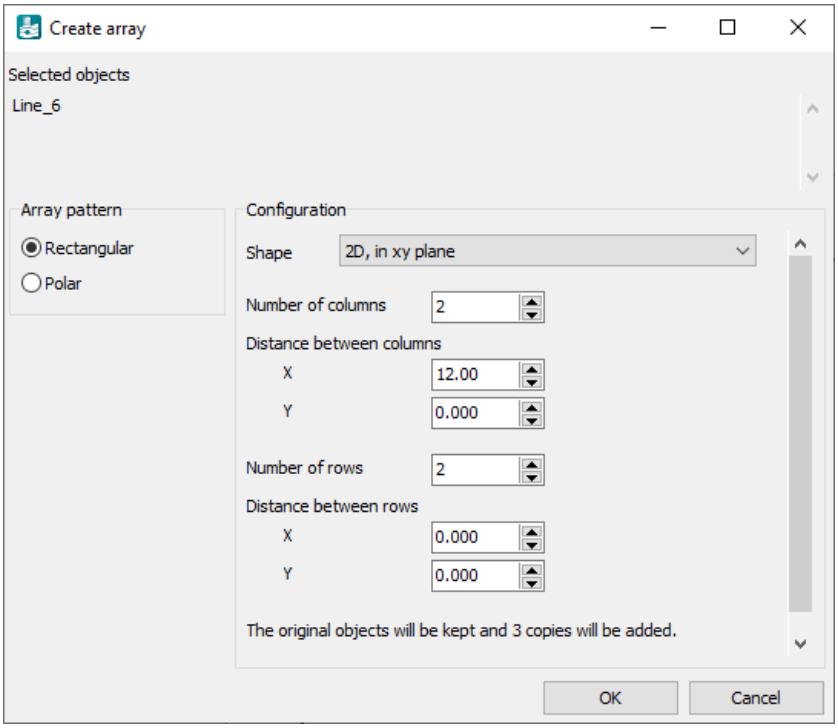


Figure 25: Create array window

13. Press **OK** to create the array. A total of 2x2=4 piles will be created.

1.3.3 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 in Figure 26 (on page 39):

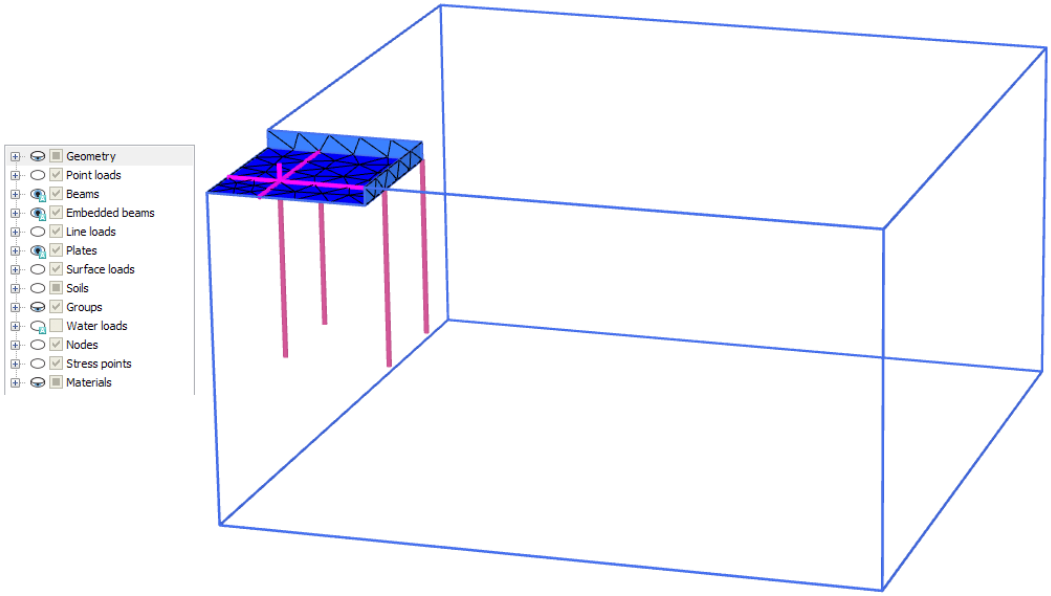


Figure 26: Partial geometry of the model in the Output

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

1.3.4 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.

1.3.5 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 in Figure 27 (on page 40):

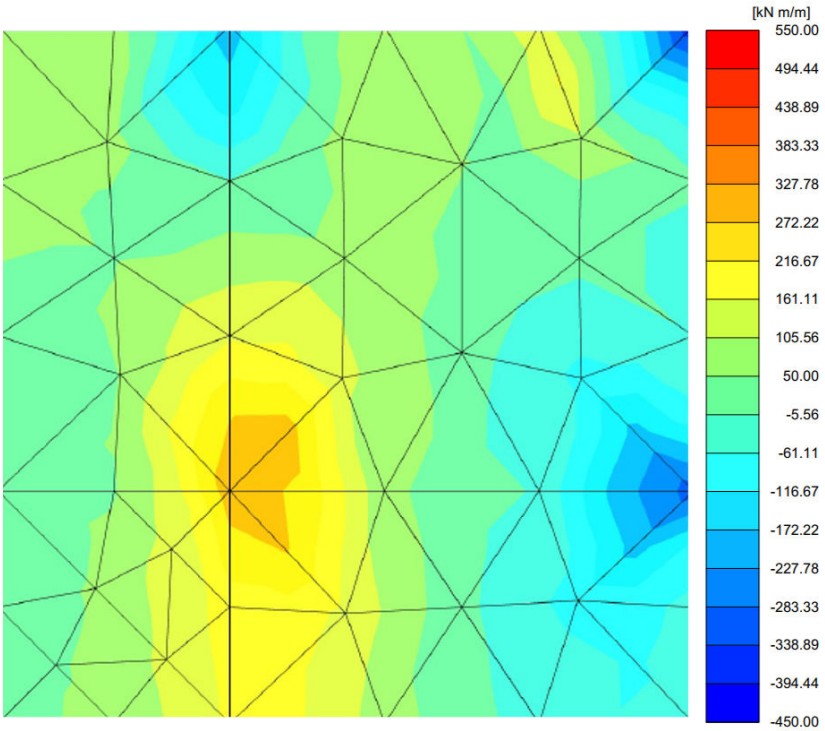
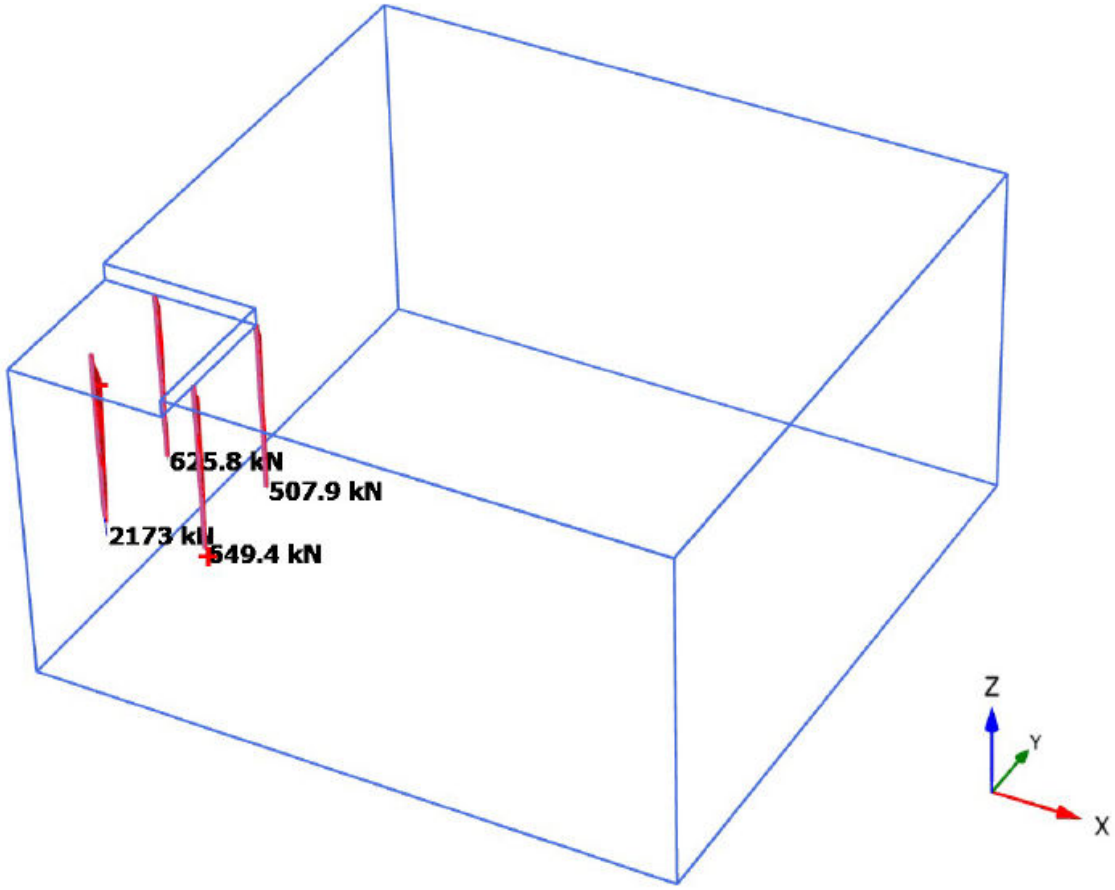


Figure 27: Bending moments in the basement floor

3. Adjust the legend by double-clicking on the legend and the **Legend settings** are displayed. Ensure to **lock the legend** to achieve the desired scaling.
   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:



Axial forces N (scaled up 0.500*10^{-3} times)

Maximum value = -796.7 kN (Element 16 at Node 4507)

Minimum value = -9156 kN (Element 1 at Node 4454)

Figure 28: Resulting axial forces (N) in the embedded beams

2 Excavation in sand

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.

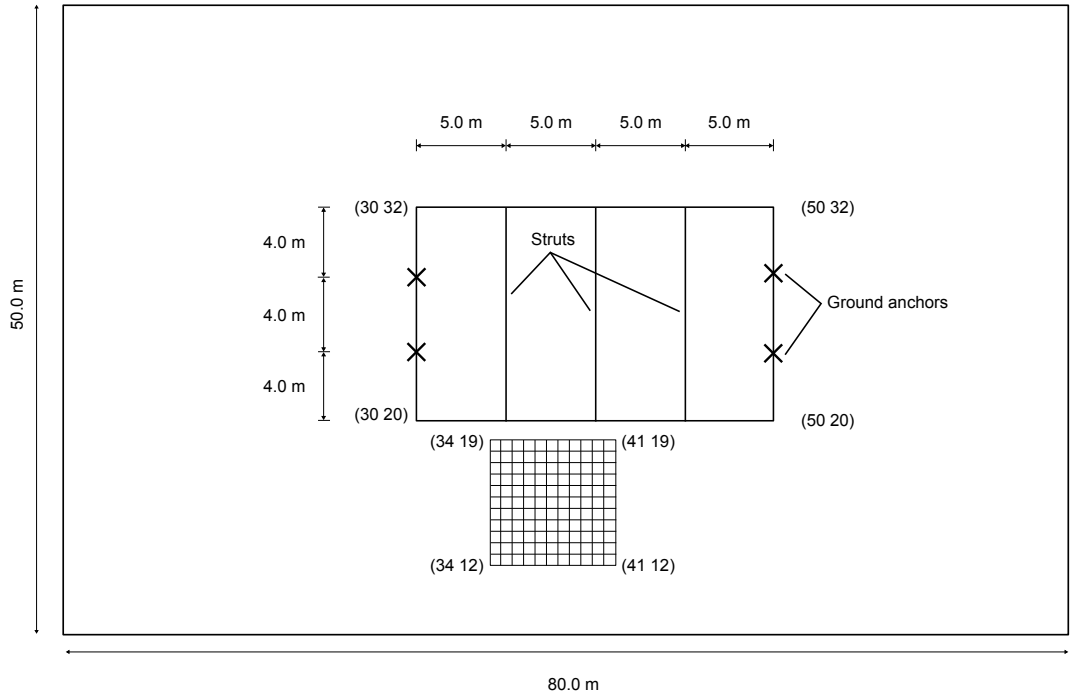


Figure 29: Top view of the excavation pit

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

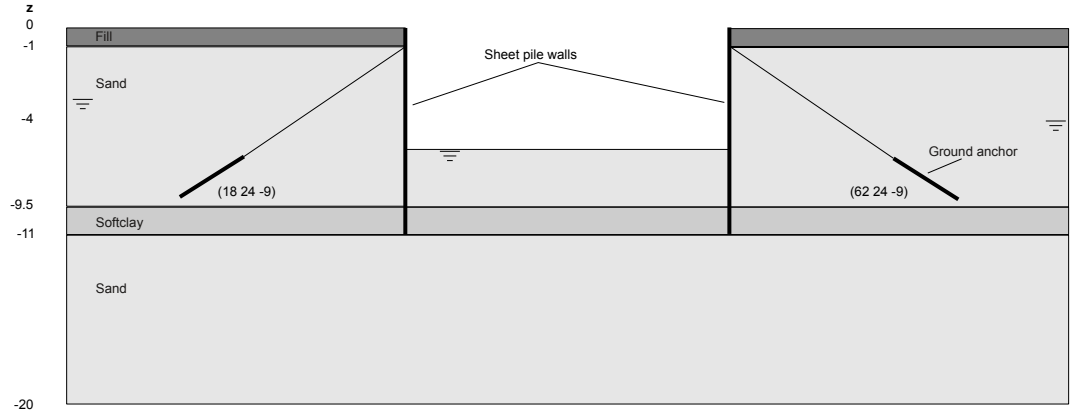


Figure 30: Cross section of the excavation pit with the soil layers

2.1 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
   a. x_{min} = 0.0 and x_{max} = 80.0,
   b. y_{min} = 0.0 and y_{max} = 50.0.

2.2 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.

2.3 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 Table 5 (on page 44) .

Table 5: Material properties for the soil layers

Property	Name	Fill	Sand	Soft Clay	Unit
General					
Soil model	Model	Hardening Soil	Hardening Soil	Hardening Soil	-
Drainage type	Type	Drained	Drained	Undrained A	-
Unsaturated unit weight	γ_{unsat}	16.0	17.0	16.0	kN/m^3
Saturated unit weight	γ_{sat}	20.0	20.0	17.0	kN/m^3
Mechanical					
Secant stiffness for CD triaxial test	E_{50}^{ref}	$2.2 \cdot 10^4$	$4.3 \cdot 10^4$	$2.0 \cdot 10^3$	kN/m^2
Tangent oedometer stiffness	E_{oed}^{ref}	$2.2 \cdot 10^4$	$2.2 \cdot 10^4$	$2.0 \cdot 10^3$	kN/m^2
Unloading/reloading stiffness	E_{ur}^{ref}	$6.6 \cdot 10^4$	$1.29 \cdot 10^5$	$1.0 \cdot 10^4$	kN/m^2

Mechanical					
Poisson's ratio	ν_{ur}	0.2	0.2	0.2	-
Power for stress level dependency of stiffness	m	0.5	0.5	1.0	-
Cohesion	c'_{ref}	1	1	5	kN/m^2
Friction angle	φ' *(phi)*	30.0	34.0	25	°
Dilatancy angle	ψ*(psi)*	0.0	4.0	0.0	°
Interfaces					
Strength determination	-	Manual	Manual	Manual	-
Interface reduction factor	R_{inter}	0.65	0.7	0.5	-
Initial					
K_0 determination	-	Automatic	Automatic	Automatic	-
Pre-overburden pressure	*POP*	0.0	0.0	0.0	-
Over-consolidation ratio	*OCR*	1.0	1.0	1.5	-

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 44).
6. In the **Mechanical** tabsheet, enter values for E_{50}^{ref}, E_{oed}^{ref}, E_{ur}^{ref}, m, c'_{ref}, φ'_{ref}, ψ and ν'_{ur}.
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 **Groundwater** tabsheet.
8. In the **Interfaces** tabsheet, in the **Strength** box select **Manual** and enter a value of 0.65 for the parameter R_{inter}.

   This parameter relates the strength of the interfaces to the strength of the soil, according to the equations:

   $c_i = R_{inter} c_{soil}$ and $\tan\varphi_i = R_{inter} \tan\varphi_i \leq \tan\varphi_{soil}$

   Hence, using the entered R_{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 list, the interface has the same strength properties as the soil (R_{inter} = 1.0).
- Note that a value of R_{inter} < 1.0, reduces the strength as well as the stiffness of the interface (for more information see the Reference Manual).

9. In the **Initial** tabsheet, define the OCR-value according to Table 5 (on page 44).
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 **Soil_1** > **Set material** > **Fill**.
13. In the same way assign the **Soft Clay** material to the soil layer between y = -9.5 m and y = -11.0 m.
14. For the remaining two soil layers assign the **Sand** material.

Note: The **Tension cut-off** option is activated by default at a value of 0 kN/m^2. This option is found in the tabsheet **Mechanical** > **Strength** > **Tension**. Here the **Tension cut-off** value can be changed or the option can be deactivated entirely.

2.4 Define the structural elements

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

2.4.1 Walings and Struts

The material properties for the structural elements are shown in the Table 6 (on page 46) . These are needed for defining the material in a later step.

Table 6: Material properties of waling and strut

Property	Name	Strut	Waling	Unit
General				
Material type	Type	Elastic	Elastic	-
Unit weight	γ	78.5	78.5	kN/m^3
Mechanical				
Cross section type	Type	User-defined	User-defined	-
Cross section area	A	0.007367	0.008682	m^2

Mechanical				
Moment of Inertia	I_2	$5.073 \cdot 10^{-5}$	$3.66 \cdot 10^{-4}$	m^4
	I_3	$5.073 \cdot 10^{-5}$	$1.045 \cdot 10^{-4}$	m^4
Young's modulus	E	$2.1 \cdot 10^{8}$	$2.1 \cdot 10^{8}$	kN/m^2

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 perimeter at level z=-1m. 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 46) and assign the materials accordingly.
12. Copy the strut into a total of three struts at x = 35 (existing), x = 40, and x = 45.

The created struts and walings can be seen in Figure 31 (on page 48)

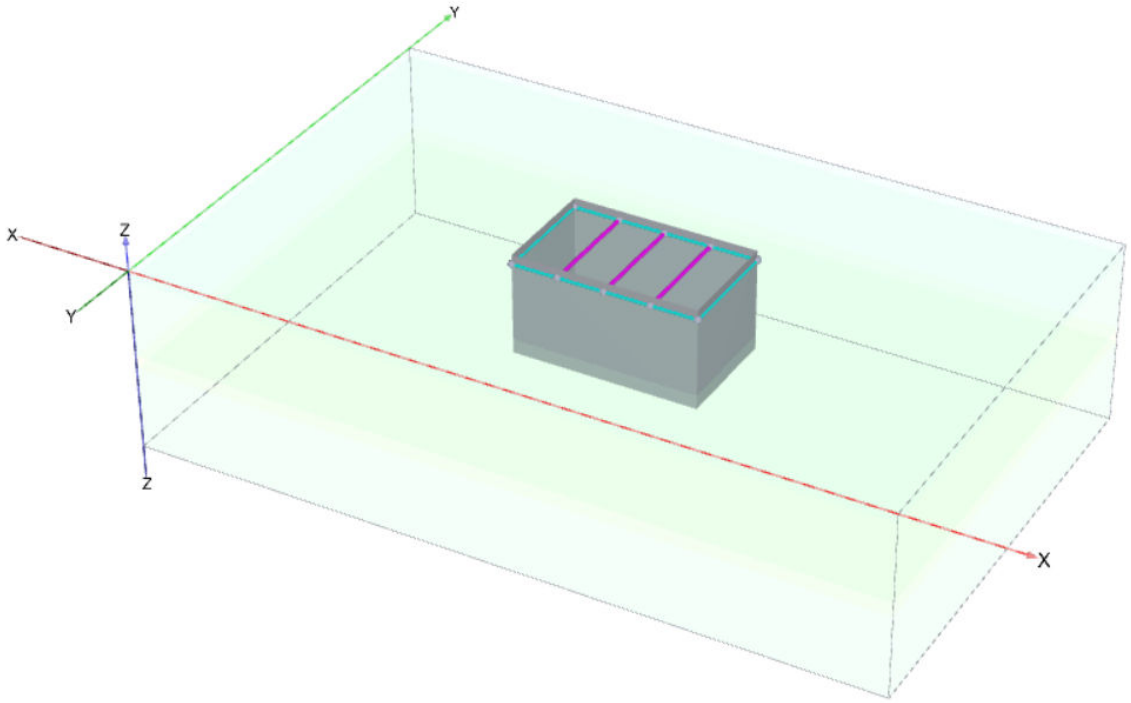


Figure 31: Visualization of struts and walings - Structures mode

2.4.2 Ground anchors

The material properties for the structural elements are shown in Table 7 (on page 48) and Table 8 (on page 48). These are needed for defining the material in a later step.

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

Property	Name	Node-to-node anchor	Unit
Material type	Type	Elastic	-
Axial stiffness	*EA*	$6.5 \cdot 10^5$	kN

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

Property	Name	Grout	Unit
General			
Material type	Type	Elastic	-
Unit weight	γ	24	kN/m^3

Mechanical			
Cross section type	Type	Predefined	-
Predefined cross section type	Type	Solid circular beam	-
Diameter	-	0.14	m
Young's modulus	E	$3 \cdot 10^7$	kN/m^2
Axial skin resistance	Type	Linear	-
Skin resistance at the top of the embedded beam	$T_{skin,start,max}$	200	kN/m
Skin resistance at the bottom of the embedded beam	$T_{skin,end,max}$	0.0	kN/m
Base resistance	F_{max}	0.0	kN

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

1. First the ungrouted part of the anchor is created using the Node-to-node anchor feature. Start creating the structure by clicking the create line on the side tool bar and selecting the **Create node-to-node anchor** button.
2. To create the ungrouted part of the first ground anchor click on the command line and type `30 24 -1 21 24 -7`. Press **<Enter>** and **<Esc>**
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 48) and Table 8 (on page 48) respectively. Assign the data sets to the node-to-node anchors and to the embedded beams.

   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. To do this, inside the array function on the Shape drop-down menu select the 1D in y direction option, then define number of colums as 2 and 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).

2.4.3 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

Parameter	Name	Sheet pile wall	Unit
General			
Type of behaviour	Type	Elastic	-
Weight	γ	2.55	kN/m^3
Mechanical			
Isotropic	-	No	-
Young's modulus	E_1	$1.46 \cdot 10^7$	kN/m^2
	E_2	$7.3 \cdot 10^5$	kN/m^2
Poisson's ratio	ν_{12}	0.0	-
Thickness	d	0.379	m
Shear modulus	G_{12}	$7.3 \cdot 10^5$	kN/m^2
	G_{13}	$1.27 \cdot 10^6$	kN/m^2
	G_{23}	$3.82 \cdot 10^5$	kN/m^2

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 **Create** > **Create plate** option from the appearing menu.

2. Create a data set for the sheet pile walls (plates) according to Table 9 (on page 50). 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 (**Create** > **Create positive interface/Create negative interface)**.

   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. 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.

   To consider the non-isotropic behaviour in the geometry, in the **Model explorer** tree expand the subtrees **Geometry** > **Surfaces** > **Polygon_Volume_3_2** and set **AxisFunction** to Manual and set $Axis1_z$ to -1. Do this for all the pile wall surfaces.

   Note:

   - 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 (see Figure 32 (on page 51)).

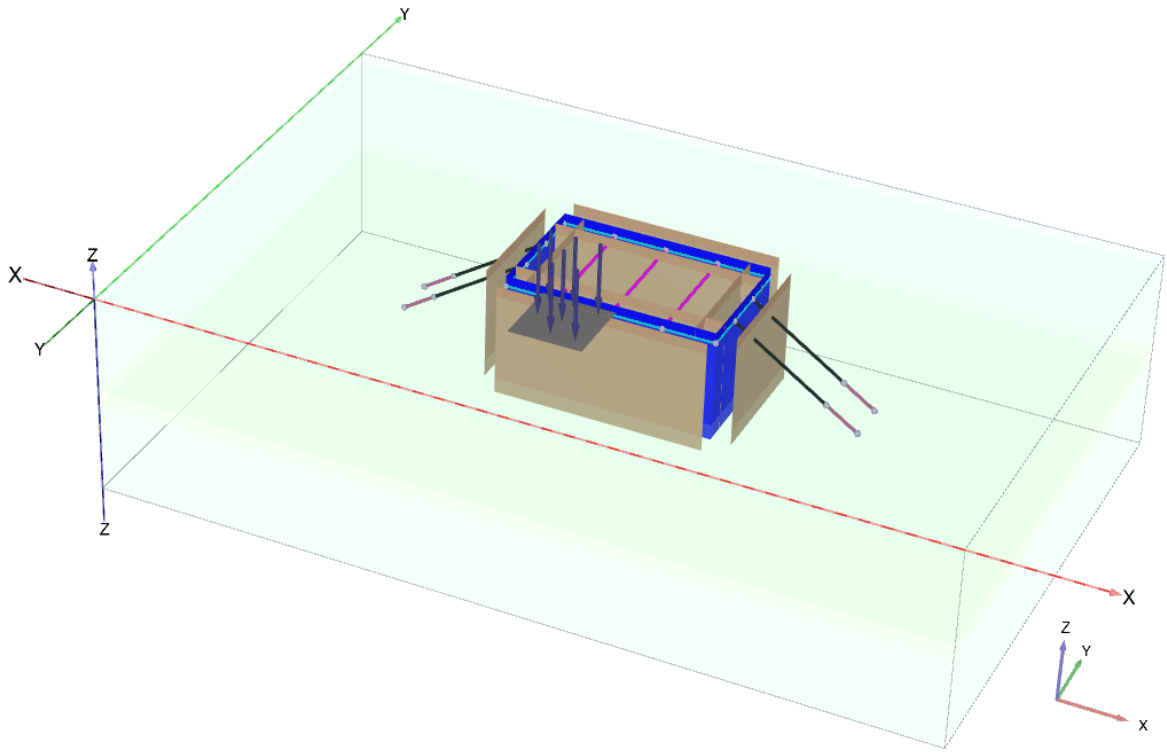


Figure 32: Structures mode - Complete project geometry

2.5 Generate the mesh

1. Proceed to the **Mesh Mode**.
2. Select the surface representing the excavation. Then 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.

2.6 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**.

   Note: To visualize more clearly the activated elements the soil layers can be hidden, this can be done by right clicking the soil volume of interest and selecting **Hide**.

   .
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 200kN as displayed in Figure 33 (on page 53).

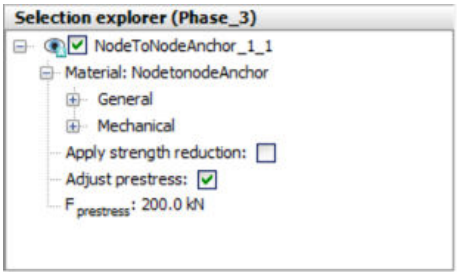


Figure 33: Node-to-node anchor in the Selection explorer

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 as in Figure 34 (on page 53).

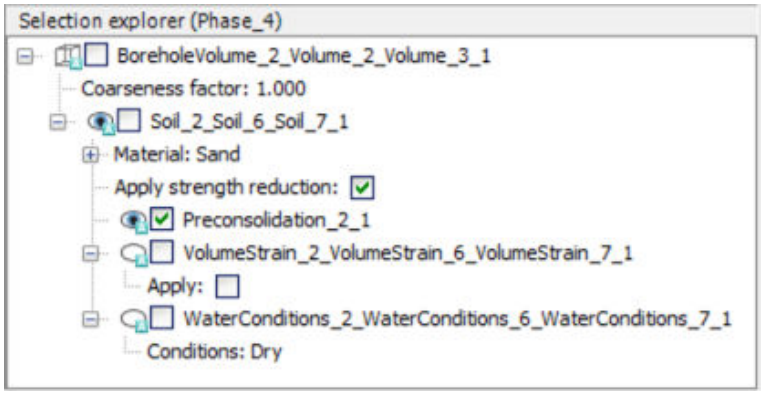


Figure 34: Water conditions in the Selection explorer

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 (vertically)**.
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 middle of excavation.

    Note: Hold **<Shift>** when drawing to get a straight line.
25. From **Stresses** > **Pore pressures** menu select the p_{steady} option.
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 Figure 35 (on page 54). Scroll the wheel button of the mouse to zoom in or out to get a better view.
27. Change the legend settings to:
    - Scaling: manual
    - Maximum value: 0
    - Number of intervals: 18

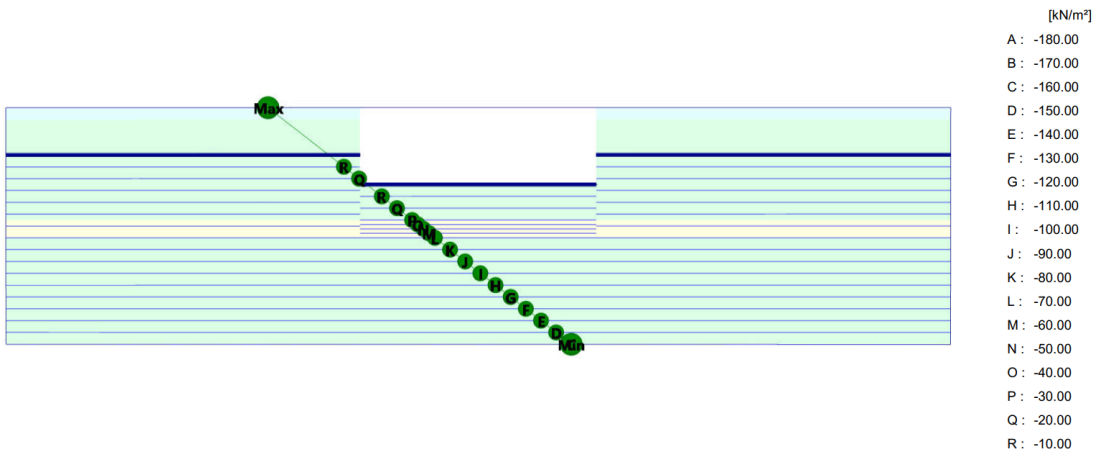


Figure 35: Preview of the steady state pore pressures in Phase_4 in a cross section

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 σ_z = -20 kN/m^2.

2.6.1 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.

2.7 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 (See Figure 36 (on page 56)).

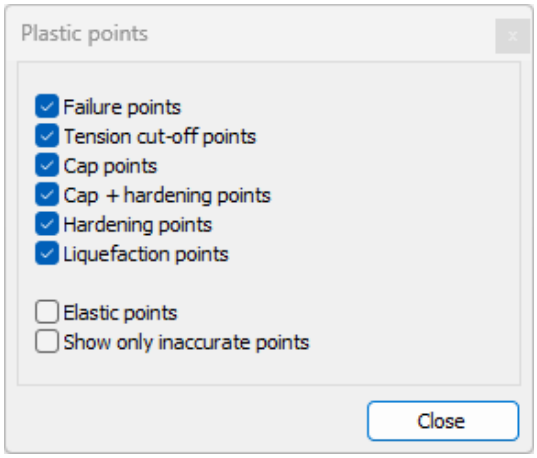


Figure 36: Plastic points window

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

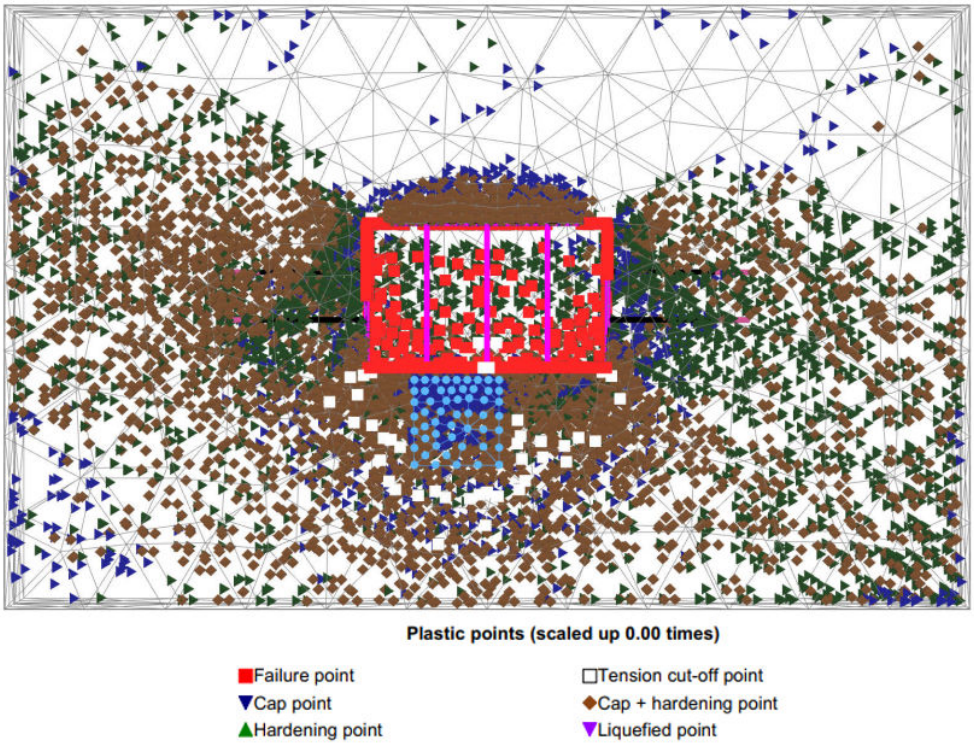


Figure 37: Plastic points at the end of the final phase

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. Click on **Curves manager** option and a window pops up. Click on **New** and select the stress points for X-Axis and Y-Axis as shown in Figure 38 (on page 57).

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

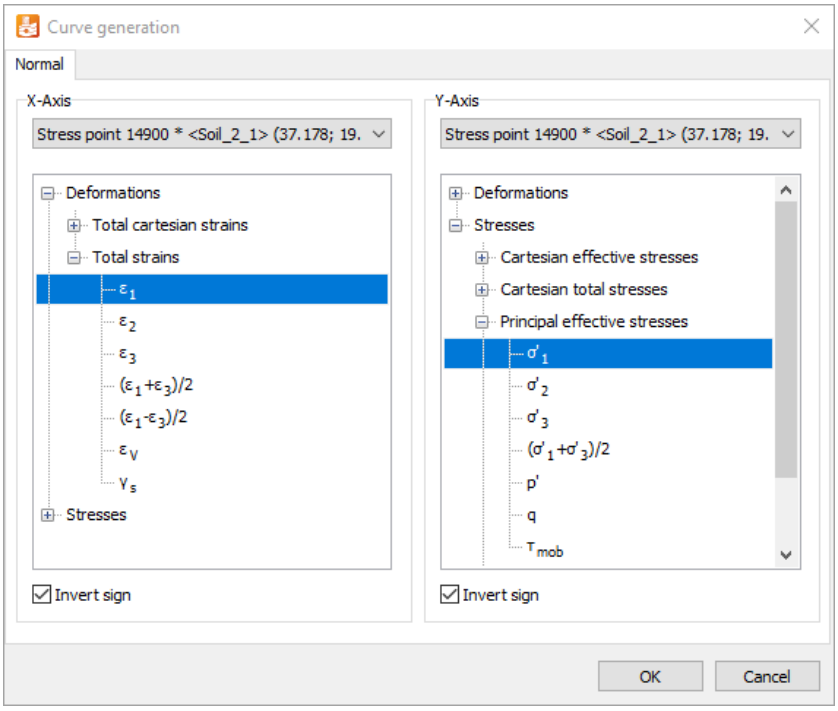


Figure 38: Curve generation window

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

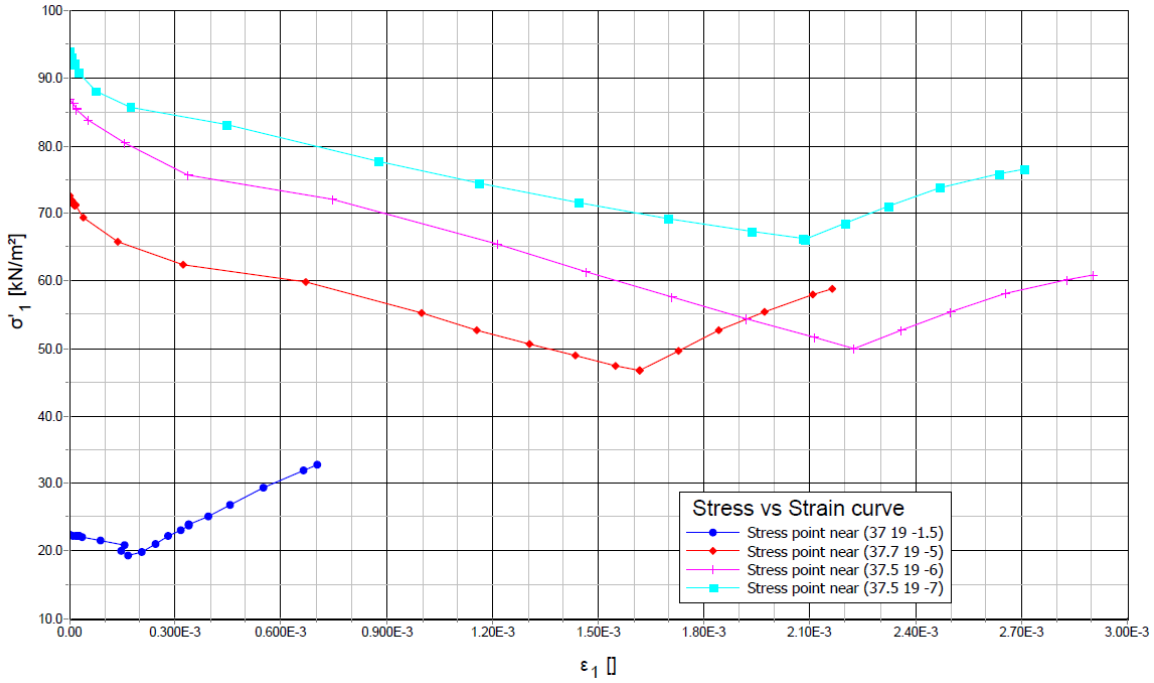


Figure 39: Stress - Strain curve

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 σ'_{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 σ'_{zz} under **Cartesian effective stresses**.
4. Click **OK** to confirm the input.

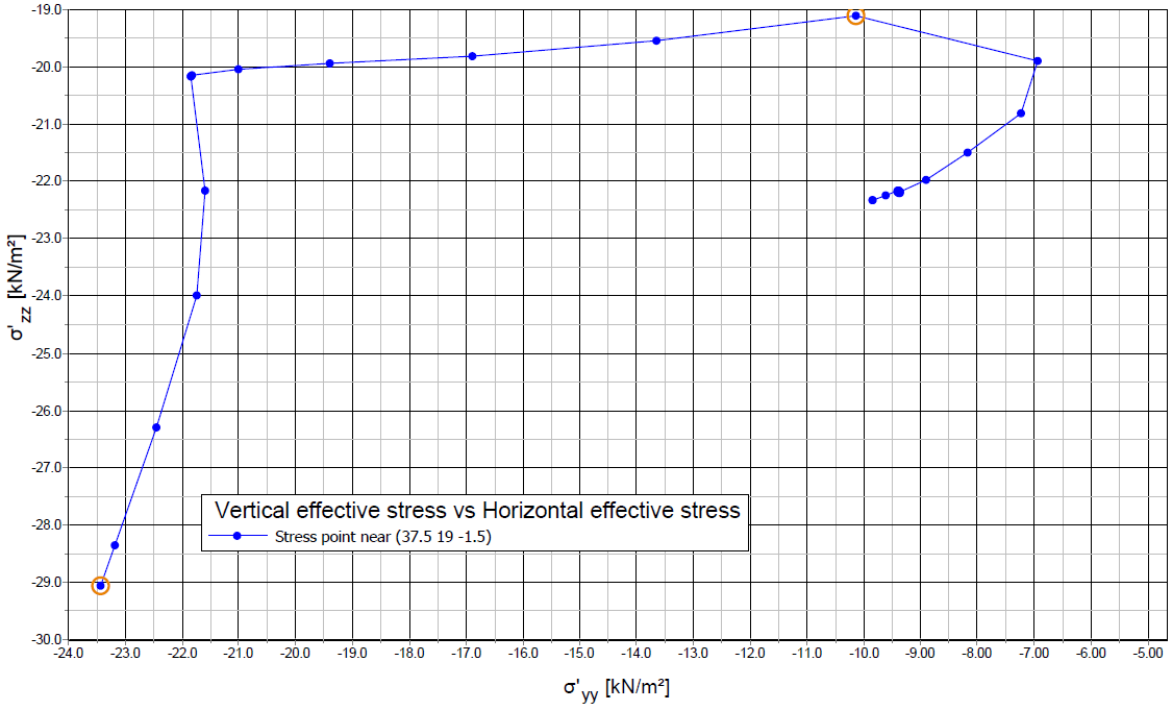


Figure 40: Vertical effective stress (σ'_{yy}) versus horizontal effective stress (σ'_{yy}) at stress node located near (37.5 19 -1.5)

3 Loading of a suction pile

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 the polycurve 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.

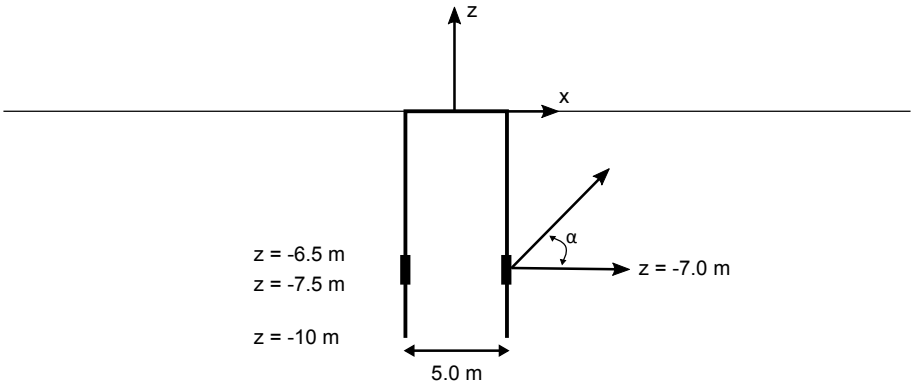


Figure 41: Geometry of the suction pile

3.1 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_{min} =-30.0 and x_{max} = 30.0,
   b. y_{min} = 0.0 and y_{max} = 30.0.
4. Click **OK**.

3.2 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 m and bottom boundary at z = -30 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.

3.3 Create and assign the material data sets

The material properties for the data sets are shown in Table 10 (on page 61).

Table 10: Material properties for the soil and interface

Property	Name	Clay	Interface	Unit
General				
Soil model	Model	Mohr-Coulomb	Mohr-Coulomb	-
Drainage type	Type	Undrained B	Undrained B	-

Property	Name	Clay	Interface	Unit
General				
Unsaturated unit weight	γ_{unsat}	20	20	kN/m^3
Saturated unit weight	γ_{sat}	20	20	kN/m^3
Mechanical				
Young's modulus	E'_{ref}	1000	1000	kN/m^2
Poisson's ratio	$\nu(nu)$	0.35	0.35	-
Increase in stiffness	E'_{inc}	1000	1000	$kN/m^2/m$
Reference level	z_{ref}	0.0	0.0	m
Undrained shear strength	$s_{u,ref}$	1.0	1.0	kN/m^2
Increase in undrained shear strength	$s_{u,inc}$	4.0	4.0	$kN/m^2/m$
Tension cut-off	-	Inactive	Inactive	-
Interfaces				
Strength determination	-	Rigid	Rigid	-
Interface reduction factor	R_{inter}	1.0	1.0	-
Initial				
K_0 determination	-	Manual	Manual	-
Initial lateral earth pressure coefficient	$K_{0,x}$, $K_{0,y}$	0.5	0.5	-

1. Click the **Materials** button .
2. Create the data sets given in Table 10 (on page 61). In the **Mechanical** tabsheet in the advanced parameters for strength deselect the **Tension cut-off** option. 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.

Note:

The **Interface** data set can be quickly created by copying the 'Clay' data set.

3. Assign the 'Clay' material data set to the soil layer and close the **Material sets** window.

3.4 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 set for local mesh refinements.

3.4.1 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.

1. Click the **Structures** tab to proceed to **Structures mode** .
2. Click the **Start designer** button  and then select the **Create polycurve** 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 **Polycurve designer** window pops up.

4. The polycurve is drawn in the xy-plane (see the Figure 42 (on page 64)). Hence the default orientation axes are valid for this example.

Note: For more information refer to "*Create polycurve*" in the Reference Manual .

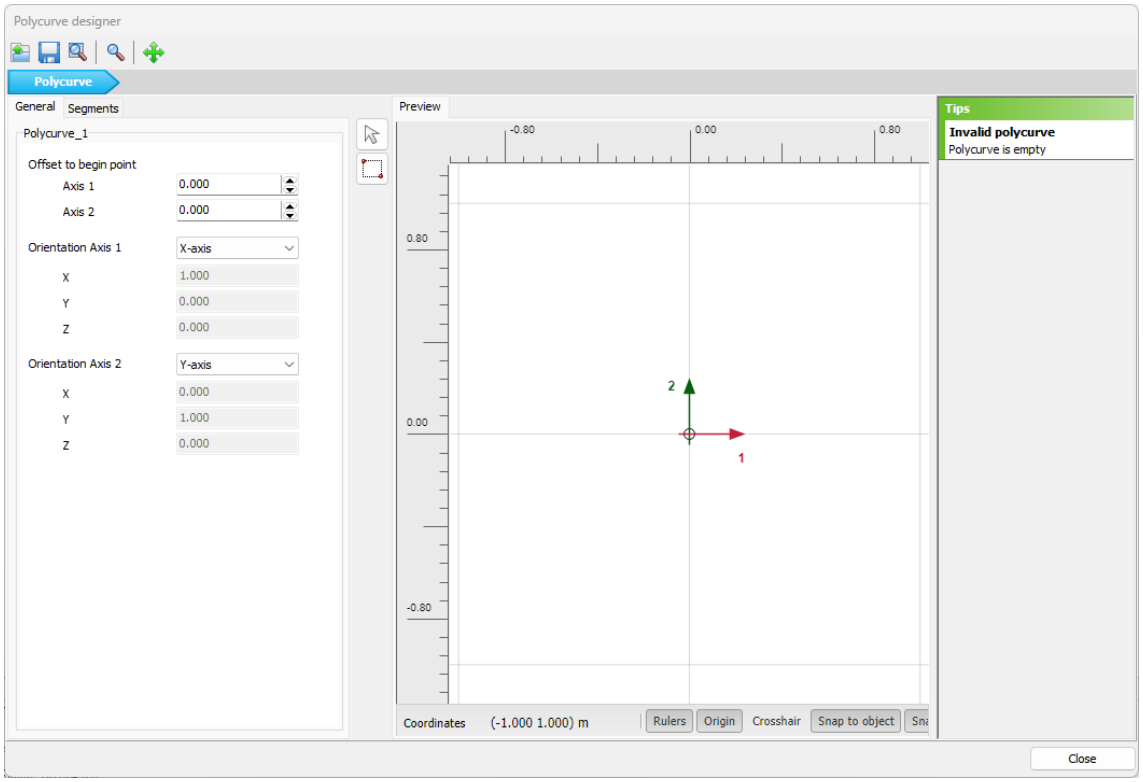


*Figure 42: **General** tabsheet of the **Polycurve designer***

5. In the **Segments** tabsheet, click on the **Add section** button in the side toolbar.
6. In the Segment explorer - for the newly created element - set the **Segment type** to **Arc**, the **Relative start angle** to 90°, the **Radius** to 2.5 m and the **Segment angle** to 180°. The geometry will appear automatically in the **Preview area**.

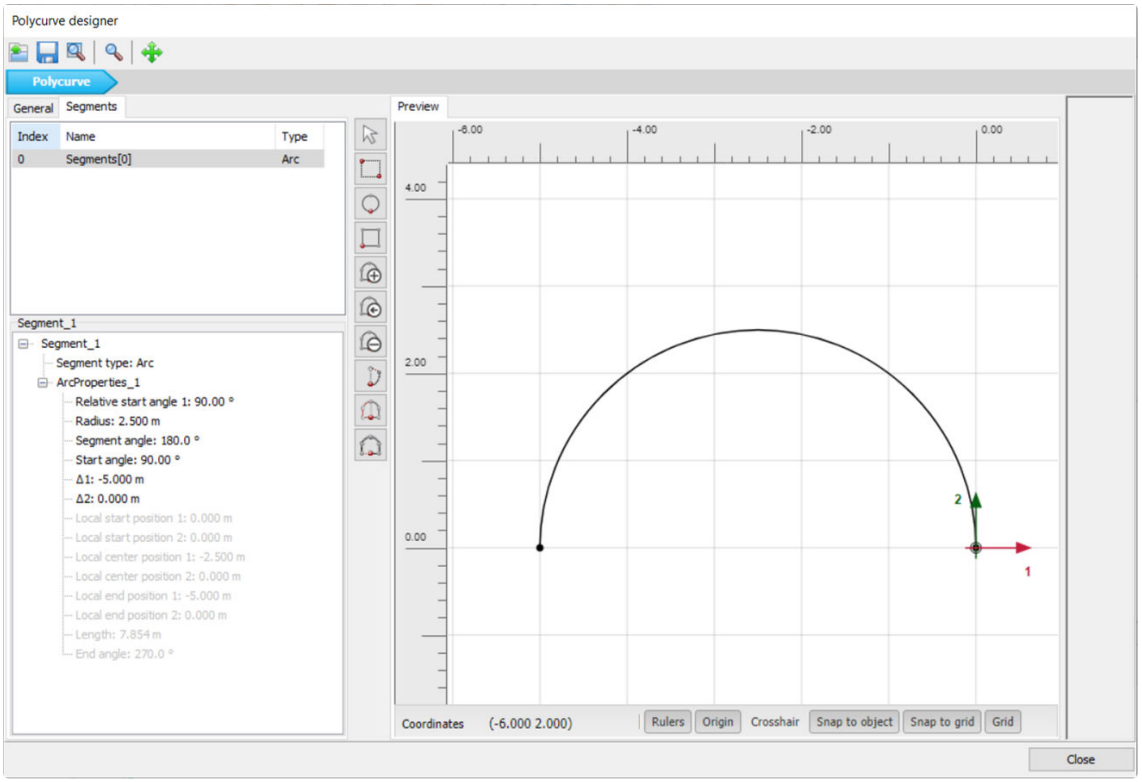


*Figure 43: **Segment** tabsheet of the **Polycurve designer***

7. Click on **close**, the polycurve will be automatically inserted into the model.
8. Click on the created polycurve and select the **Extrude object** and set the z value to -10 m.
9. Right-click the created surface, and select create positive interface to create an interface on the inside of the the suction pile. Similarly create a negative interface for the outside of the suction pile.
10. Right-click the polycurve and select **Close** from the appearing menu. Further, right click the closed polycurve and select **Create** > **Create surface**.
    This creates the top surface of the suction pile.
11. Right-click the top surface and create a negative interface.
12. In the **Model explorer** select each **Interface**. Once the interfaces are selected, in the **Selection Explorer** inside the **Material mode** select **Custom** from in the displayed dropdown menu (see Figure 44 (on page 66)).
13. For the **Material** option from the dropdown menu select **Interface**.

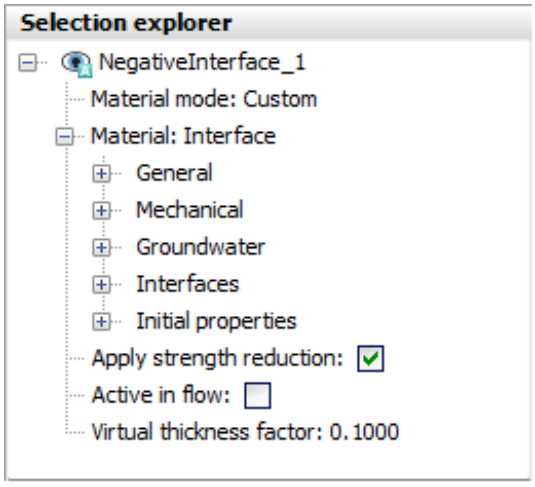


Figure 44: Interface material assignment in Selection explorer

14. Multi-select the top and the curved surface. Right-click on the selected surfaces and select the option **Create > Create rigid body** from the appearing menu (see Figure 45 (on page 67)).

Note:

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 .

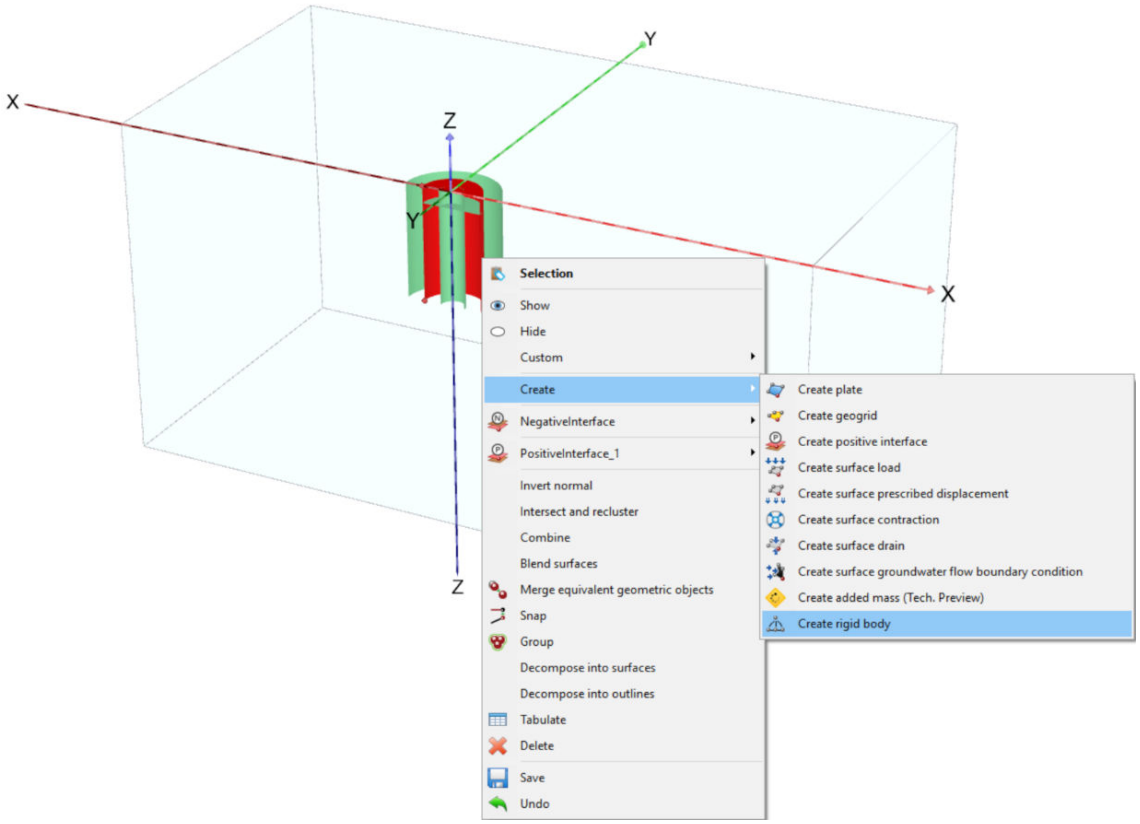


Figure 45: Rigid body creation

15. Select the created rigid body and in the **Selection explorer** set the reference point as (2.5 0 -7) assigning the values to x_{ref}, y_{ref} and z_{ref}.
16. As displayed in Figure 46 (on page 68) set the Translation condition$_y$ to **Displacement**, the Rotation condition$_x$ and Rotation condition$_z$ to **Rotation**. Their corresponding values are $u_y = \phi_x = \phi_z = 0$.

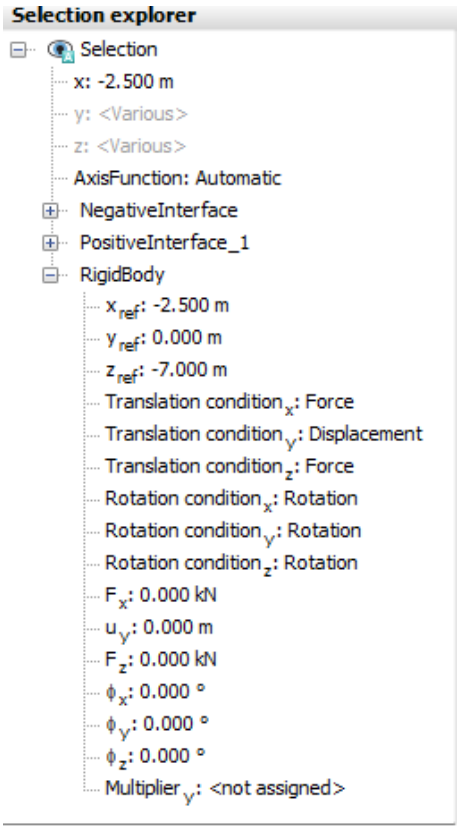


Figure 46: Rigid body in the Selection explorer

3.4.2 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 **Polycurve designer**.

1. Click the Start designer button  select 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 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. From the toolbar click **Close polycurve** to close the polycurve.
5. Close the **Polycurve designer**.
6. Click on the created polycurve and select the **Extrude object**  and set the z value to -15 m.
7. Select the just created polycurve and the polycurve of the suction pile created earlier, right-click and select **Delete** from the appearing menu as shown in Figure 47 (on page 69).

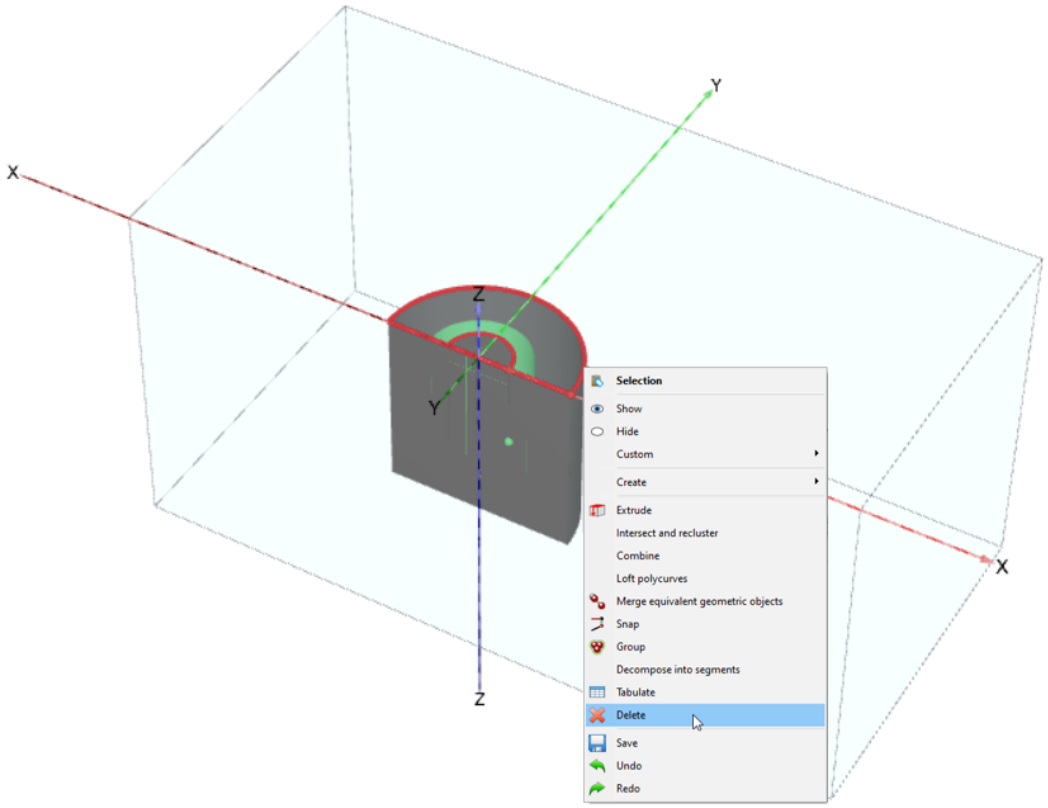


Figure 47: Deleting the two created polycurves

The geometry of the project is defined as shown in Figure 48 (on page 69).

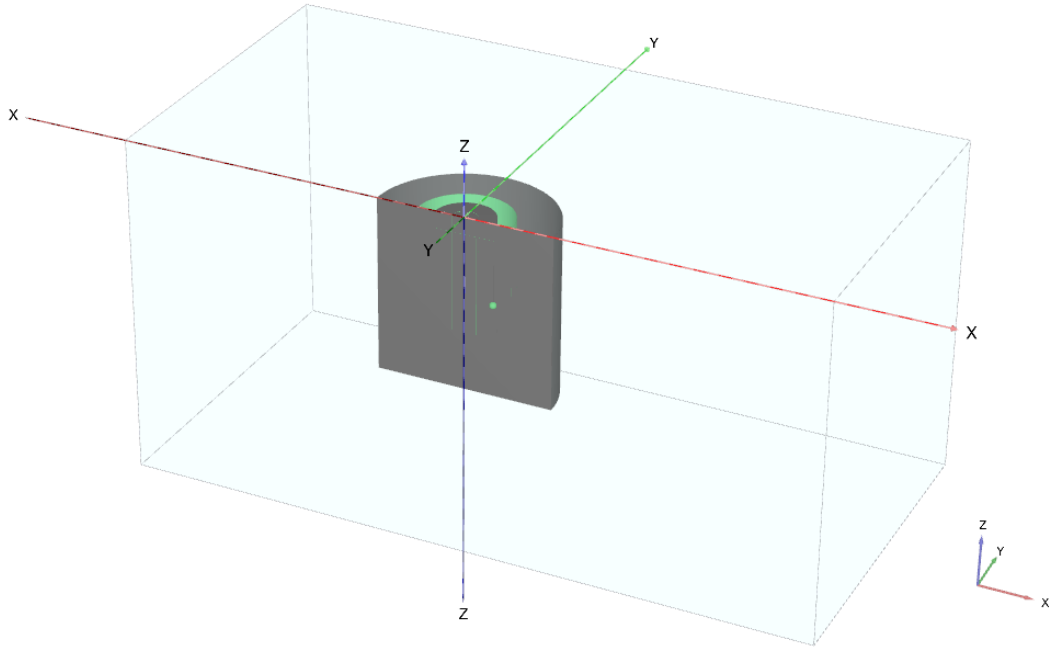


Figure 48: Geometry of the suction pile

3.5 Generate the mesh

In order to generate the mesh:

1. Click on the **Mesh** tab to proceed to the **Mesh mode**.
2. Hide the outer surface of the suction pile and the surface that limit the mesh refinement area. 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. Click the **Generate mesh** button  to generate the mesh. The element distribution is **Medium**.
5. Proceed to the **Staged construction mode** .

3.6 Define the calculation

The calculation for this exercise consists 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.

3.6.1 Initial phase: Initial conditions

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

1. Keep the calculation type of the Initial phase to **K0 procedure**. Ensure that all the structures and interfaces are switched off.

3.6.2 Phase 1: Installation of suction pile

1.  Add a new calculation phase and rename it as `Install pile`.
2. For this phase, we use the option of **Ignore undrained behaviour**.
3. Activate all the rigid bodies and interfaces in the project.

3.6.3 Phase 2: Load Pile 30 degrees

1. Add a new phase and rename it as `Load pile 30 degrees`.
2. In the **Phases** window, go to the **Deformation control parameters** subtree and check the **Reset displacements to zero** checkbox.
3. In the **Numerical control parameters** subtree set:
   a. The **Solver type** to **Pardiso (multicore direct)** to enable a faster calculation for this particular project.
   b. Uncheck the **Use default iter parameters** checkbox, which allows you to change advanced settings.
   c. Set the **Max load fraction per step** to 0.1.
4. In the **Model explorer** click on the **Rigid bodies**.
5. In the **Selection explorer** tree, set F_x = 1949 kN and F_z = 1125 kN for the selected rigid bodies.

3.6.4 Phase 3,4,5,6: Load Pile with different direction angles

Define the remaining phases according to the information in Table 11 (on page 71). 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

Phase	Start from phase	F_x	F_z
Load pile 30 degrees [Phase_2]	Phase_1	1949 kN	1125 kN
Load pile 40 degrees [Phase_3]	Phase_1	1724 kN	1447 kN
Load pile 50 degrees [Phase_4]	Phase_1	1447 kN	1724 kN
Load pile 60 degrees [Phase_5]	Phase_1	1125 kN	1949 kN

The order of the phases is indicated in the **Phases explorer** (see Figure 49 (on page 72)). 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.

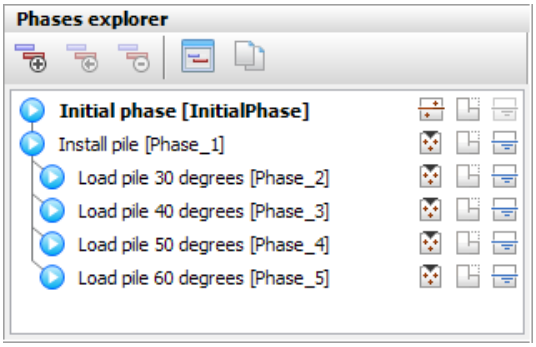


Figure 49: Phases explorer

3.6.5 Execute the calculation

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

3.7 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 Y-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.

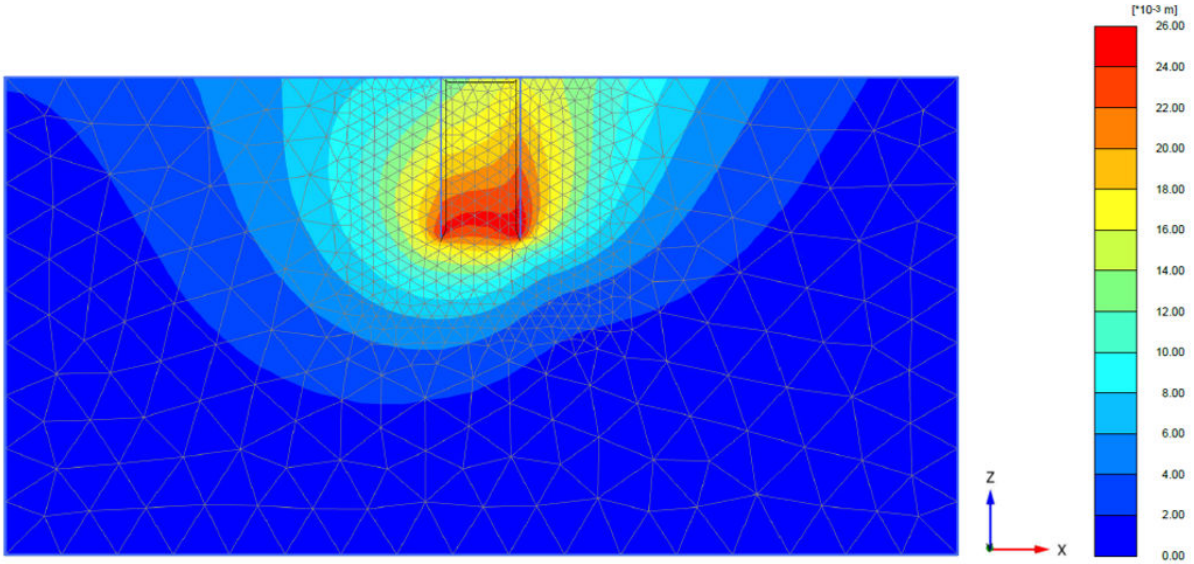


Figure 50: Total displacement of the suction pile at the end of Phase_2

4

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 Figure 51 (on page 74). 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.

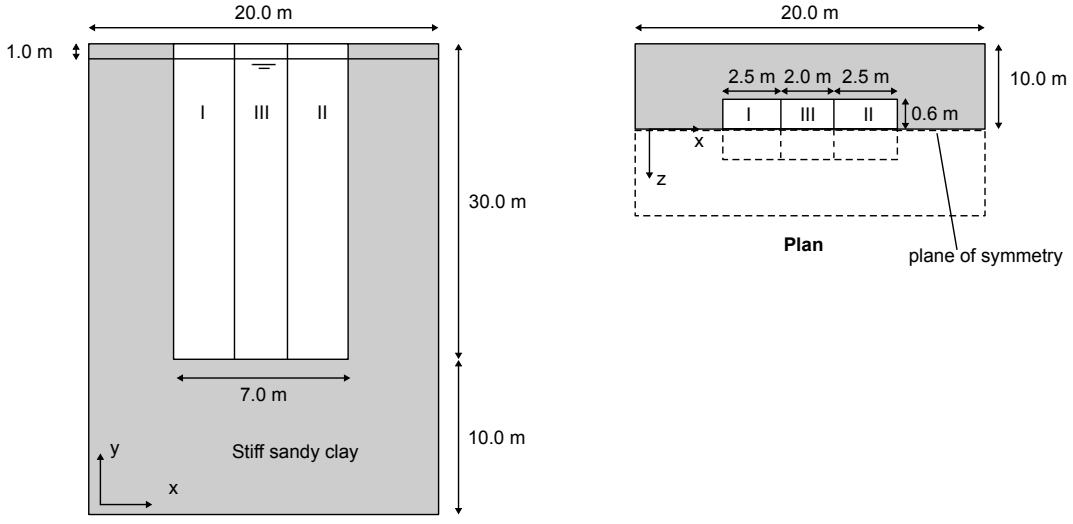


Figure 51: Geometry of the diaphragm wall

4.1 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_{min} =0 and x_{max} = 20,
   b. y_{min} = 0 and y_{max} = 10.
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 = 40m and bottom boundary at z = 0m.
3. Set the **Head** to 39m.

4.3 Create and assign the material data sets

The material properties for the data sets are shown in Table 12 (on page 75).

Table 12: Material properties for the soil and concrete

Property	Name	Stiff sandy clay	Concrete	Unit
General				
Soil model	Model	Mohr-Coulomb	Linear Elastic	-
Drainage type	Type	Drained	Non-porous	-

Property	Name	Stiff sandy clay	Concrete	Unit
General				
Unsaturated unit weight	γ_{unsat}	15	24	kN/m^3
Saturated unit weight	γ_{sat}	20	-	kN/m^3
Mechanical				
Young's modulus	E'_{ref}/ E_{ref}	$50 \cdot 10^3$	$2.6 \cdot 10^7$	kN/m^2
Poisson's ratio	ν(nu)	0.3	0.2	kN/m^2
Cohesion	c'_{ref}	15	-	kN/m^2
Friction angle	φ' (phi)	30	-	°
Dilatancy angle	ψ (psi)	0.0	-	°
Interfaces				
Strength determination	-	Rigid	Rigid	-
Initial				
K_0 determination	-	Automatic	Automatic	-

1. Click the **Materials** button .
2. Create the data sets for the soil layer and the concrete as specified in Table 12 (on page 75).
3. Assign the 'Stiff sandy clay' material data set to the soil layer and close the **Material sets** window.

4.4 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 Table 13 (on page 77):

Table 13: Surfaces composing the diaphragm wall

Segment	Point coordinates
I	(6.5 0 40) (9 0 40) (9 0.6 40) (6.5 0.6 40)
II	(11 0 40) (13.5 0 40) (13.5 0.6 40) (11 0.6 40)
III	(9 0 40) (11 0 40) (11 0.6 40) (9 0.6 40)

1. Click the **Create surface** button in the side toolbar and create three surfaces accordingly to Table 13 (on page 77).
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 as displayed in Figure 52 (on page 77).

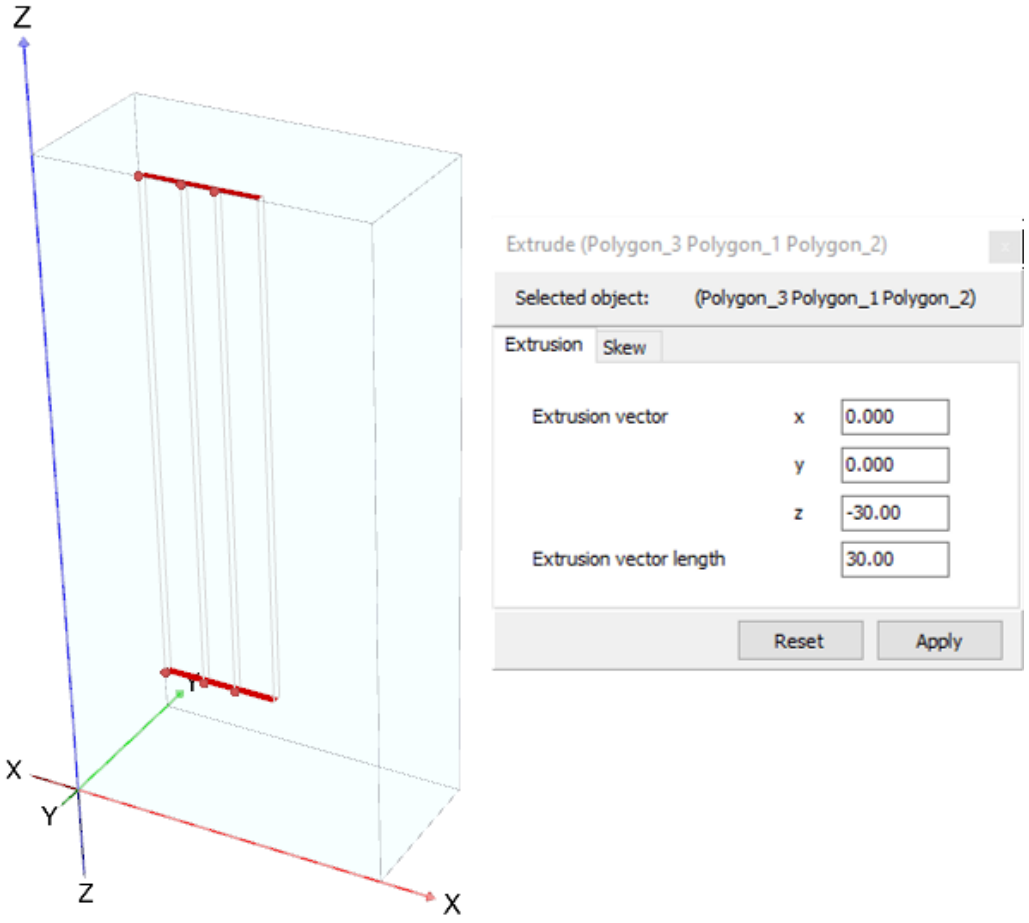


Figure 52: Extruded surfaces

4. Delete the surfaces.

4.5 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 (See Figure 53 (on page 78)).

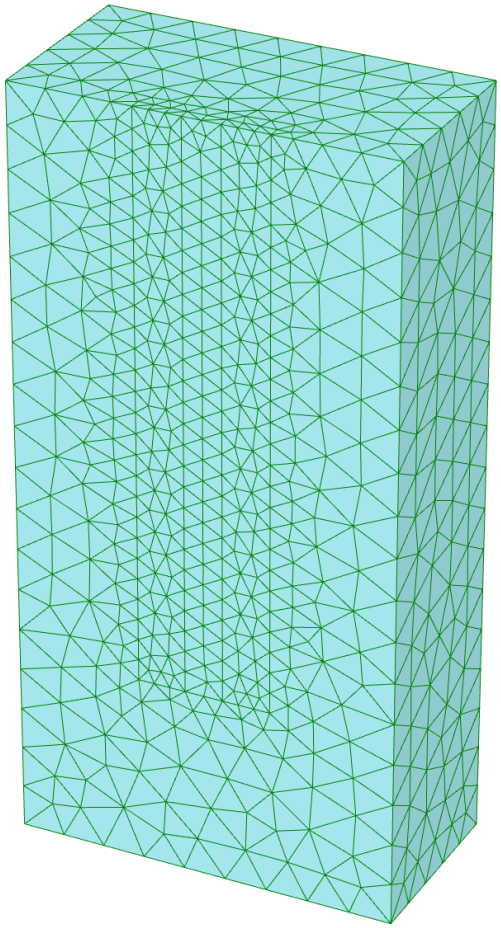


Figure 53: The generated mesh

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

4.6 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/m^3 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.

4.6.1 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.

4.6.2 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 (see Figure 54 (on page 79)), deactivate the soil volume. Set the water condition to **User-defined** and enter z_{ref} = 40 m, p_{ref} = 0.0 kN/m^2 and p_{inc} = -11 kN/m^2/m.

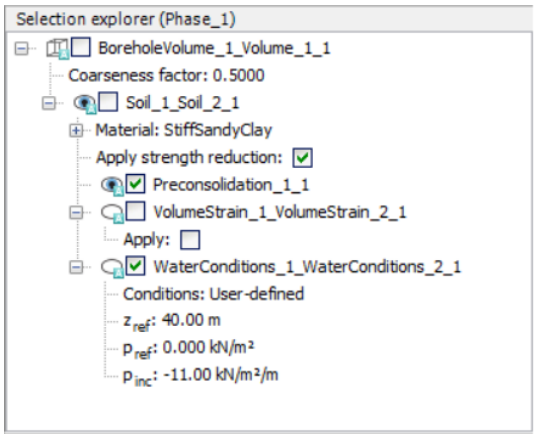


Figure 54: User-defined water condition in part I

A bentonite pressure is now defined in part I of the excavation, starting at 0 kN/m^2 at the reference level of 40 m and increasing at 11 kN/m^2 per m depth, resulting in 330 kN/m^2 at the bottom of the excavation.

4. Click the **Preview phase** button to check the settings for the current phase.

4.6.3 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 m, p_{ref} = 0.0kN/m^2 and p_{inc} = -11 kN/m^2/m.

4.6.4 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 m, p_{ref} = 0.0kN/m^2 and p_{inc} = -11kN/m^2/m.

4.6.5 Phase 4 - Fluid concrete

The bentonite in the excavation is now replaced by fluid concrete with a weight of 24.0kN/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}= -24kN/m^2/m. The other parameters must be kept at their original values (z_{ref} = 40m, p_{ref} = 0.0kN/m^2).

4.6.6 Phase 5 - Cured concrete

1. Add a new phase.
2. Select the three excavation volumes.
3. 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.

4.6.7 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. In the **Deformation control** subtree select the **Reset displacements to zero** option.
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.

4.6.8 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.

4.7 Results

The stability of the excavation can be evaluated from the calculated safety factor after each excavation stage. Use the Curves program to plot ΣMsf (the safety factor) as a function of the displacements |u| (see Figure 55 (on page 82)). In Phase 3, the stability is the lowest. However, ΣMsf 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** tabsheet.
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** > **ΣMsf**. The Safety phases are considered in the chart. As a result, the curve of Figure 55 (on page 82) appears.
5. Set x-axis interval maximum to 0.1 and for y-axis set 7.0 in **Chart** tab.

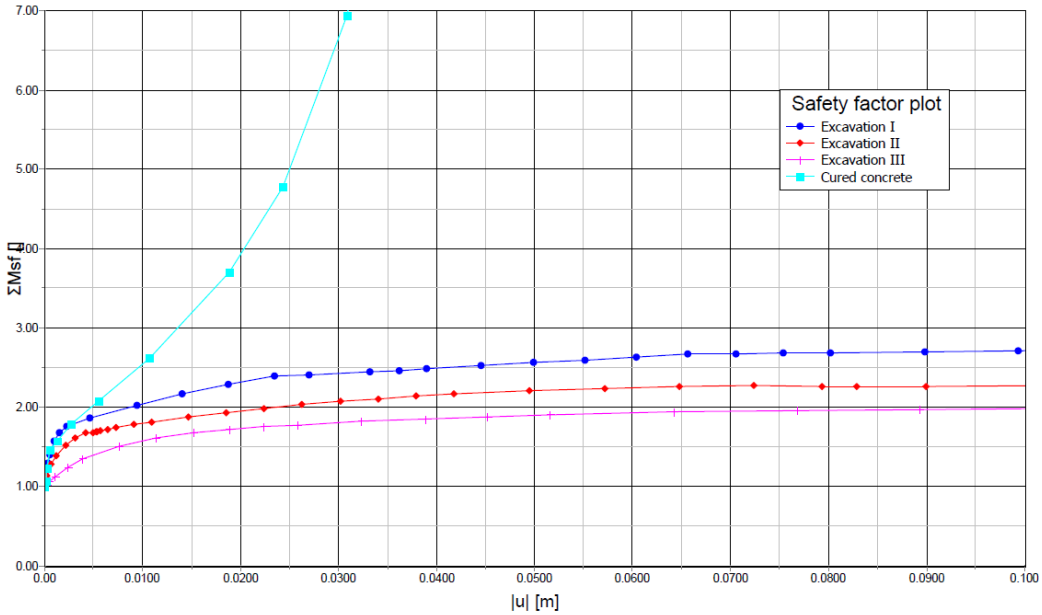


Figure 55: ΣMsf (safety factor) as a function of the total displacement

An important phenomenon that keeps the excavation stable is arching in the soil. This phenomenon is shown in Figure 56 (on page 82), Figure 57 (on page 83) and Figure 58 (on page 83). To see the principal stresses directions at a chosen depth, make a horizontal cross section by clicking the **Horizontal cross section** button.

1. To create such plots, make a horizontal cross-section by clicking the **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 **Stresses** > **Principal total stresses** > **Total principal stresses**.
4. Select the top view in **View** > **Viewpoint** to reorientate the model in order to obtain a clearer view of the arch effect.

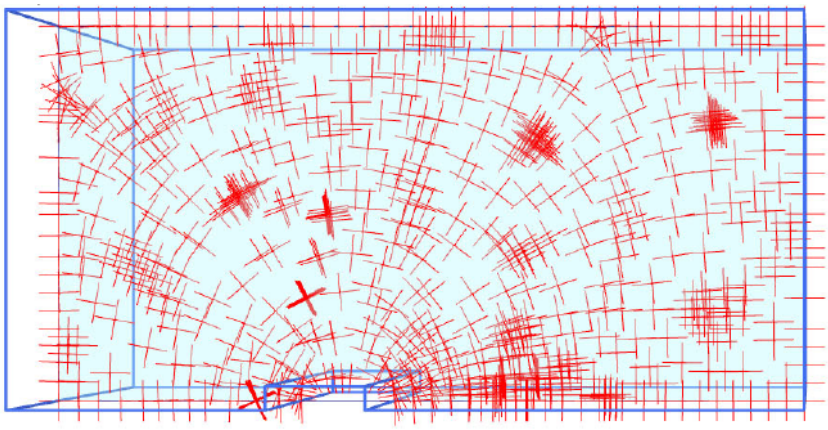


Figure 56: Principal stresses directions at z = 25 m at the end of Phase_1

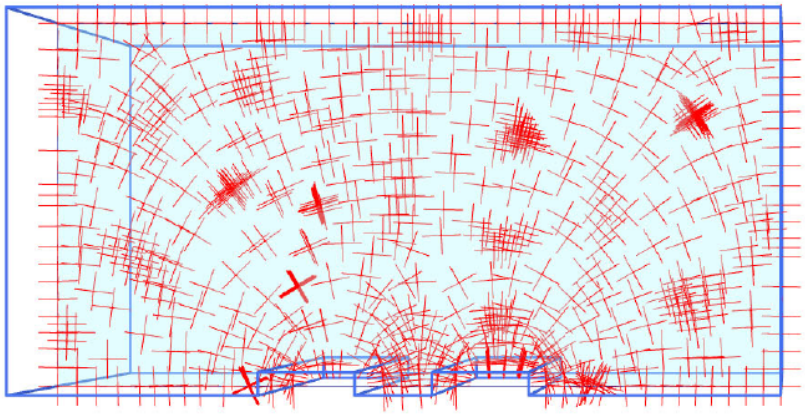


Figure 57: Principal stresses directions at z = 25 m at the end of Phase_2

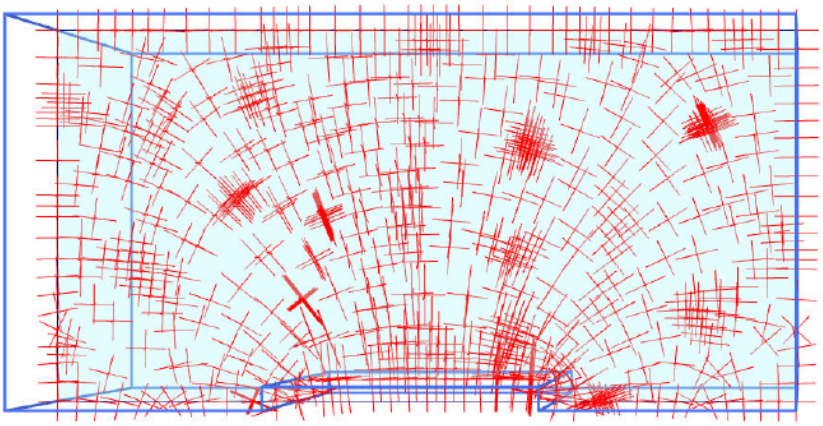


Figure 58: Principal stresses directions at z = 25 m at the end of Phase_4

5 Phased excavation of a shield tunnel [GSE]

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, the 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 (or sections) each 1.5m 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 i*nstallation 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 1.5m 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.

Geometry

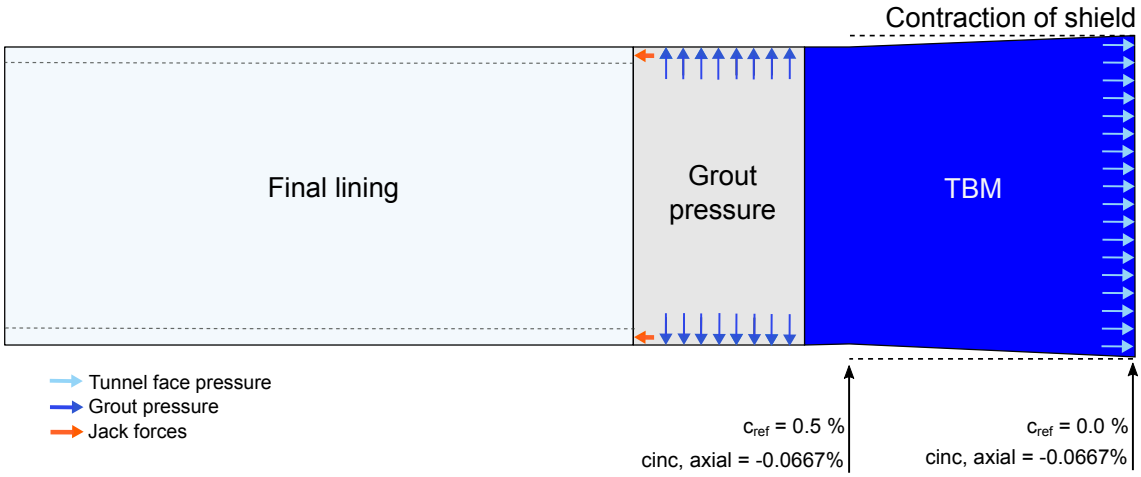


Figure 59: Construction stages of a shield tunnel model

5.1 Create a new project

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_{min} = -20 and x_{max} = 0,
   b. y_{min} = 0 and y_{max} = 80.

5.2 Define the soil stratigraphy

The subsoil consists of three layers. The soft upper sand layer is 2m deep and extends from the ground surface to Mean Sea Level (MSL). Below the upper sand layer, there is a clay layer of 12m thickness and this layer is underlain by a stiff sand layer that extends to a large depth. Only 6m of the stiff sand layer is included in the model. Hence, the bottom of the model is 18m 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 2m and the bottom at 0m, Clay with the bottom at -12m and Stiff sand with the bottom at -18m.

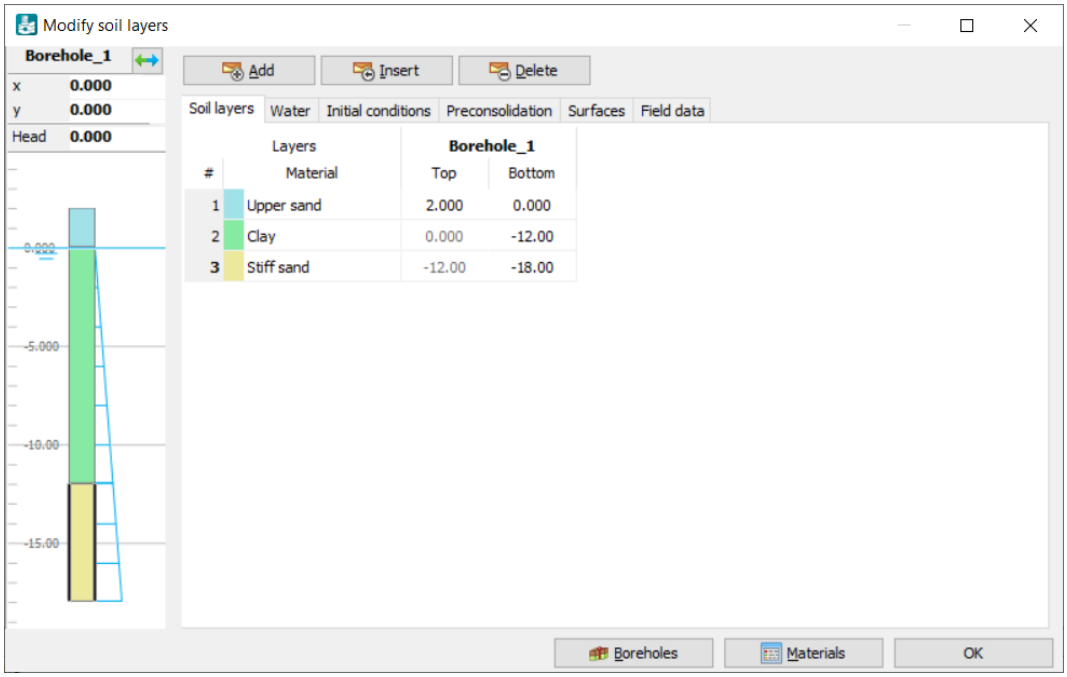


Figure 60: Soil layer distribution

5.3 Create and assign the material data sets

The material properties for the data sets are shown in Table 14 (on page 86).

Table 14: Material properties for the soil layers

Property	Name	Upper sand	Clay	Stiff sand	Concrete	Unit
General						
Soil model	Model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Linear elastic	-
Drainage type	Type	Drained	Drained	Drained	Non porous	-
Unsaturated unit weight	γ_{unsat}	17.0	16.0	17.0	27.0	kN/m^3
Saturated Unit weight	γ_{sat}	20.0	18.0	20.0	-	kN/m^3
Mechanical						
Young's modulus	E'_{ref}	$1.3 \cdot 10^4$	$1.0 \cdot 10^4$	$7.5 \cdot 10^4$	$3.1 \cdot 10^7$	kN/m^2
Poisson's ratio	$\nu(nu)$	0.3	0.35	0.3	0.1	-

Mechanical						
Cohesion	c'_{ref}	1.0	5.0	1.0	-	kN/m^2
Friction angle	$\varphi'(phi)$	31	25	31	-	°
Dilatancy angle	$\psi(psi)$	0	0	0	-	°
Interfaces						
Strength determination	-	Rigid	Rigid	Rigid	Rigid	-
Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-

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 14 (on page 86).
2. Assign the material data sets to the corresponding soil layers (Figure 60 (on page 86))and close the **Modify soil layers** window. The concrete data set will be assigned later.

5.4 Definition of structural elements

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

The material properties for the linning tunnel are presented in Table 15 (on page 87).

Table 15: Material properties of the plate representing the TBM

Property	Name	TBM	Unit
General			
Material type	-	Elastic	-

Property	Name	TBM	Unit
General			
Unit weight	γ	247	kN / m^3
Mechanical			
Isotropic	-	Yes	-
Young's modulus	E_1	200·10^6	kN / m^2
Poisson's ratio	ν_{12}	0	-
Thickness	d	0.17	m
Shear modulus	G_{12}	100·10^6	kN / m^2

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.

5.4.1 Create tunnel cross section

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

1. Click on the **Start designer** button in the side toolbar.
2. Click on 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) as shown in Figure 61 (on page 88).

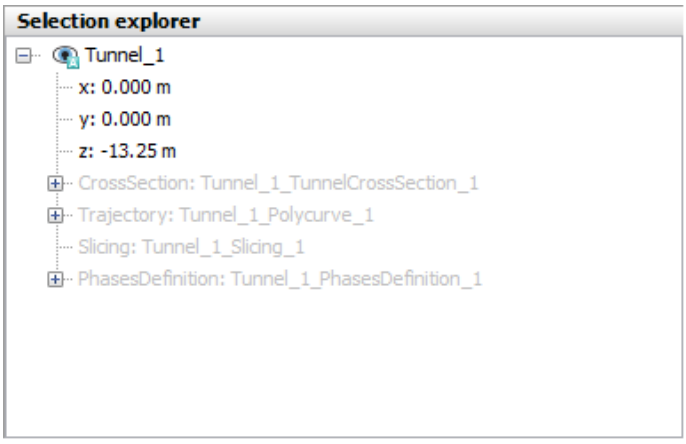


Figure 61: Insertion point of the tunnel

5. In the **General** tabsheet, as **Shape type**, select *Circular* from the drop-down menu.
6. The left half of the tunnel is generated for this example. Hence, im the **Whole or half tunnel** drop-down menu select *Define left half*.

Note:

With the *Offset to begin point* option it is possible to move the insertion point of the tunnel taking the local axes as reference.

A screenshot of the **General** tabsheet after the proper assignment is given in Figure 62 (on page 89).

.

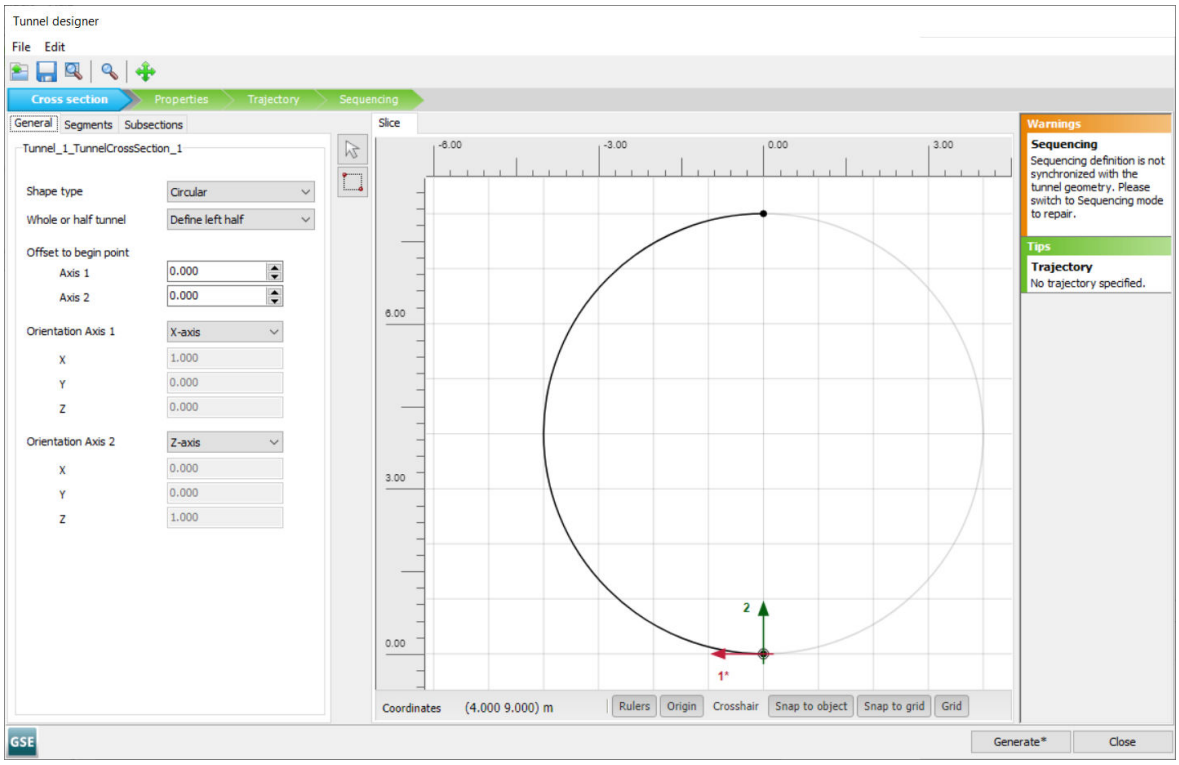


Figure 62: General tabsheet of the Tunnel designer

7. Proceeed to the **Segments** tabsheet to define the radius of the TBM and the thickness of the final lining.

Note:

Notice that the segment created is shown on the *Display area*. Also a new box is now visible in the segment list. By clicking on *Segment[0]* the *Selection Explorer* gets activated that contains the corresponding editable properties of the segment.

8. By having the created segment selected, set the *Radius* to 4 m (see Figure 63 (on page 90)). This is the inner radius of the tunnel.

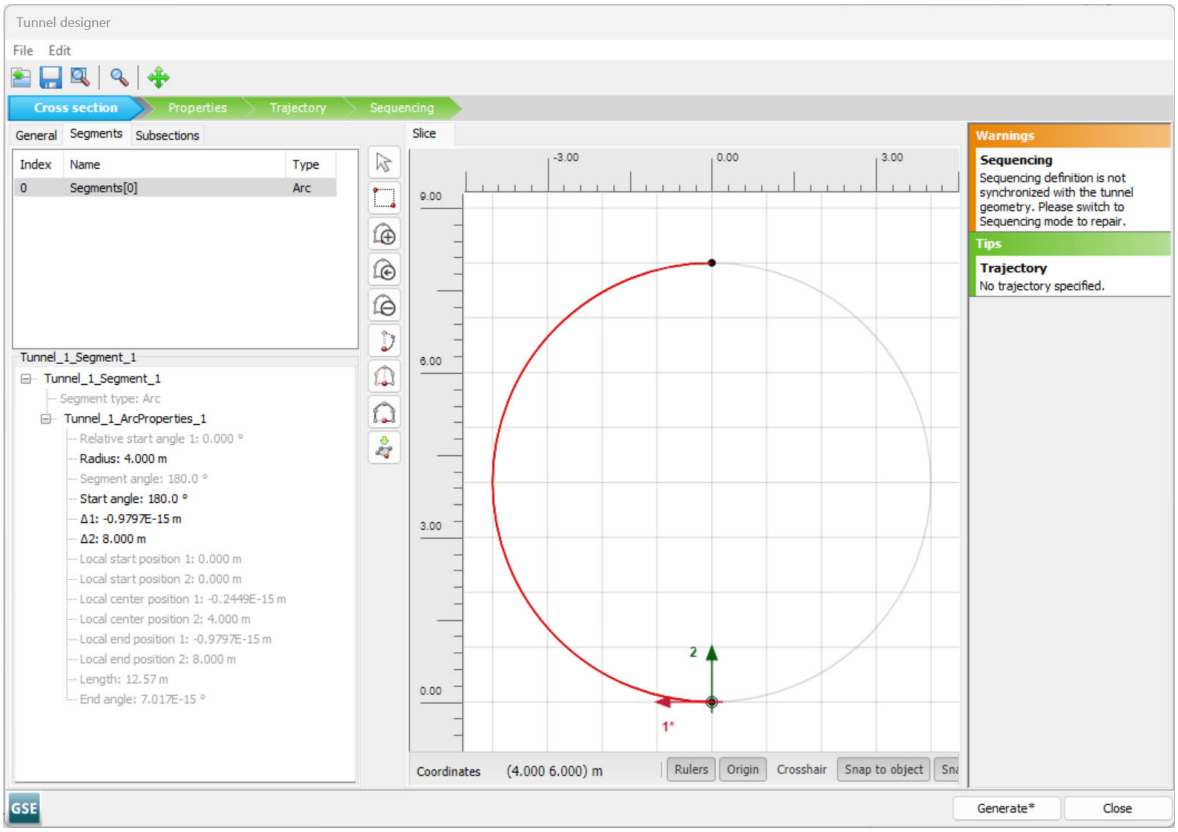


Figure 63: Definition Tunnel Radius

9. Click on the **Subsections** tabsheet.
10. Click on 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**.

The generated lining is displayed on the *Drawing area* in Figure 64 (on page 91).

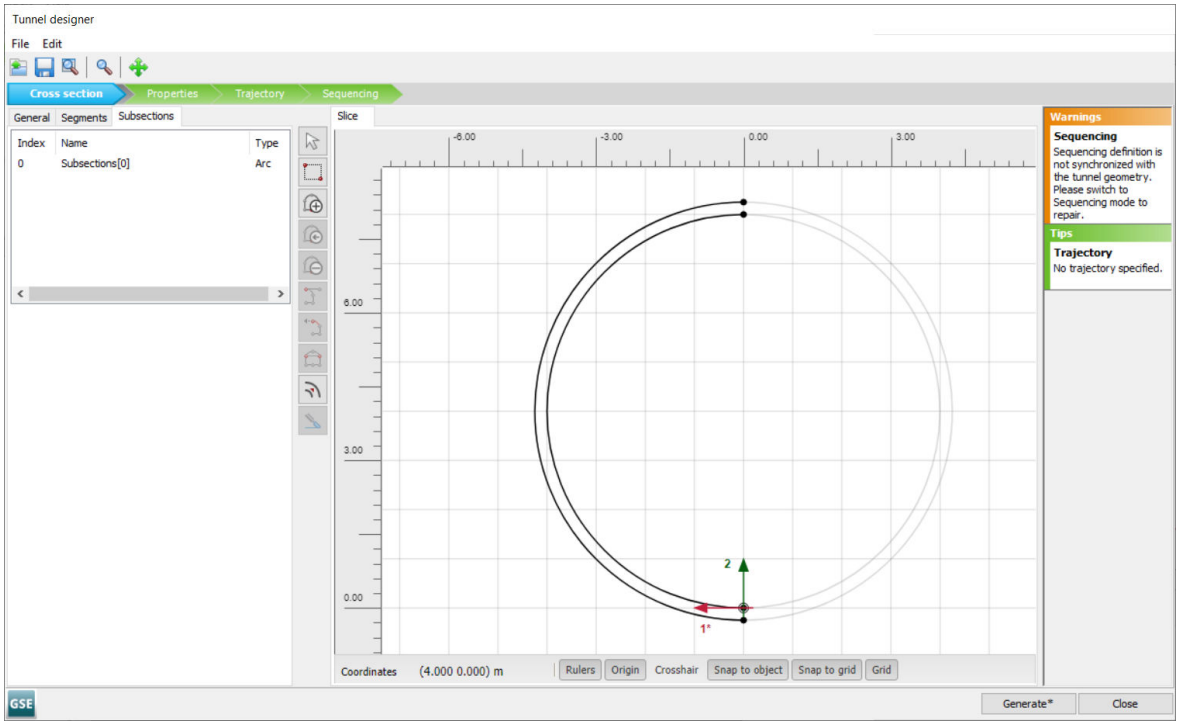


Figure 64: Generated thick lining

5.4.2 Definition of Tunnel properties

Some of the properties that are going to be defined for the tunnel are: the grout pressure, the surface contraction, the jack forces and the tunnel face pressure.

1. Click on the **Properties** tabsheet.
2. Click on the **Slice** tab (above the *Drawing area*) and right-click on the outer surface of the TBM. From the appearing menu select **Create** > **Create plate** (see Figure 64).

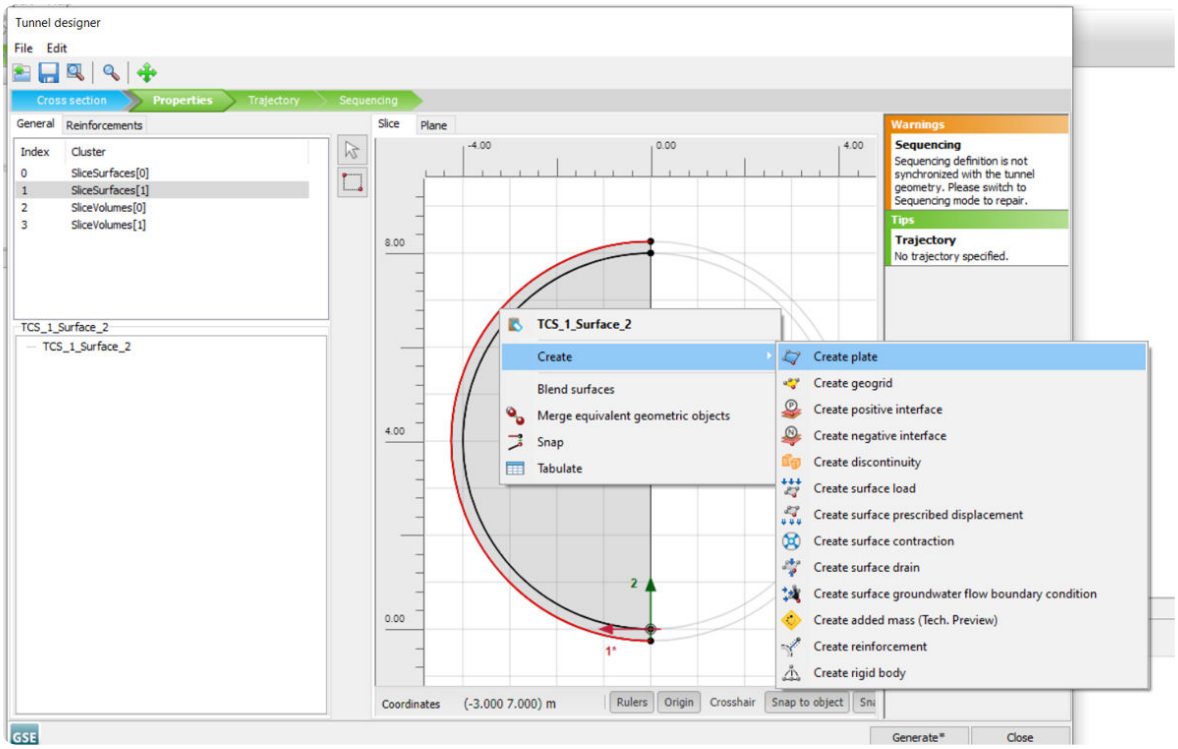


Figure 65: The Properties tabsheet of the Tunnel designer for the creation of a Plate

3. With the plate selected, in the **Selection Explorer>Material** create a new material dataset for the TBM according to Table 15 (on page 87).
4. Assign the created material to the plate.

Surface contraction

A soil-structure interaction must 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 7.5m length of the TBM (5 slices) while the last 1.5m to the tail has a constant diameter.

This means the section at the tail has a uniform contraction of 0.5% and the other 5 sections have a linear contraction with a reference value C_{ref} = 0.5% and an increment $C_{inc,axial}$ = -0.0667%. Take into account that:

- The reference is set on the front surface of the excavated slice in the tunnel during the tunnel construction.
- The surface contraction is applied while defining the tunnel Sequencing (on page 97).
- The $C_{inc,axial}$ = -0.0667%/m and remains the same in every step (see Sequencing (on page 97) <Step_1_1> to <Step_1_5>).

Note:

For further information on **Surface contraction** refer to the Reference Manual .

1. Right-click on the outer slice surface and from the appearing menu select **Create negative interface** to create a negative surface around the entire tunnel.

2. To create **Surface contraction** for the tunnel, right-click again on the outer surface and select **Create** > **Create surface contraction**.
3. While keeping selected the outer surface, in the **Selection explorer** of the tunnel designer select **Surface contraction** > **Distribution** > **Axial increment**. Also define C_{ref} = *0.5*% and $C_{inc,axial}$ = *-0.0667*%/m.

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.
- The increment $C_{inc,axial}$ must be a negative number to account for the volume loss with respect to the front of the excavation.

A view of the created interface and the surface contraction is displayed in Figure 66 (on page 93)

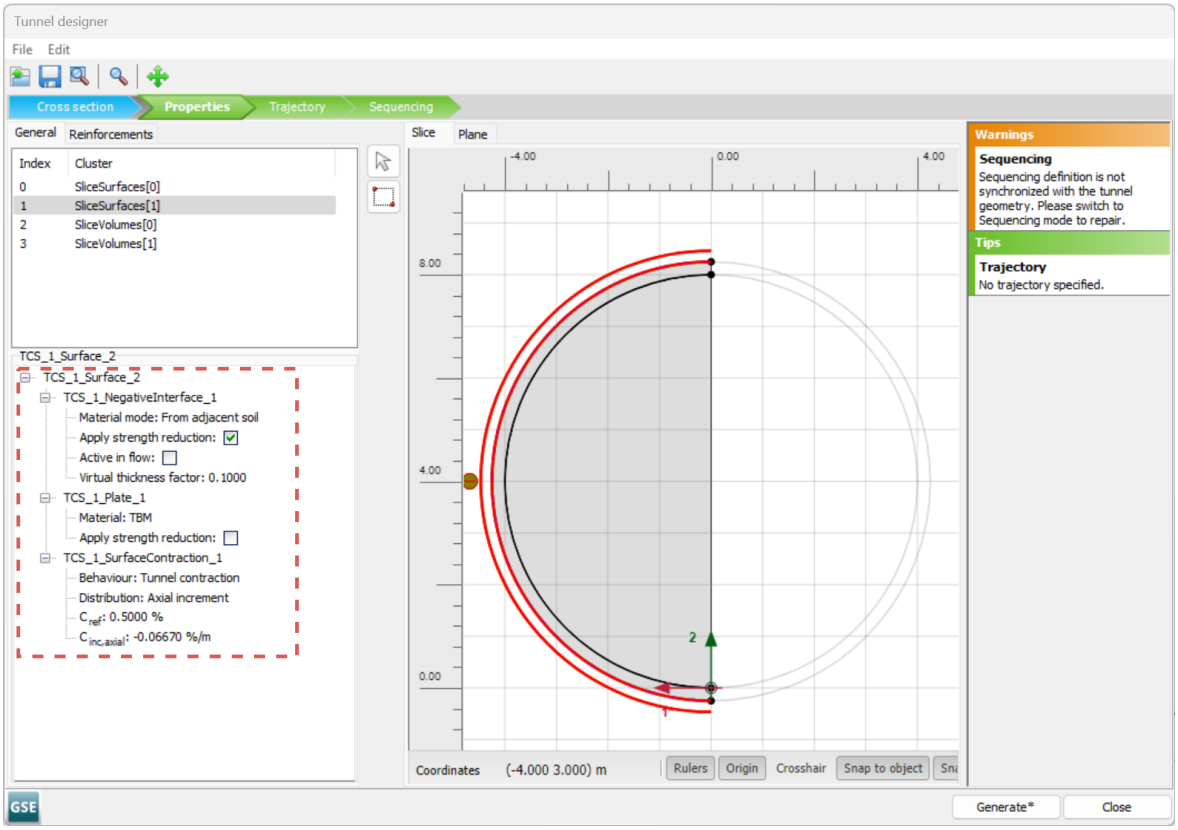


Figure 66: Created negative interface and surface contraction on the tunnel outer slice surface

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 = -4.75m) and should increase with -20 kN/m^2/m depth. To define the grout pressure:

1. Click on the **Slice** tab.
2. Right-click on the outer slice surface and select **Create** > **Create surface load** from the appearing menu to create a surface load around the entire tunnel.
3. In the **Selection explorer** of the tunnel designer, from the drop-down menu select **Surface load** > **Distribution** > **Perpendicular, vertical increment**.
4. As shown in Figure 67 (on page 94) define:
   a. $\sigma_{n,ref}$ to -100 and $\sigma_{n,inc}$ to -20.
   b. (0 0 -4.75) as the reference point for the load by assigning the values to x_{ref}, y_{ref} and z_{ref}.

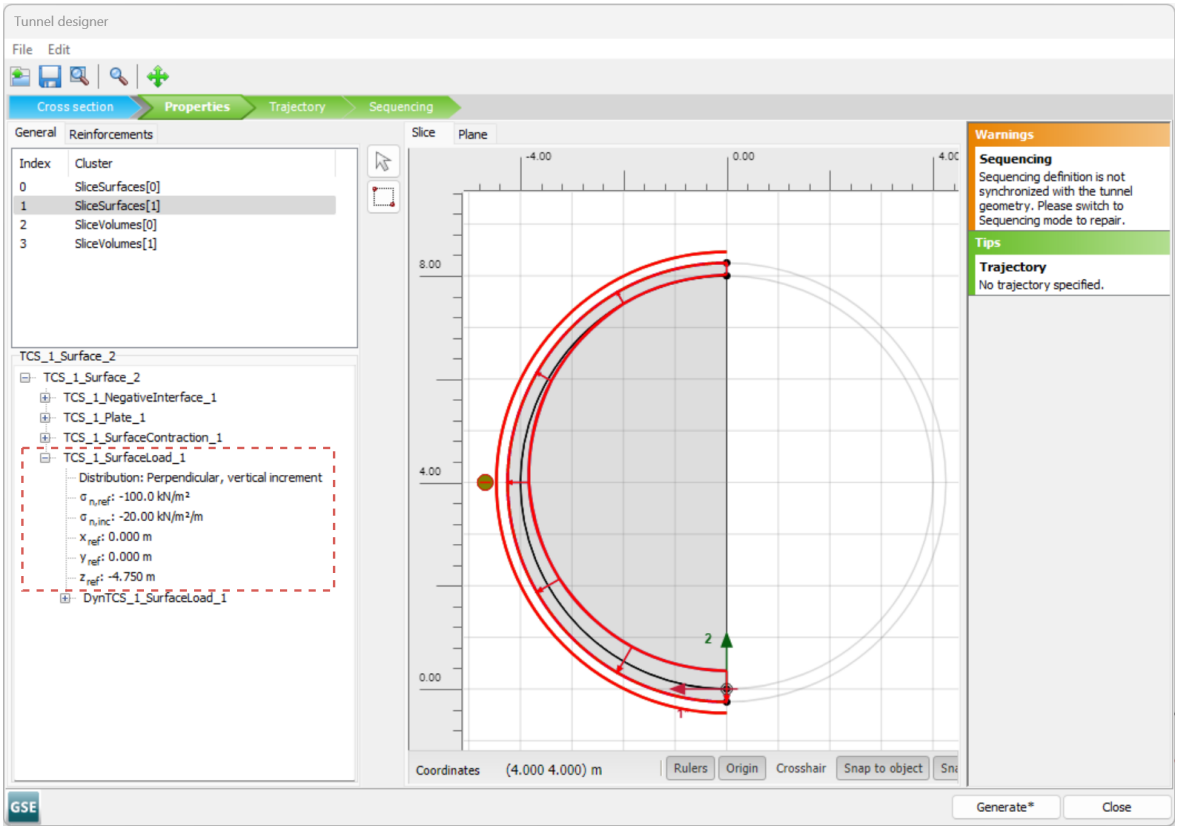


Figure 67: Slice tab Tunnel designer - Grout pressure definition

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. Click on the **Plane** tab located above the Display area with the tunnel cross section.
2. Select both face surfaces (see Figure 68 (on page 95)), followed by pressing right click. Then from the appearing menu select **Create** > **Create surface load** to create a surface load on the TBM face.
3. In the **Selection explorer** go to **Selection** > **Surface Load** and from the drop-down menu select **Distribution** > **Perpendicular, vertical increment**.
4. As shown in Figure 68 (on page 95) set:.
   a. $\sigma_{n,ref}$ to -90 kN/m^2 and $\sigma_{n,inc}$ to -14 kN/m^2/m.
   b. (0 0 -4.75) as the reference point for the load by assigning the values to x_{ref}, y_{ref} and z_{ref}.

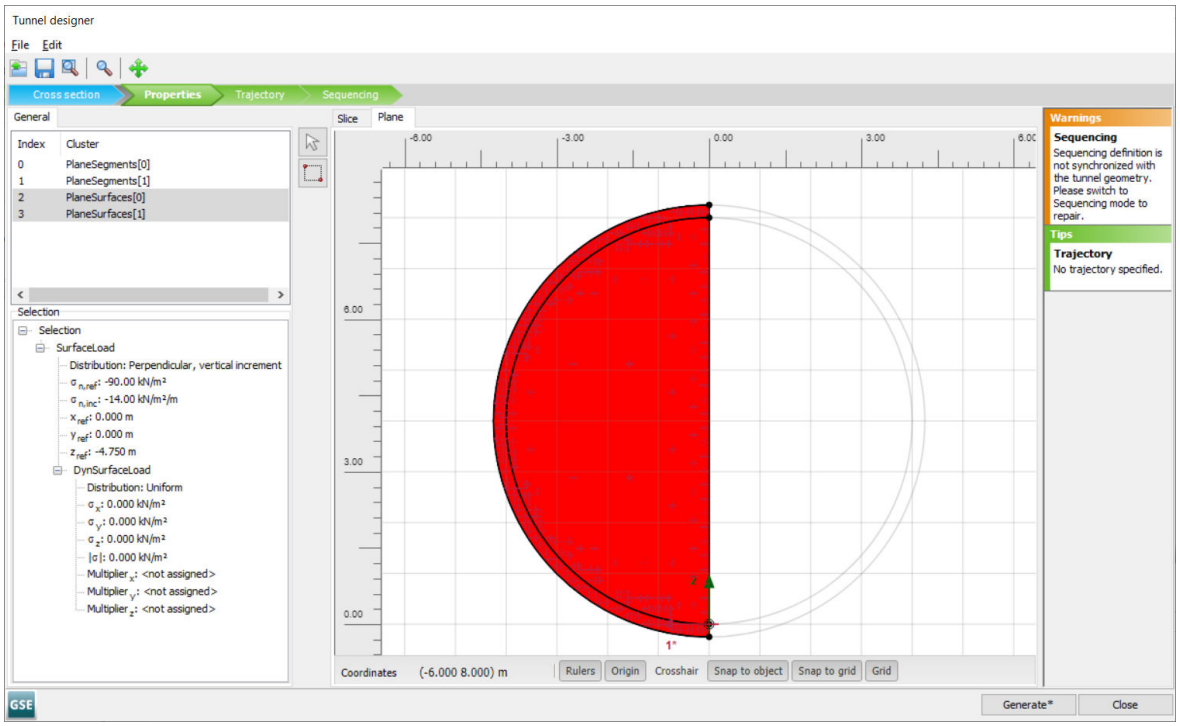


Figure 68: Plane tab Tunnel designer - Definition of TBM face pressure

Jack forces

Note:

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 the *<Step_1_7>* of the tunnel Sequencing (on page 97).

5.4.3 Trajectory

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

1. Click on the **Trajectory** tabsheet to proceed to the corresponding window.
2. In the **Segments** tab, click on the **Add segment** on the left toolbar to create the first segment.
3. In the **Selection explorer** set the length to 25 (see Figure 69 (on page 96)).

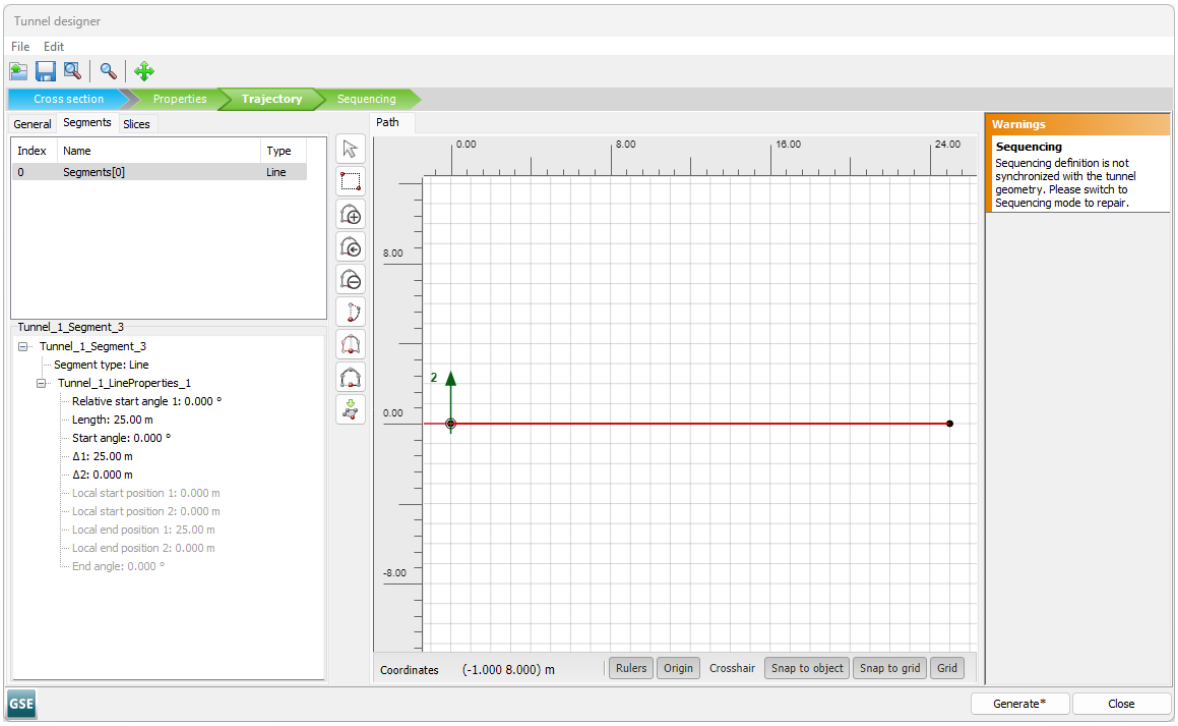


Figure 69: Trajectory tabsheet on Tunnel designer-Adding a trajectory segment

4. Add a second segment and set the *length* to 16.5.
5. Go to the **Slices** tab to create the tunnel slices till 41.5 m and click on the appearing second segment.
6. In the **Selection explorer**, go to the **Slicing method**, chose *Length* and set the **Slice length** as 1.5m (Figure 70 (on page 97)).

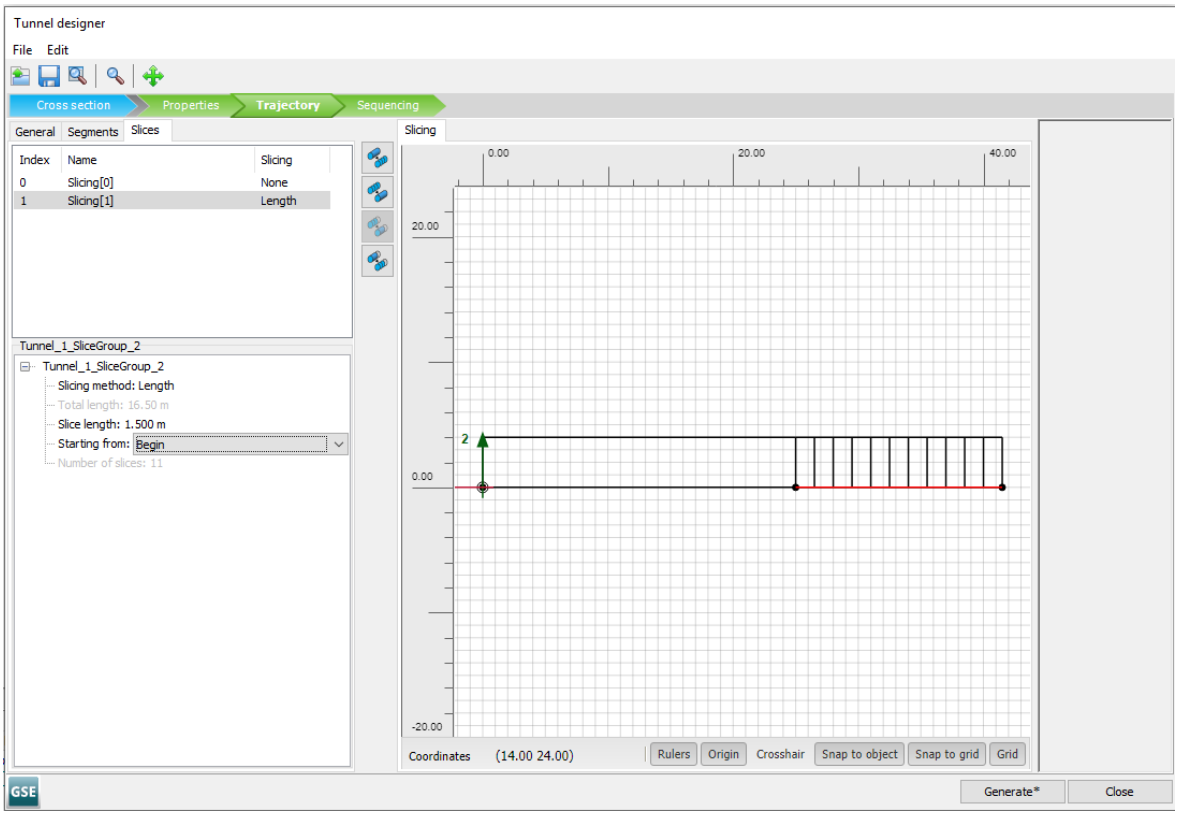


Figure 70: Trajectory tabsheet of tunnel designer-Slicing a trajectory segment

5.4.4 Sequencing

In order to simplify the definition of the phases in the **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. Then at the back of the TBM, the pressure due to the grouting will be modelled. Subsequently, given that it is also required to model a driving force applied by hydraulic jacks on the already installed TBM lining. Finally, a new lining ring will be placed.

1. Click on the **Sequencing** tabsheet to proceed to the corresponding window.
2. In the **Sequencing** tabsheet, under the **General** submenu set the **Excavation method** as *TBM*.

a. *<Step_1_1> Face excavation*

- Click on the **Slice** tab. Notice that below the **General** submenu a first step (i.e., Step_1_1) is now available by default.
- Select the volumes inside the tunnel and in the **Selection explorer**, deactivate the soil volumes and set the **WaterConditions** to *Dry* as displayed in Figure 71 (on page 98).

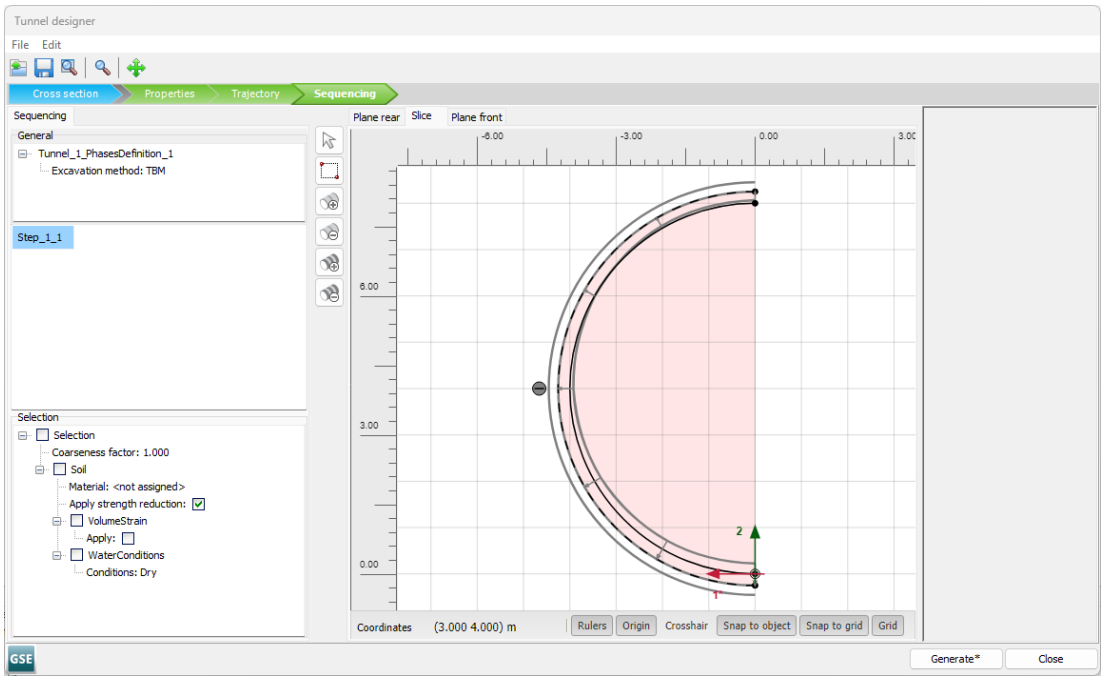


Figure 71: Slice from the face excavation - Deactivated soil volumes

- Staying on the **Slice** tab, select the outer surface. Then go to the **Selection explorer** and activate: the negative interface, the plate and the surface contraction (Figure 72 (on page 99)).
- For the surface contraction C_{ref} set a value of 0% (since this is on the front of the excavation).

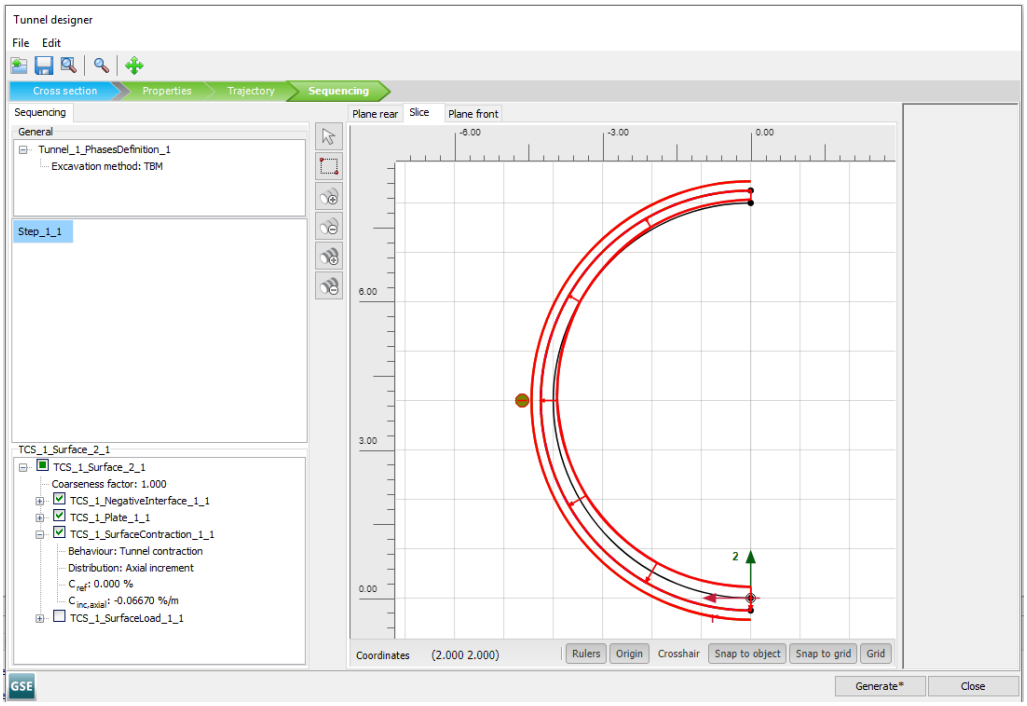


Figure 72: Slice tab Tunnel designer <Step_1_1> TBM activation and definition of surface contraction

- Click on the **Plane Front** tab and select all the surfaces. In the **Selection Explorer**, activate the **Surface load** corresponding to the TBM face pressure (Figure 73 (on page 99)).

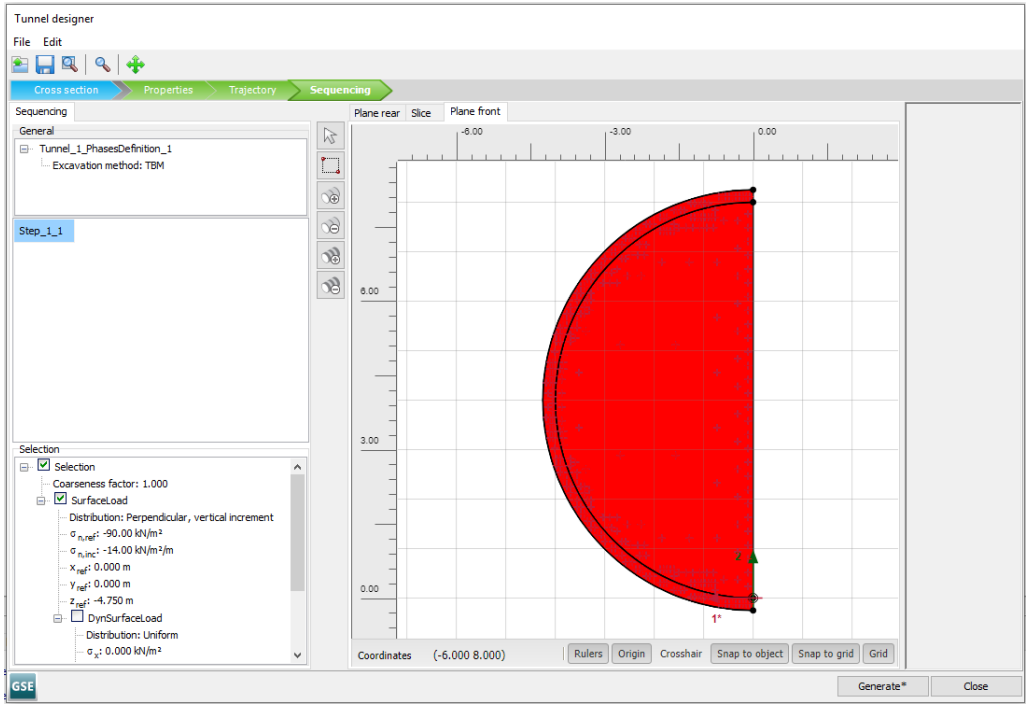


Figure 73: Plane front tab Tunnel designer- <Step_1_1> Definition of face pressure

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

- Continue in the **Plane front** tab.
- Click on the **Add Step** button to add a new step.

  Note: The difference with the previous step is that the load corresponding to face pressure gets deactivated by default (Face pressures are only considered in the front of the TBM).
- Go to the **Slice** tab, select the outer surface and set C_{ref} = 0.1%, see Figure 74 (on page 100).

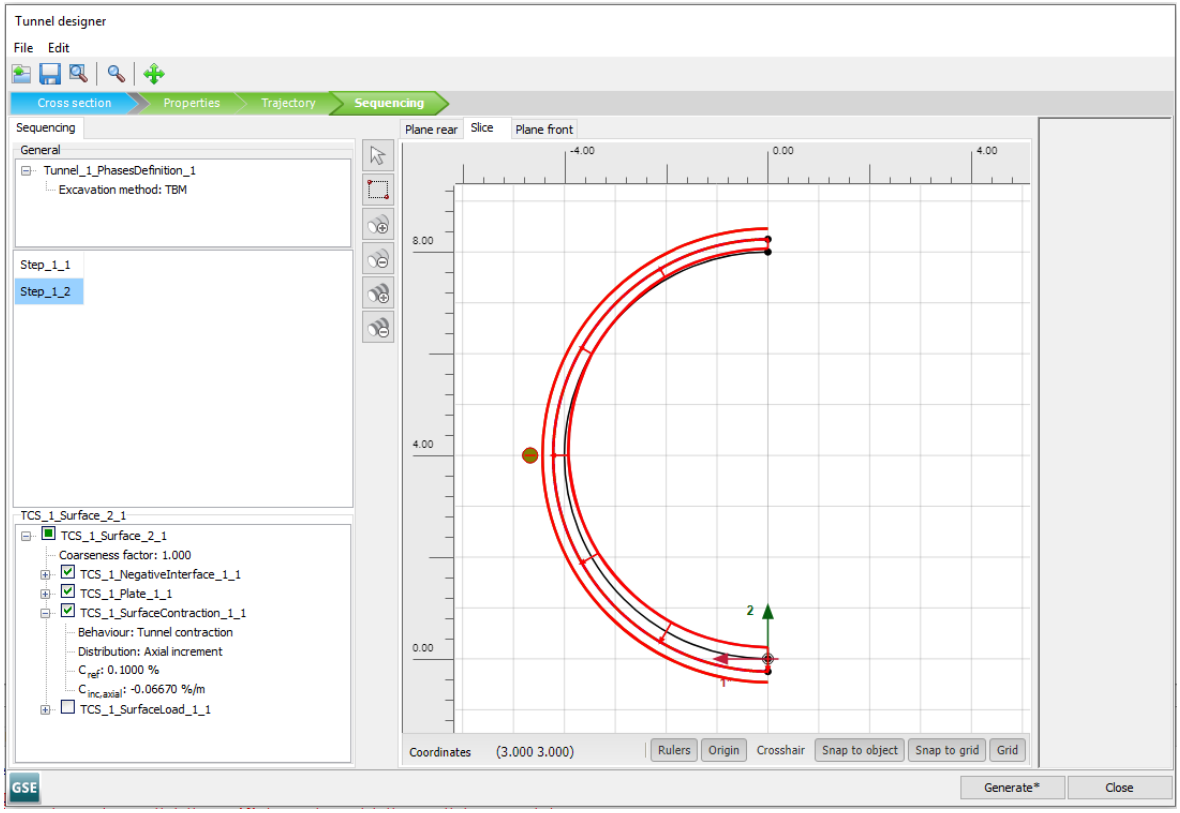


Figure 74: Slice tab Tunnel designer - <Step_1_2>Definition of Surface contraction

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 75 (on page 101)).
- For each step verify the **Slice** tab is chosen, select the outer surface and in the **Selection explorer** set the following values for the surface contraction:

  Step_1_3: C_{ref}= 0.2%

  Step_1_4: C_{ref}= 0.3%

  Step_1_5: C_{ref}= 0.4%, see Figure 75 (on page 101)

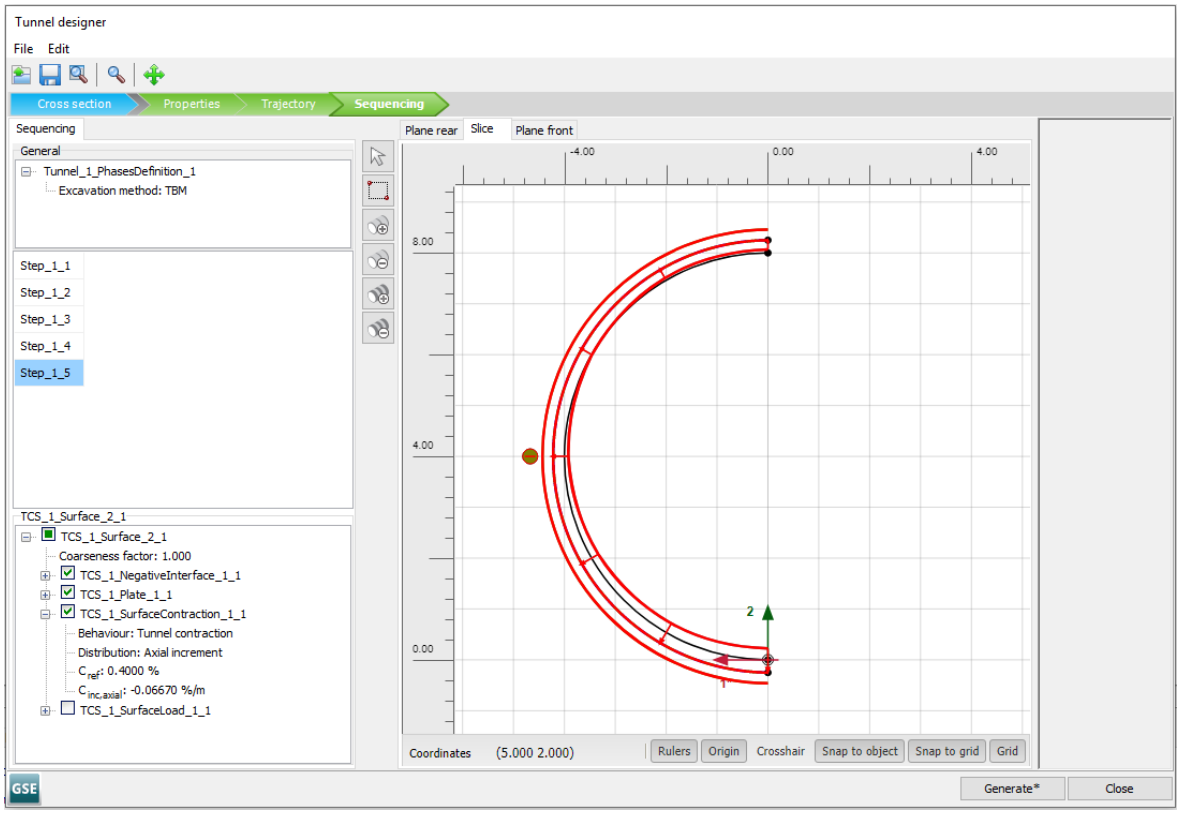


Figure 75: Slice tab Tunnel designer <Step_1_3 to Step_1_5>Definition of surface contraction

d. *<Step_1_6> Tail of the shield*

- Click on the **Add Step** button.
- The last slice of the shield has a constant diameter. From the **Slice** tab select the outer surface.
- In the **Selection explorer** select the **Surface contraction** > **Distribution** > **Uniform** option with C_{ref} = *0.5*% (Figure 76 (on page 102)).

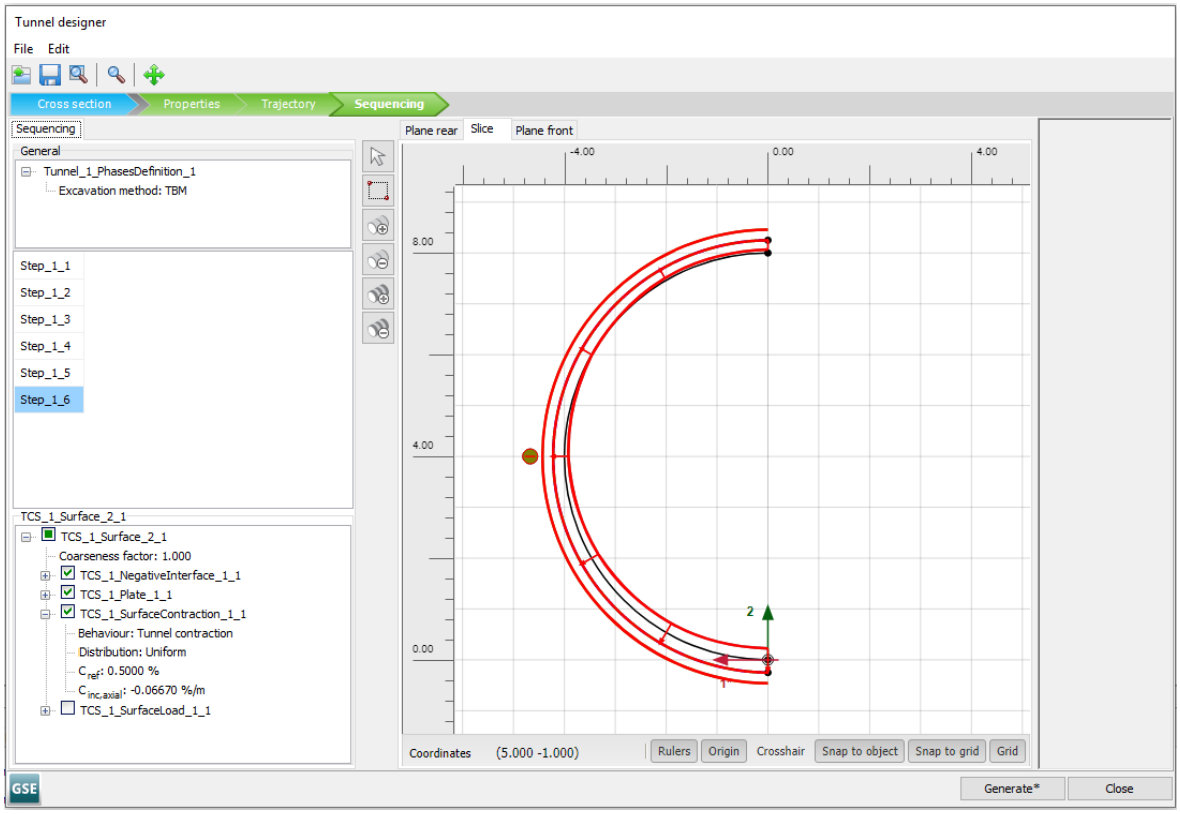


Figure 76: Slice tab Tunnel designer <Step_1_6> Definition of surface contraction

e. *<Step_1_7 > Grouting and jack thrusting*

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. Also the grout pressure is applied.

- Click on the **Add Step** button.
- From the **Slice** tab select the outer surface.
- Deactivate the negative interface, the plate and the surface contraction.
- In the **Selection explorer**, activate the **Surface load** corresponding to the *grout pressure* (Figure 73).

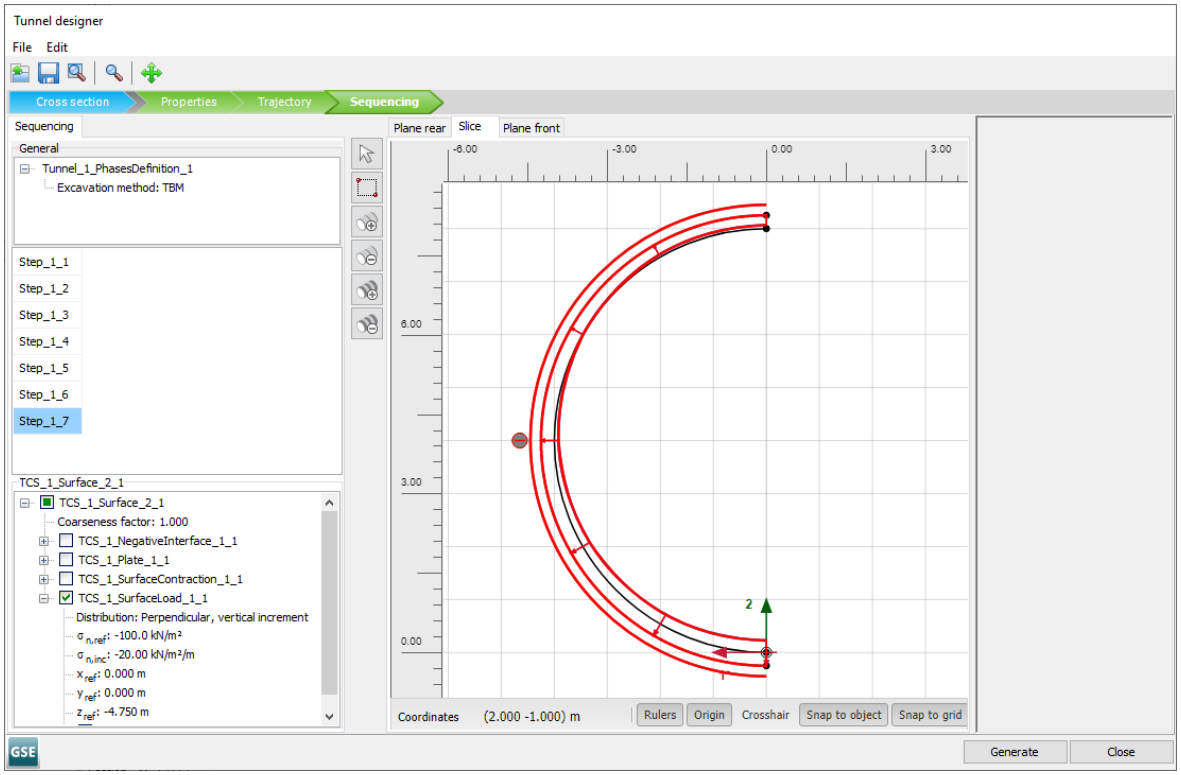


Figure 77: Slice tab Tunnel Designer <Step_1_7> Grout pressure TBM

- Click on the **Plane rear** tab and select the outerrear-plane surface (i.e, the created thick lining) to define the jack thrusting against the final lining.
- In the **Selection explorer**, activate the **Surface load** and select the *Perpendicular* option for the distribution with $\sigma_{n,ref}$ = 635.4 kN/m^2 (Figure 78 (on page 104)).

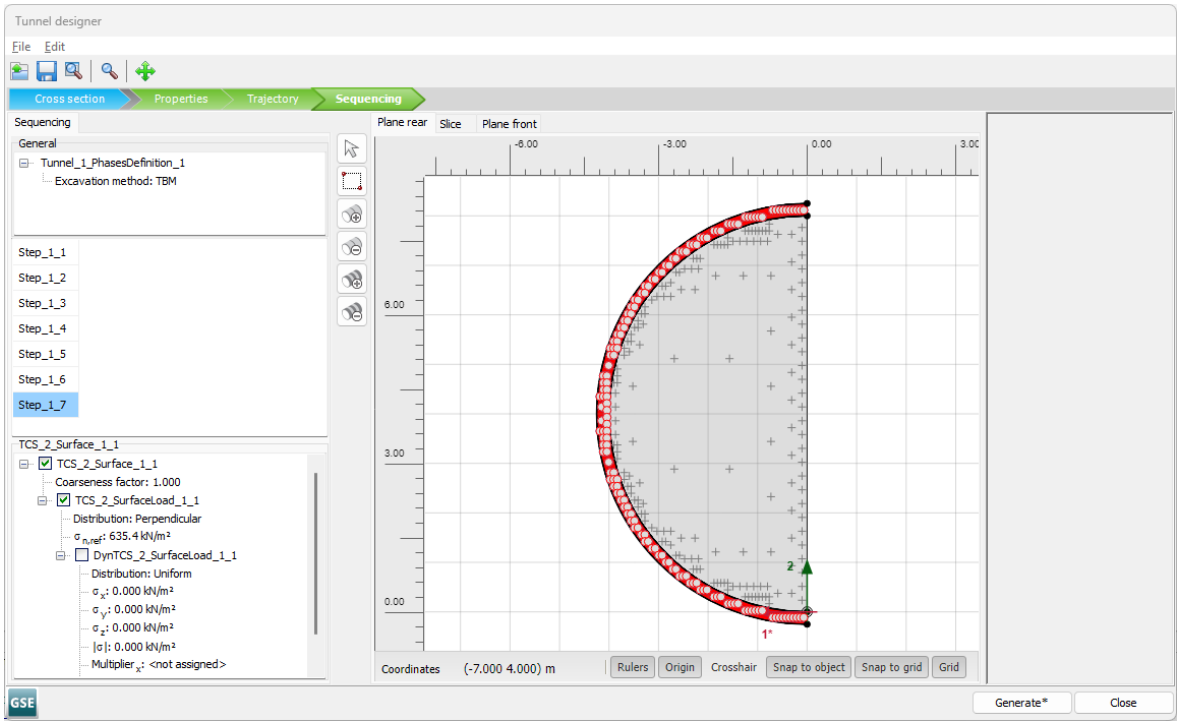


Figure 78: Plane Rear tabsheet Tunnel designer <Step_1_7> Application of jack forces

f. *<Step_1_8 > Final lining*

- Click on the **Add Step** button.
- Go to the **Slice** tab and select the outer surface.
- In the **Selection explorer**, deactivate the surface load corresponding to grout pressure and activate the negative interface.
- Continuing in the **Slice** tab again, select the outer volume (or lining). Activate it and in the **Selection explorer** click on the **Material** drop-down list and select the *Concrete* option (Figure 79 (on page 105)).

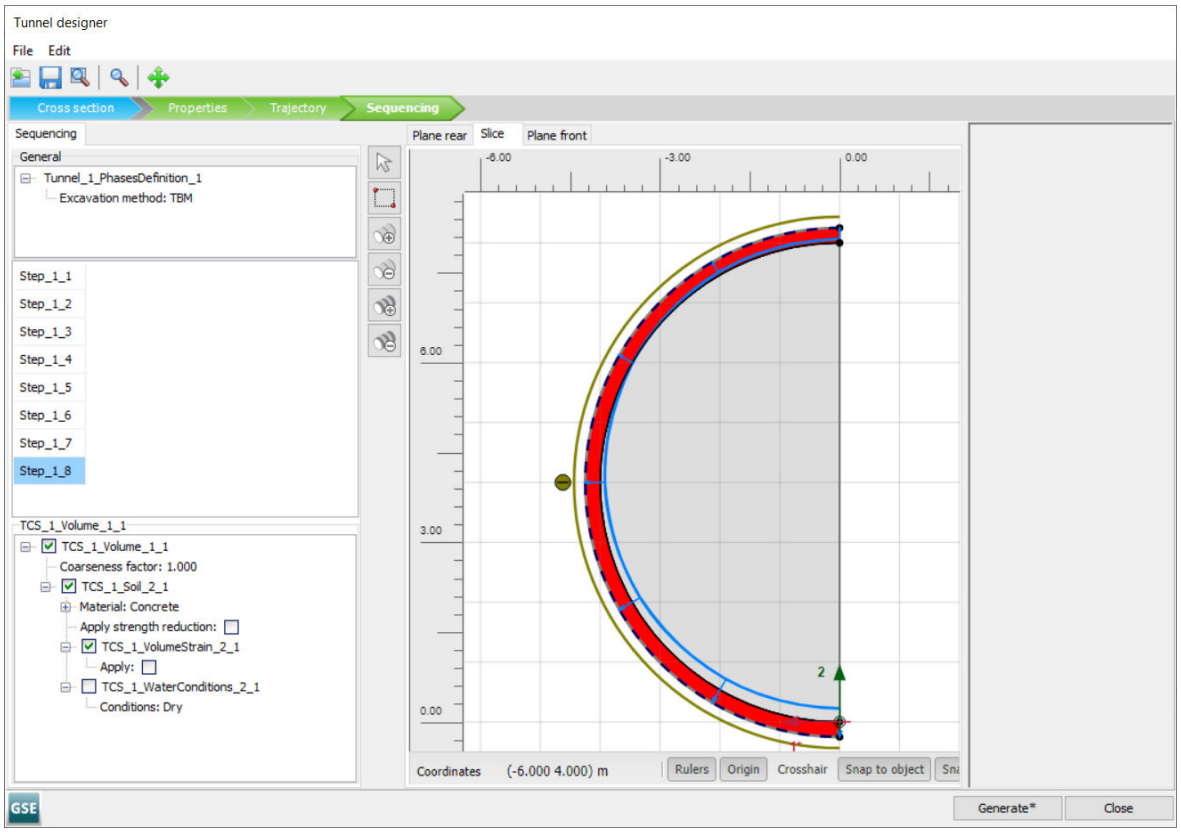


Figure 79: Slice tab Tunnel designer <Step_1_8> Material definition for final lining

- Click on the **Plane rear** tab and select the rear outer plane surface (or back area of the lining).
- In the **Selection explorer**, deactivate the surface load corresponding to the thrusting jacks (Figure 80 (on page 106)).

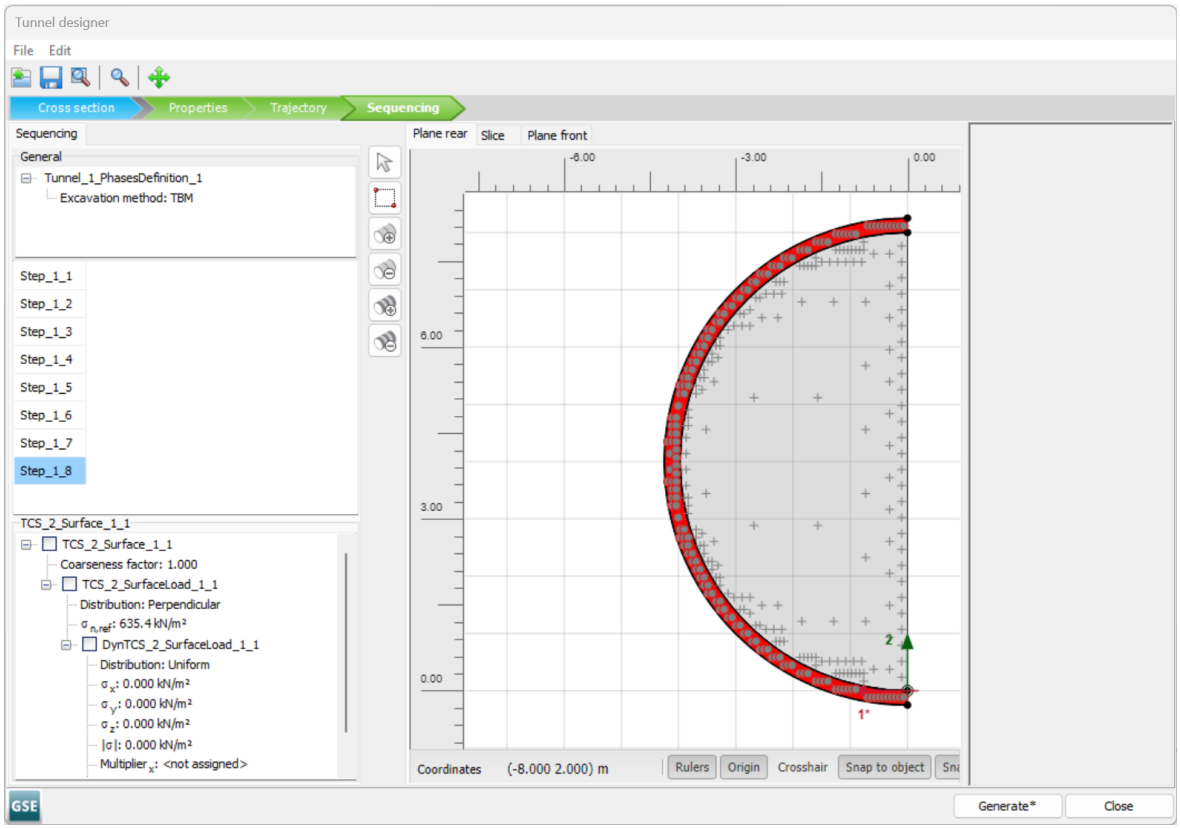


Figure 80: Plane rear tab Tunnel designer <Step_1_8> Deactivation of jacks forces

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. Close the **Tunnel designer** window.
   >The model is created in the **Structures mode**. Click on the **Options** menu and then select **Show local axes on surfaces with structures** this is shown on Figure 81 (on page 107).

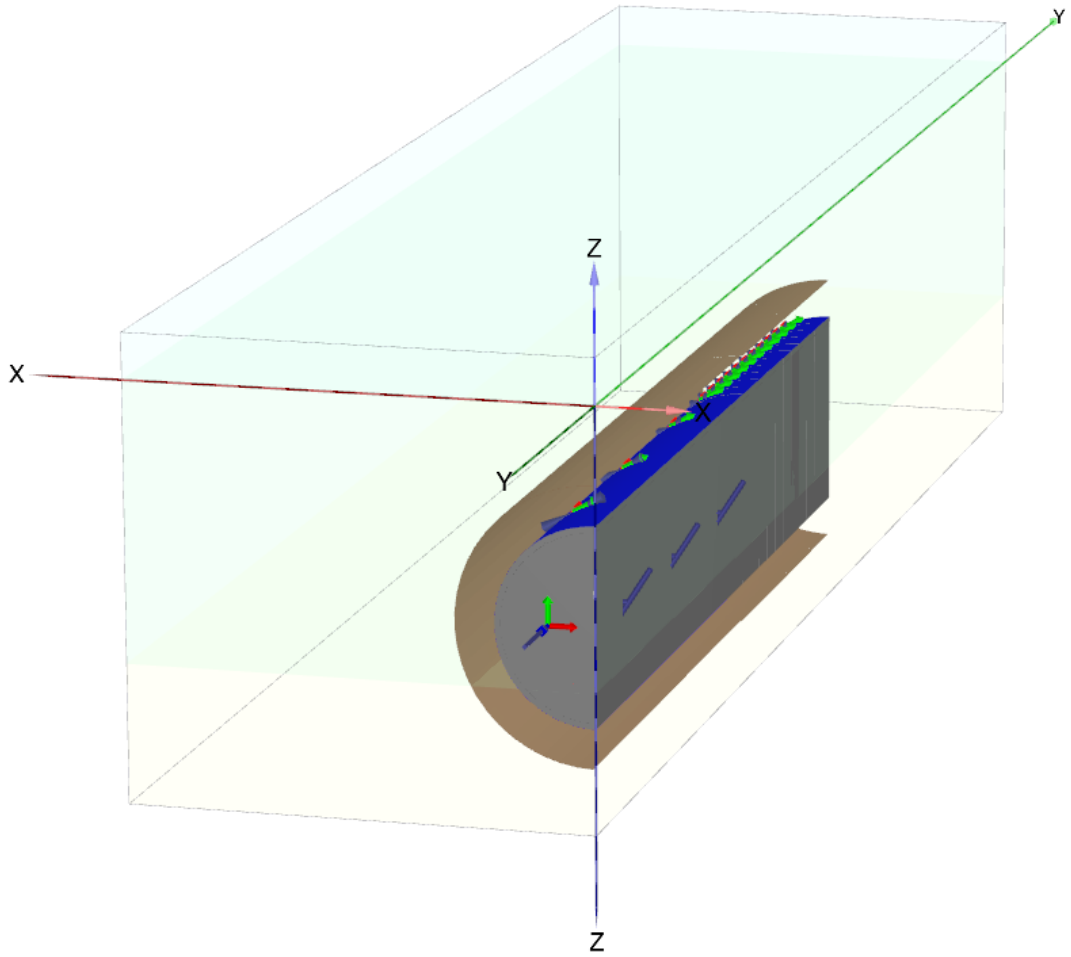


Figure 81: The created tunnels with local axes in the Structures mode

Note:

It is possible to edit the tunnel after being generated. To do this, in the **Structures** mode go to the **Model Explorer**, expand the *Tunnels tree*, do right click on the tunnel of interest, and from the pop up menu select the **Edit** option.

5.5 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 82 (on page 108)).

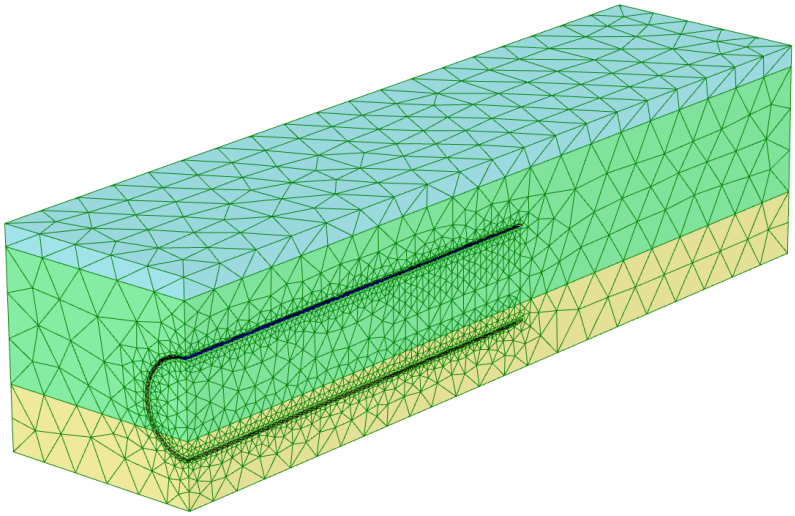


Figure 82: The generated mesh

After inspecting the mesh, the output window can be closed. Mesh generation has now been finished, and so creating all necessary input for defining the calculation phases has been finished.

5.6 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.

5.6.1 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.

1. From the Menu toolbar select **View** > **Default views** > **Right view** to reorientate the model and obtain a clearer perspective of the of the tunnel as displayed in Figure 83 (on page 109).

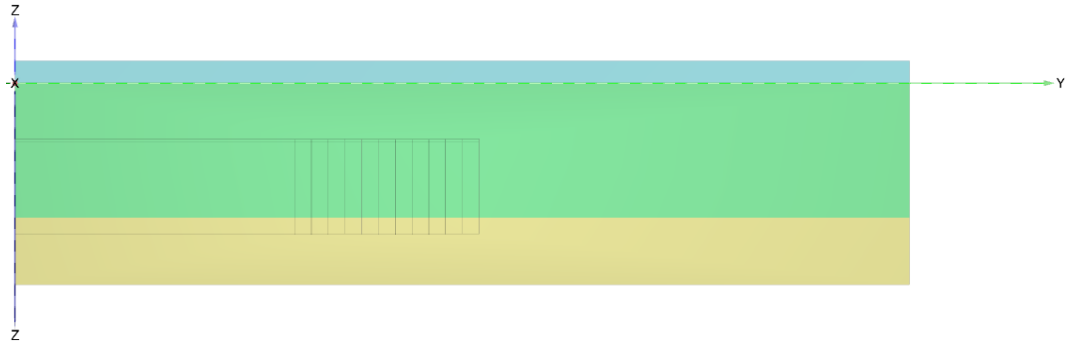


Figure 83: Pre-excavated Tunnel contour

5.6.2 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 where 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 until the option **Advancement step** and set it to 7 in order to simulate the advancement of the first 25 m. Notice that the final lining is automatically activated (see Figure 84 (on page 109)).

Note:

An Advancement step with a value of -1 represents the soil without tunnel excavation (Initial phase). In turn, for this tutorial, the Advancement step 7 represents the position of the rear of the TBM at 25 m; in this position the jack forces will be applied on the ring lining installed between 23.5-25 m).

From the Advancement step 8, the front of the TBM will advance another 1.5 m, leaving a new 1.5m lining ring installed(between 25 - 26.5 m) next to the rear of the TBM..

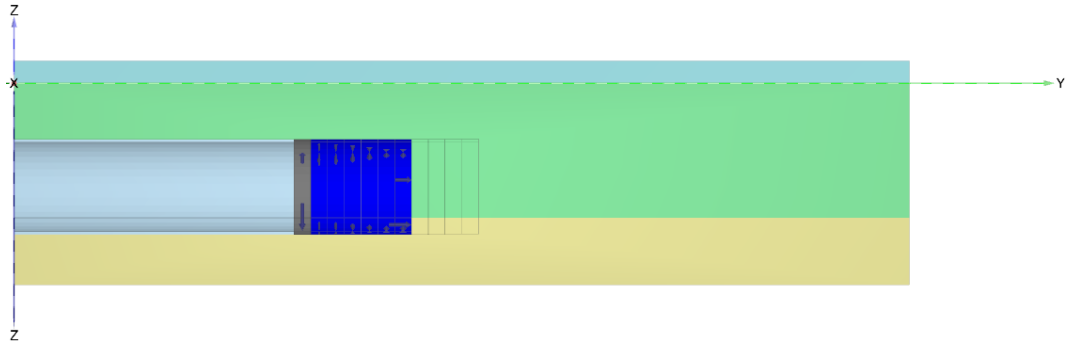


Figure 84: Advancement tunnel after the first 25m

3. In order to consider the conicity of the TBM for the first 25 m, it is necessary to: deactivate the final lining clusters, activate the plates representing the TBM, and apply a surface contraction of 0.5%.
   a. In the drawing area select the soil volumes representing the final lining of the first 25 m of tunnel and deactivate them (Figure 85 (on page 110))

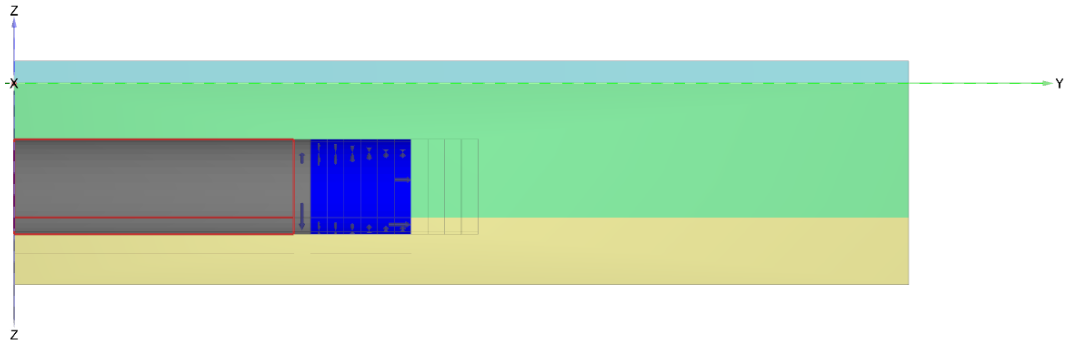


Figure 85: Deactivated soil volumes corresponding to final lining(0 m - 25 m), wireframe view

Notice that the lining is switched off, but the wireframe representing the deactivated volume is still coloured red as the deactivated lining 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 a non-active object can be adjusted in the **Visualization settings** window (see Reference Manual - Modifying the display settings.)

4. Select the outer tunnel surfaces between 0 m and 25 m (inside the wireframe lines) as shown in Figure 86 (on page 110).

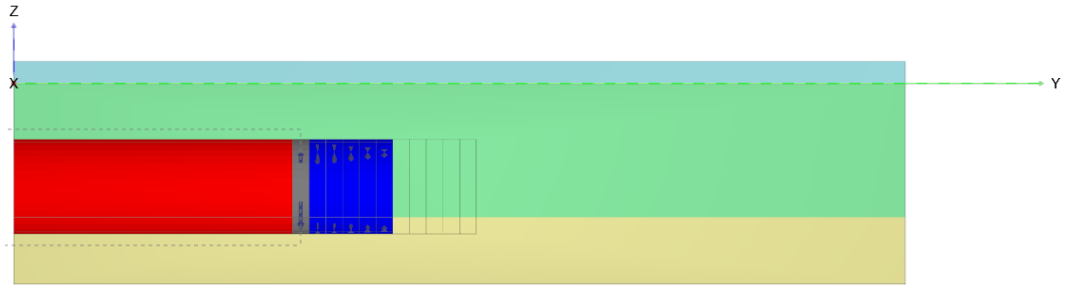


Figure 86: Selection of plate (0 m - 25 m)

5. In the **Selection explorer**:
   a. Check that the interface around the tunnel is activated. Notice that this is done by default.

Tip: If necessary, for a better visualisation of the active interfaces hide the clay and sand soil volumes.

   b. Activate the **Plate** and the **Surface Contraction** by clicking on the corresponding check boxes (see Figure 87 (on page 111)).

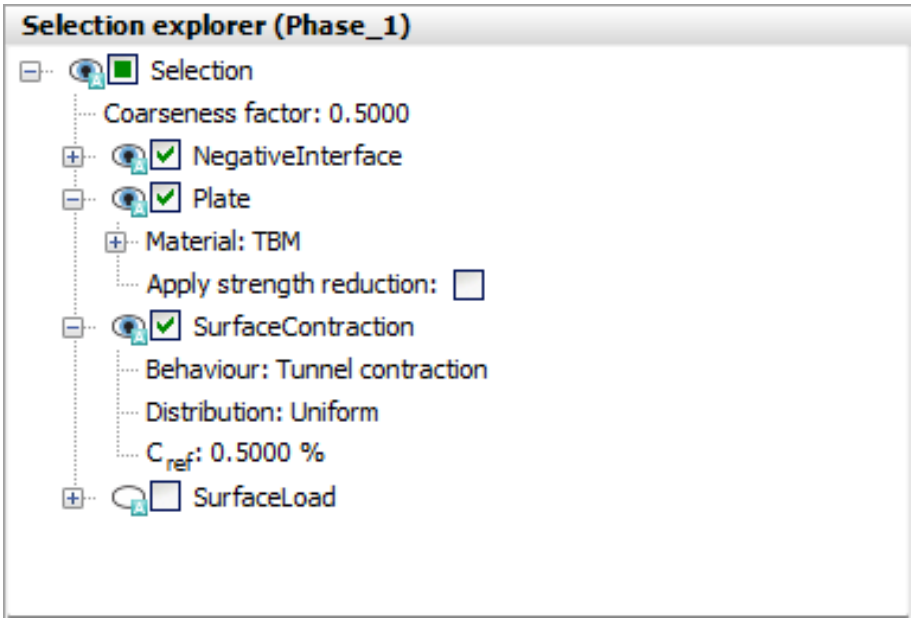


Figure 87: Phase 1 -Activation of plate and surface contraction in the tunnel 0 - 25 m

6. In the drawing area select the lateral surfaces between 0 to 25.0 m at the rear of the TBM lining as displayed in Figure 88 (on page 112). Then in the **Selection explorer**, deactivate the **Surface load** by deselecting the corresponding check box.

Note:

Notice that this done because the TBM is only placed in this phase and it is not moving, therefore jack thrusting is not taken into account here.

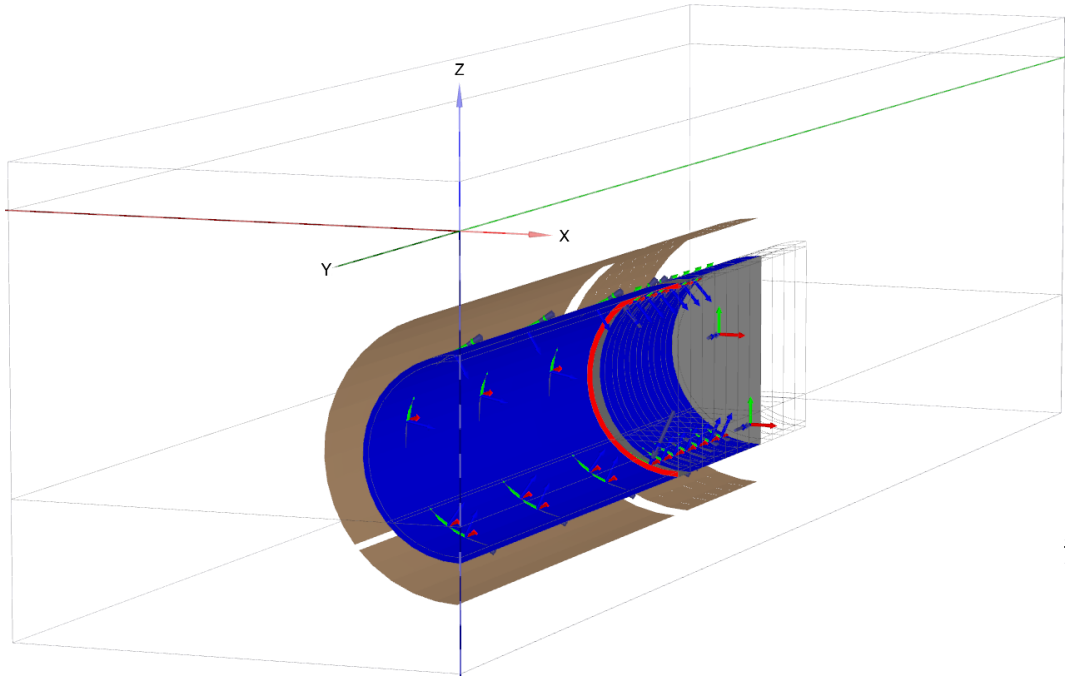


Figure 88: Deactivation of jack forces (25.0 m) for non-moving TBM

7. Click the **Preview** button to get a preview of the excavation up to the first 35.5 m (Figure 89 (on page 112)). Make sure that both grout pressure and tunnel face pressure are applied and that both increase from top to bottom.

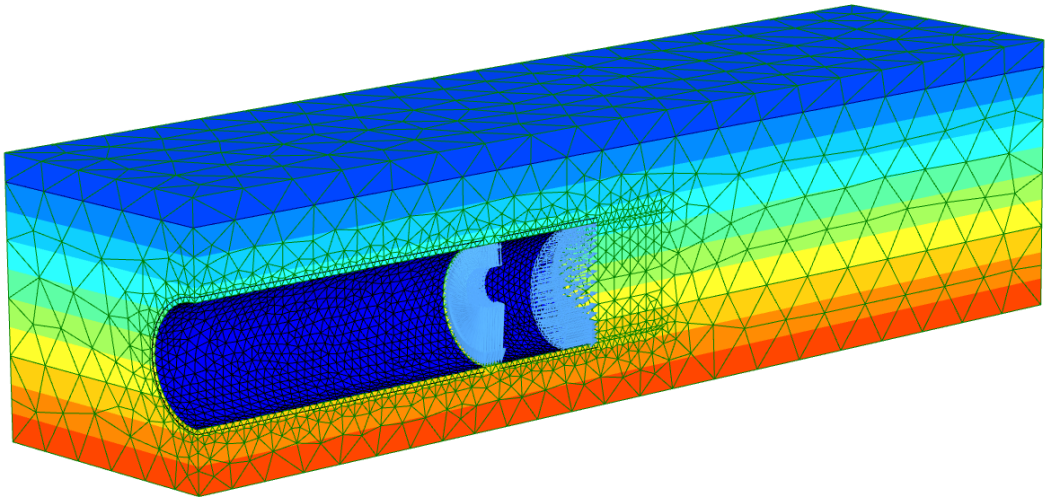


Figure 89: Preview of the Phase 1

Note: In the following phase, the final lining will be automatically reactivated.

5.6.3 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.

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. Alternatively, the Advancement step can be executed as displayed in Figure 90 (on page 113).

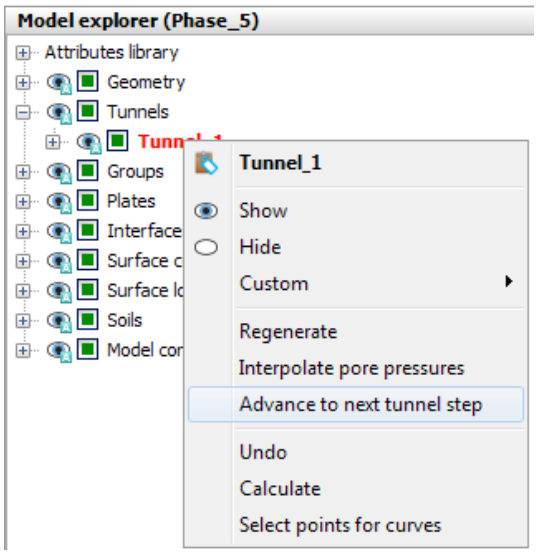


Figure 90: The Advance to next tunnel step option from Model explorer

Note: By default after Advancing to the next tunnel step, the final lining, in the fist 25 m, gets reactivated by default while the TBM plate and the surface contraction get automatically deactivated.

At the end of this phase, the TBM will excavate between 35.5 to 37 m (in the front) and a new lining section will be installed at the TBM rear between 25 to 26.5 m (see Figure 91 (on page 114)).

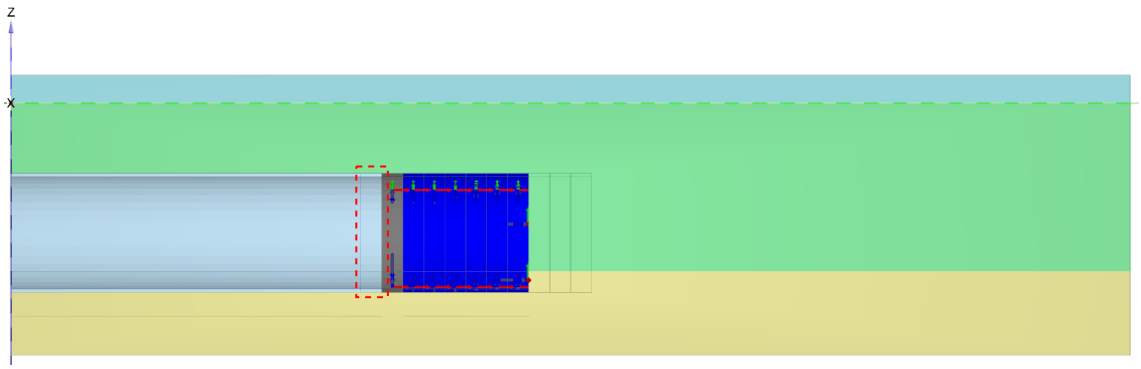


Figure 91: TBM advancement 35.5 to 37 m (front) - In the dotted line a new lining ring from 25 - 26.5 m (rear)

5.6.4 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**.

5.6.5 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**.

5.6.6 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 no load-displacement curves are drawn in this example, and start the calculation.

5.7 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 92 (on page 115)).
4. 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 1.95m.
   - In the **View** menu select **View Point** > **Right view**.
   - In the **Mesh** menu make the **Cluster borders** visible.
   - In the **View** menu select the **Distribution plane** option.

   As a result, the window with the cross section is displayed (Figure 93 (on page 116)). The maximum settlement at ground level is about 1.9 cm.

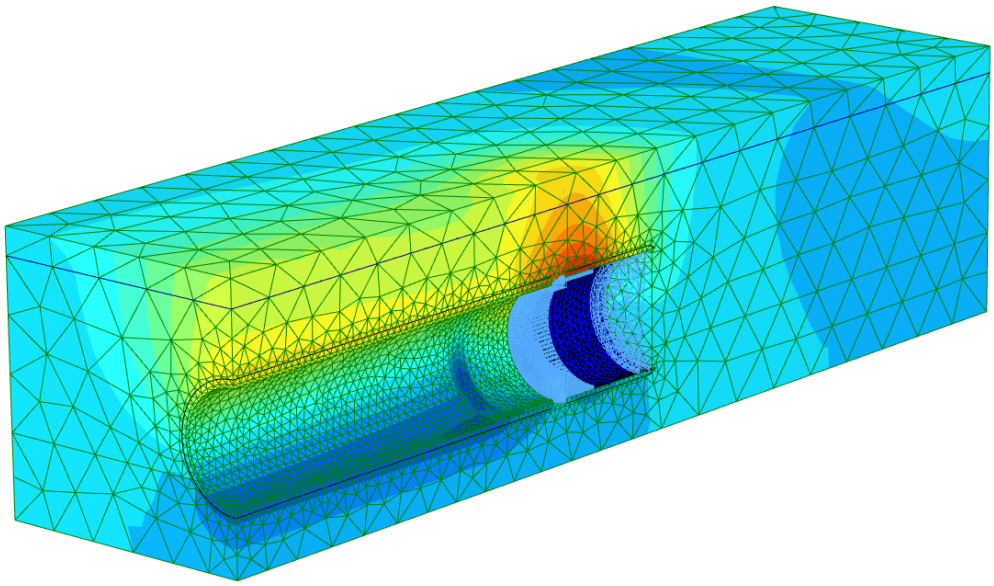


Figure 92: Total vertical displacements after the final phase $u_z \approx 3.1cm$

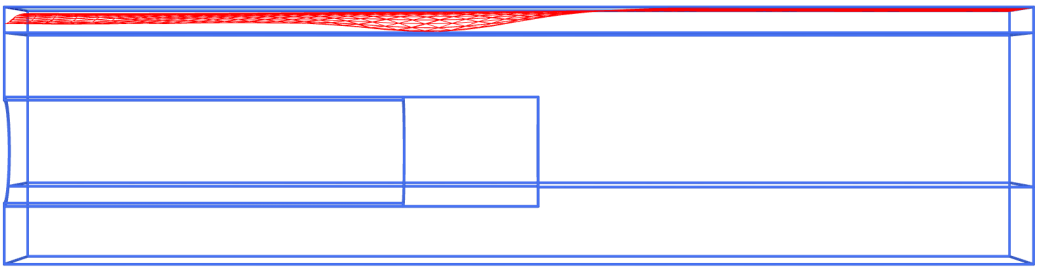


Figure 93: Settlement trough at ground level $u_z \approx 1.9cm$

6 Construction of a road embankment [ADV]

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

This tutorial concerns the construction of a road embankment in which the mechanism described above is analysed in detail. In the analysis 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

Figure 94 (on page 117) shows a cross section of a road embankment.

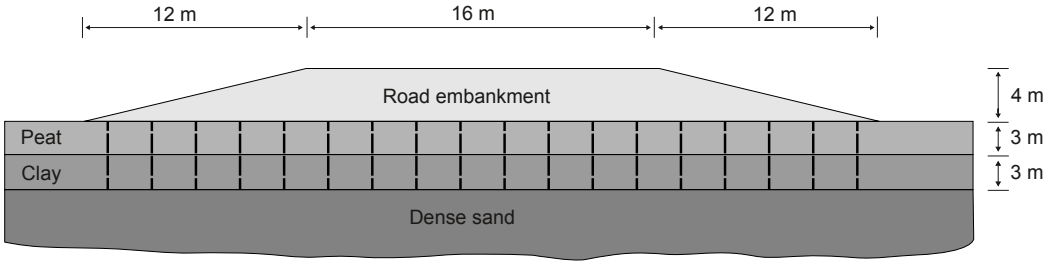


Figure 94: Situation of a road embankment on soft soil

6.1 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. $x_{min} = 0$ and $x_{max} = 60$.
   b. $y_{min} = 0$ and $y_{max} = 2$.

6.2 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 95 (on page 118).
3. The water level is located at z = -1 m. In the borehole column specify a value of -1 to **Head**.

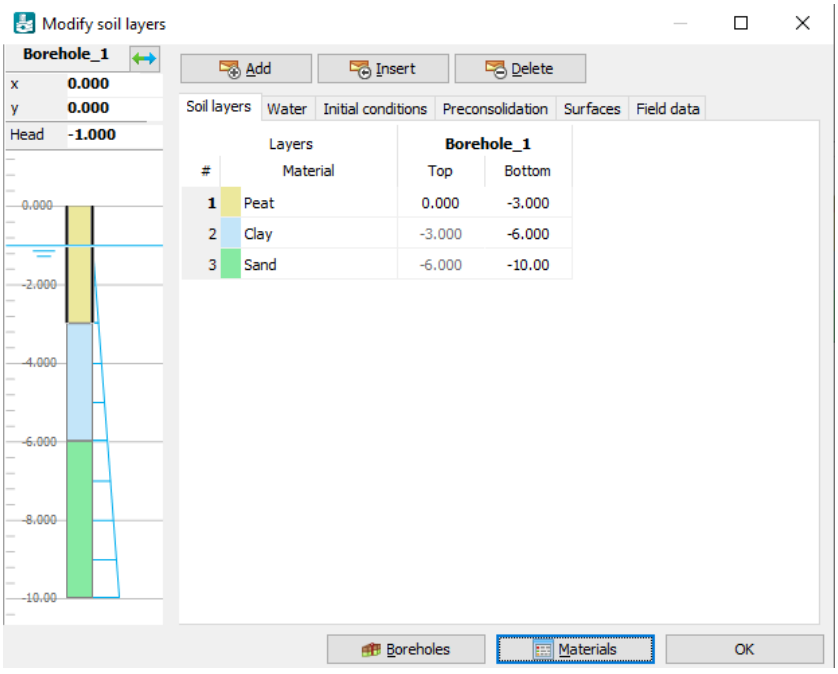


Figure 95: Soil layer distribution

6.3 Create and assign the material data sets

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

Table 16: Material properties of the road embankment and subsoil

Parameter	Name	Embankment	Sand	Peat	Clay	Unit
General						
Soil model	Model	Hardening Soil	Hardening Soil	Soft Soil model	Soft Soil model	-
Type of material behaviour	Type	Drained	Drained	Undrained (A)	Undrained (A)	-
Unsaturated unit weight	γ_{unsat}	16	17	8	15	kN/m^3
Saturated unit weight	γ_{sat}	19	20	12	18	kN/m^3
Initial void ratio	e_{init}	0.5	0.5	2.0	1.0	-
Mechanical						
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.5 \cdot 10^4$	$3.5 \cdot 10^4$	-	-	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$2.5 \cdot 10^4$	$3.5 \cdot 10^4$	-	-	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$7.5 \cdot 10^4$	$1.05 \cdot 10^5$	-	-	kN/m^2
Modified compression index	λ^*	-	-	0.15	0.05	-
Modified swelling index	κ^*	-	-	0.03	0.01	-
Power for stress-level dependency of stiffness	m	0.5	0.5	-	-	-
Cohesion (constant)	c'_{ref}	1.0	0.0	2.0	1.0	kN/m^2
Friction angle	φ'	30	33	23	25	°
Dilatancy angle	ψ	0.0	3.0	0	0	°

Groundwater						
Classification type	-	USDA	USDA	USDA	USDA	-
Model	-	Van Genuchten	Van Genuchten	Van Genuchten	Van Genuchten	-
Soil class	-	Loamy sand	Sand	Clay	Clay	-
<2µm	-	6.0	4.0	70.0	70.0	%
2µm - 50µm	-	11.0	4.0	13.0	13.0	%
50µm - 2mm	-	83.0	92.0	17.0	17.0	%
Use defaults	-	From data set	From data set	None	From data set	-
Horizontal permeability (x-direction)	k_x	3.499	7.128	0.1	0.04752	m/day
Horizontal permeability (y-direction)	k_y	3.499	7.128	0.1	0.04752	m/day
Vertical permeability	k_z	3.499	7.128	0.05	0.04752	m/day
Void ratio dependency	-	No	No	Yes	Yes	
Change in permeability	c_k	$1 \cdot 10^{15}$	$1 \cdot 10^{15}$	1.0	0.2	-

Interfaces						
Strength determination	-	Rigid	Rigid	Rigid	Rigid	-
Interface reduction factor	R_{inter}	1.0	1.0	1.0	1.0	-

Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-
Pre-overburden pressure	*POP*	0.0	0.0	5.0	0.0	kN/m^2
Over-consolidation ratio	*OCR*	1.0	1.0	1.0	1.0	-

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 16 (on page 119) and assign them to the corresponding layers in the borehole (Figure 95 (on page 118)).
3. Close the **Modify soil layers** window and proceed to the **Structures mode** to define the structural elements.

6.4 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 Figure 96 (on page 121) and click **Apply**.

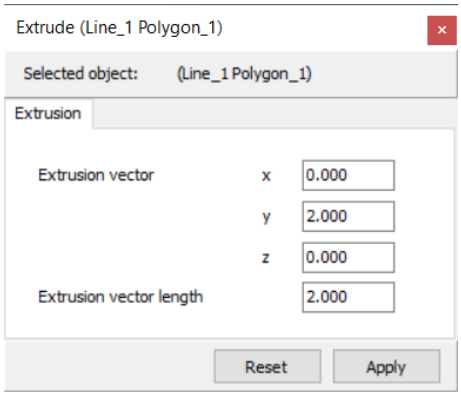


Figure 96: Extrusion window

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..

Drains are arranged in a square pattern, having a distance of 2m 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 Figure 97 (on page 122).

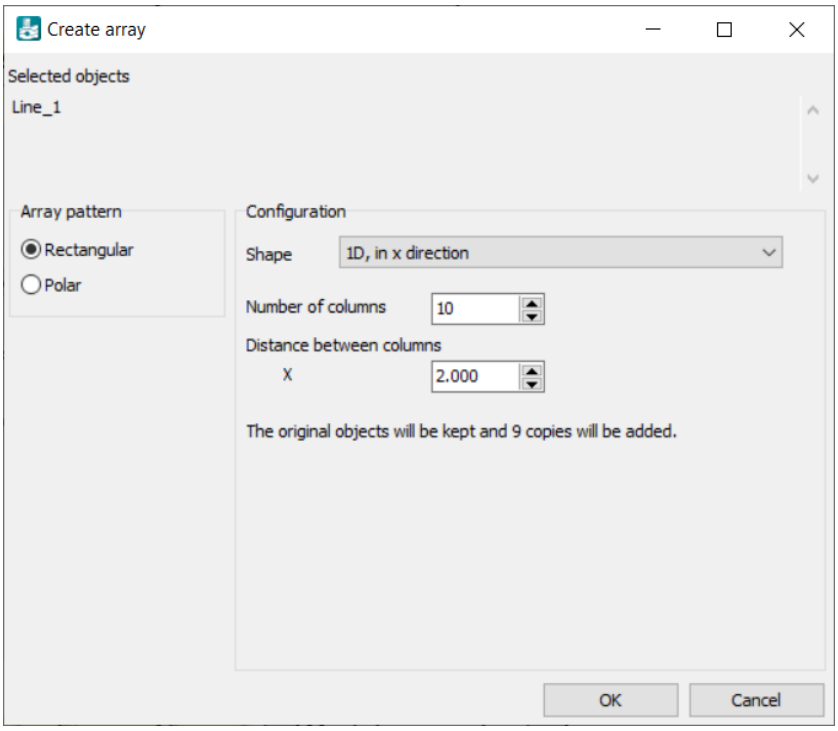


Figure 97: Settings of the drain pattern

The model geometry is shown in Figure 98 (on page 122):

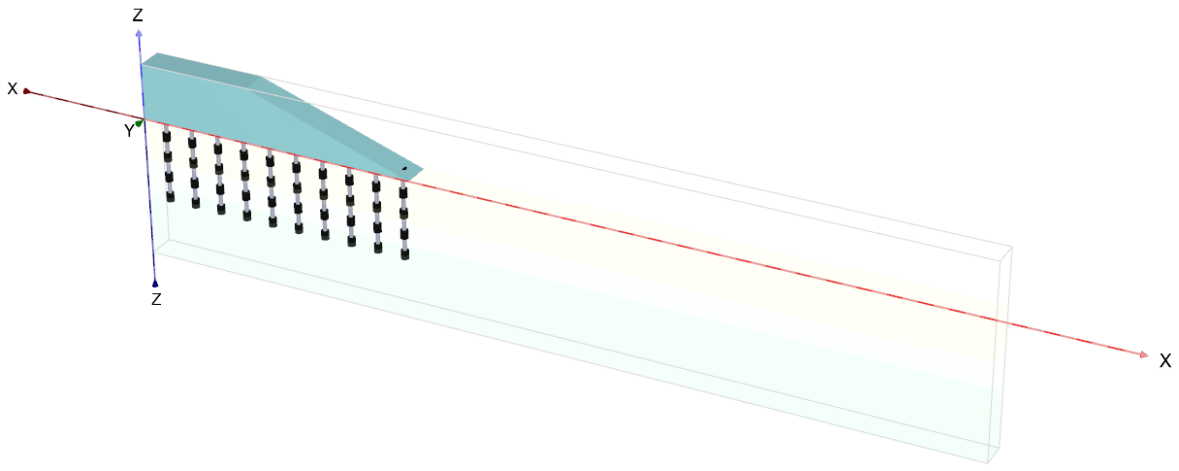


Figure 98: Model geometry

6.5 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 in Figure 99 (on page 123).

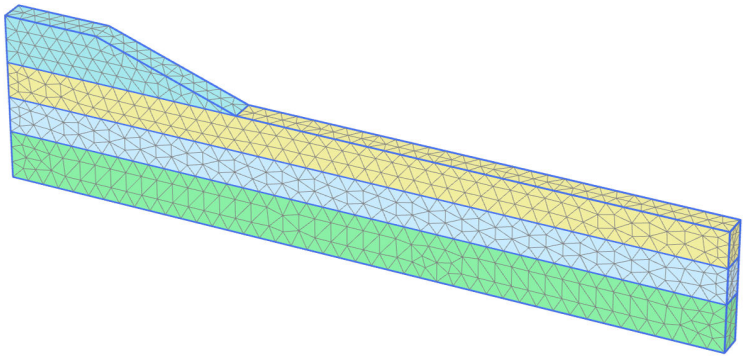


Figure 99: The generated mesh

6.6 Define the calculation

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

6.6.1 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** . 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 explorer** by selecting **Model conditions** > **Groundwater**. 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 100 (on page 124).

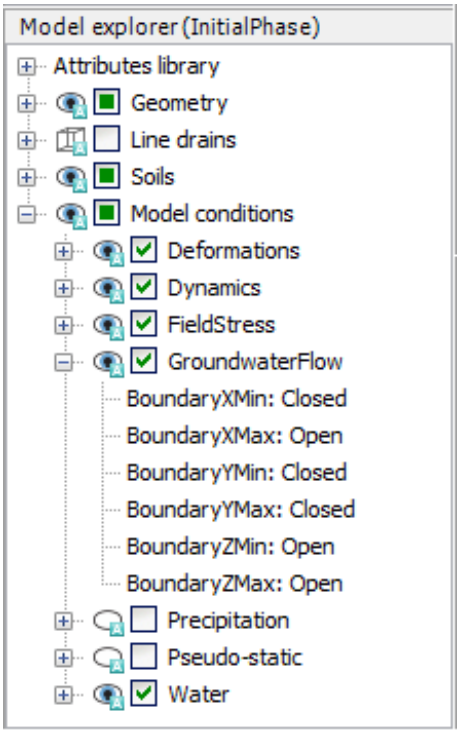


Figure 100: Boundary conditions for groundwater flow

6.6.2 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:

Phase 1 - First Construction

1. Click the **Add phase** button to introduce the first construction phase.
2. In the **General** subtree in the **Calculation type** select the **Consolidation** option from the 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 subtree **Model explorer** > **Model conditions** > **Water**. . .
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 - First consolidation

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 - First construction

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 - Long term consolidation

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. In the **Loading type** Select the **Minimum excess pore pressure** option from the drop-down menu. The default value for the minimum pressure (|P-stop| = 1.0 kN/m^2) as well as the default values for other parameters are used.

The definition of the calculation phases is complete.

6.6.3 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 (see Figure 101 (on page 127)). In addition to the multipliers, a parameter P_{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.

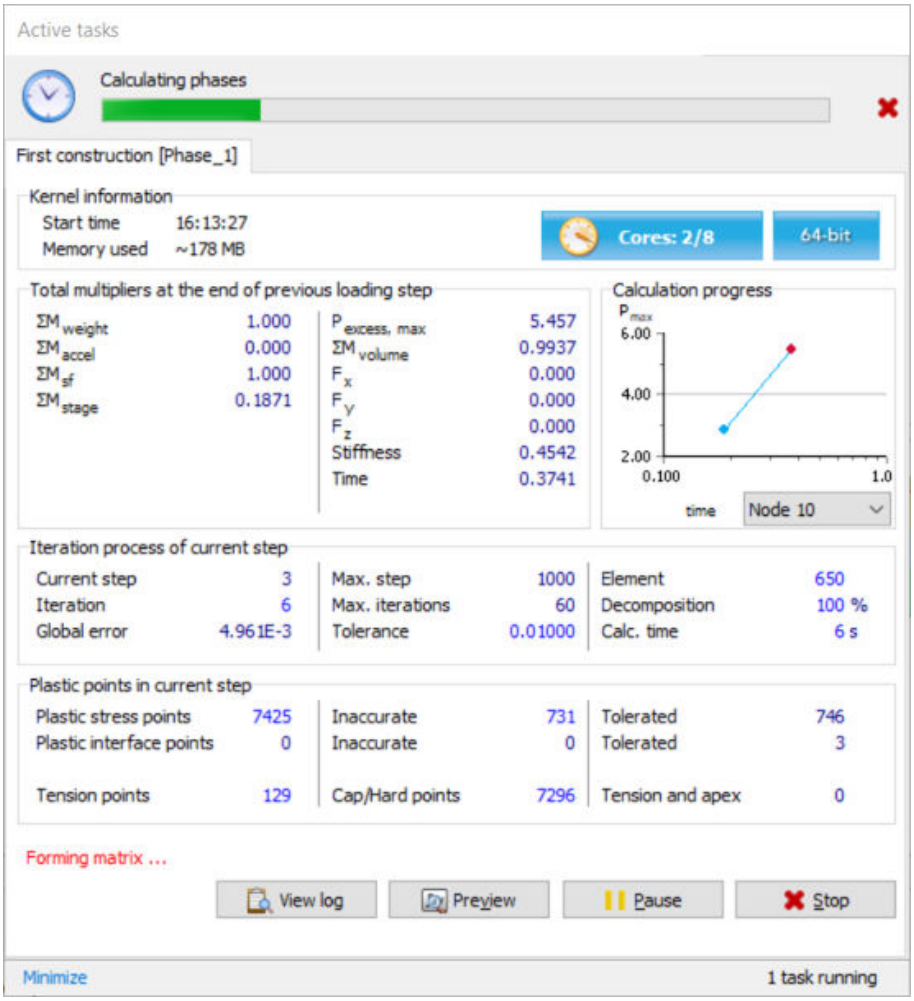


Figure 101: Calculation progress displayed in the Active tasks window

6.7 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 (see Figure 102 (on page 128)).

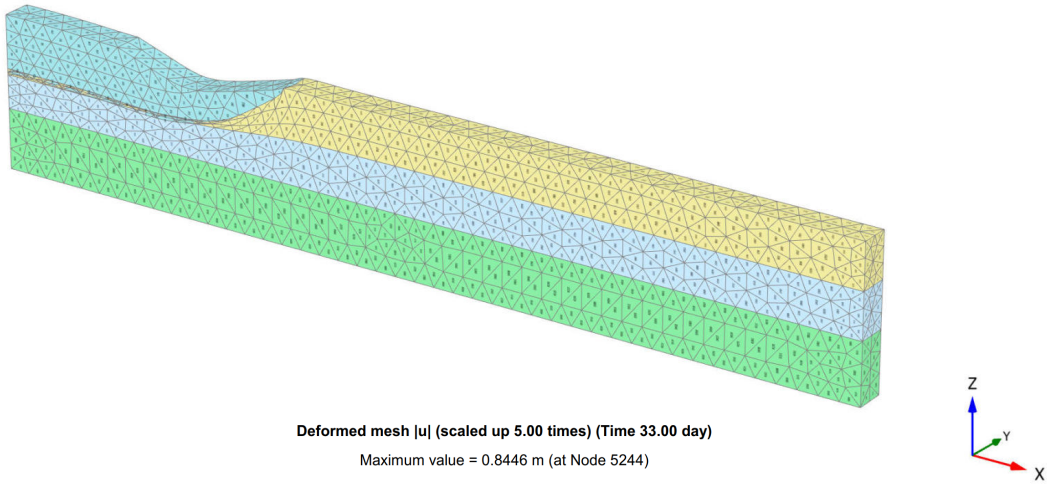


Figure 102: Deformed mesh after undrained construction of embankment (Phase 3)

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 in Figure 103 (on page 128) that a failure mechanism is developing:

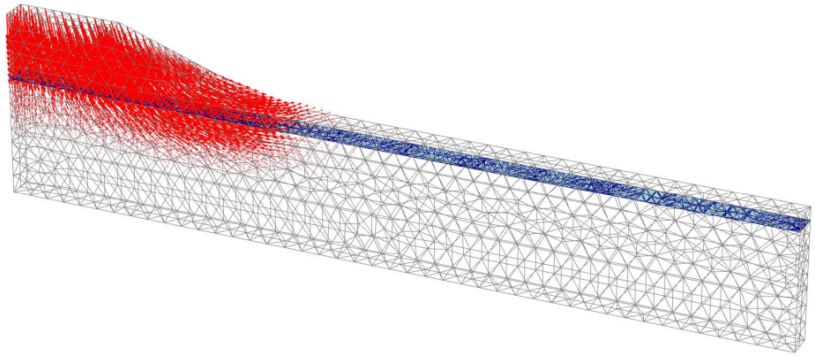


Figure 103: Displacement increments after undrained construction of embankment

1. Click **<Ctrl>** + **<7>** to display the developed excess pore pressures (see Appendix G 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. For a coloured visualization, in the general toolbar select **Coloured principal directions**. The results are displayed in Figure 104 (on page 129). It is clear that the highest excess pore pressure occurs under the embankment centre.

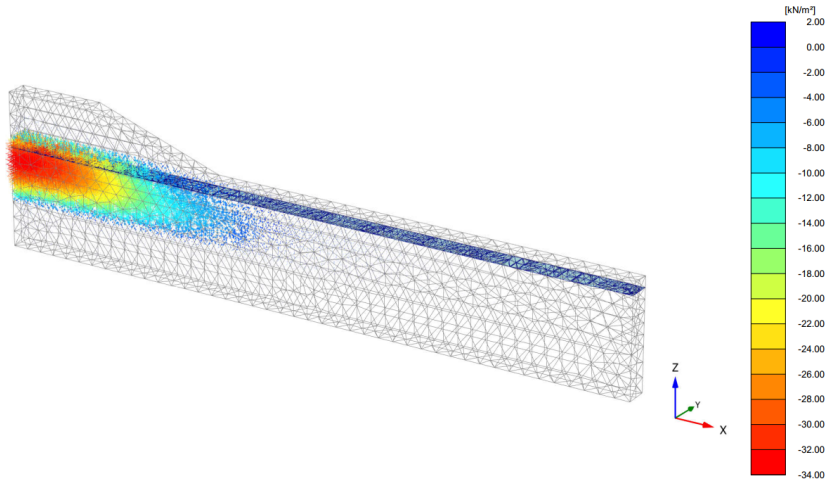


Figure 104: Excess pore pressures after undrained construction of embankment (Phase 3)

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 in Figure 105 (on page 129):

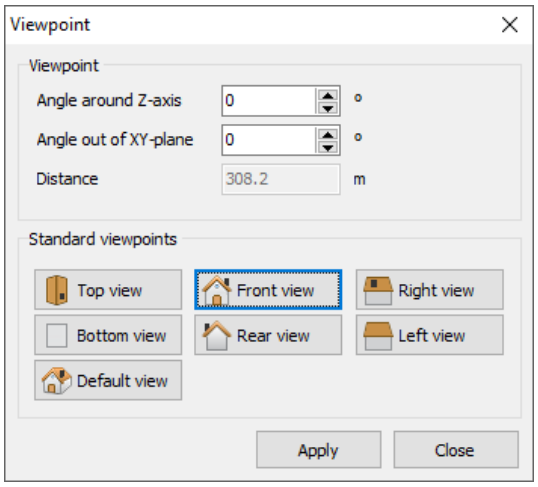


Figure 105: Viewpoint window

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/m^2.

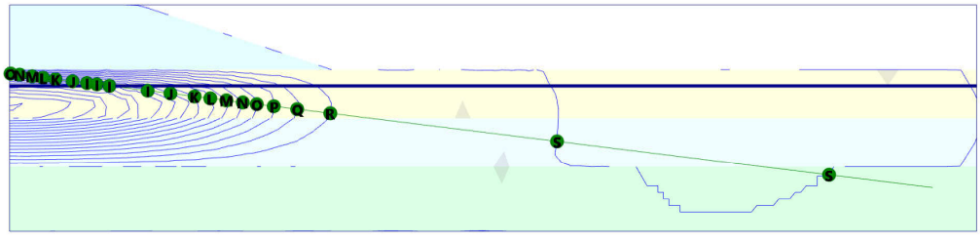


Figure 106: Excess pore pressure contours after consolidation to P_{excess} < 1.0 kN/m²

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 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. In the tabsheet corresponding to the studied node, 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**.

Figure 107 (on page 131) 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.

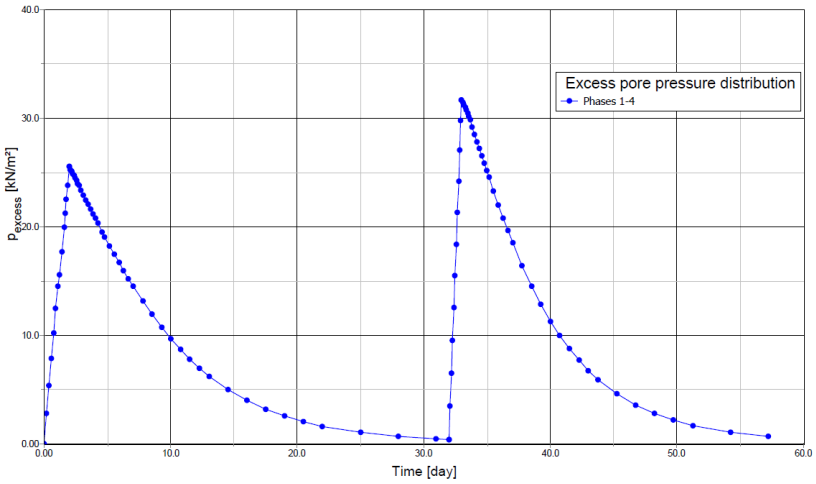


Figure 107: Development of excess pore pressure under the embankment

6.8 Safety analysis

6.8.1 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:

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

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

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

Where c and φ are the input strength parameters and σ_n is the actual normal stress component. The parameters c_r and φ_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:

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

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

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

6.8.2 Define the calculation

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

1. Add a new calculation phase.
2. In the **Phases** window in **Start from phase** select Phase 1 from the drop-down menu.
3. In the **General** subtree as calculation type select **Safety**.
4. The **Loading type** is automatically changed to **Incremental multipliers**.
5. The first increment of the multiplier that controls the strength reduction process, **Msf**, 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 in Figure 108 (on page 132).

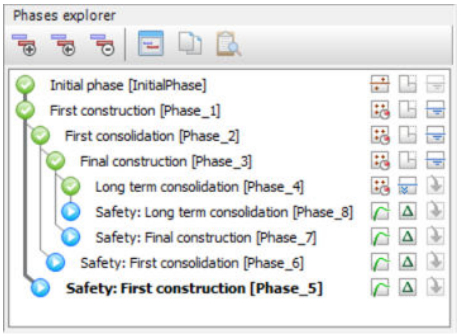


Figure 108: Phases explorer displaying the Safety calculation phases

9. Calculate the safety phases.

Note:

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

6.8.3 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 (see Figure 109 (on page 133)). The magnitude of the displacement increments is not relevant.

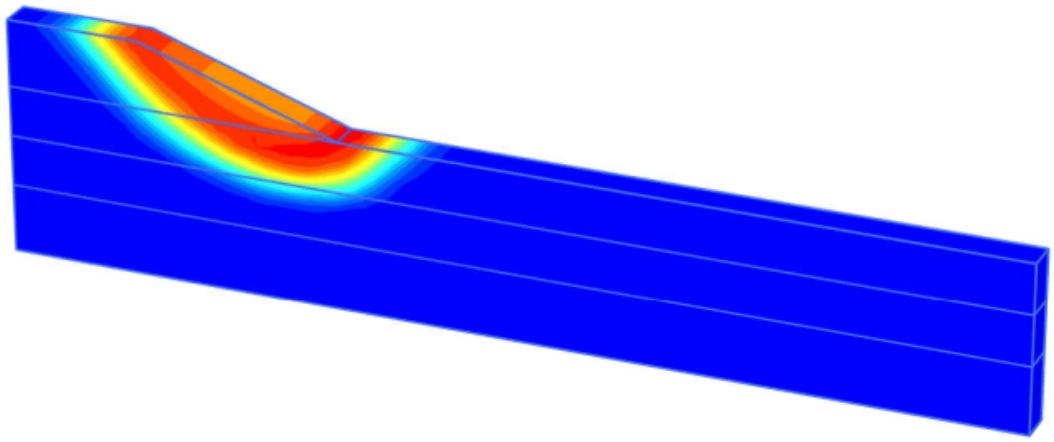


Figure 109: Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

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

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

In order to evaluate the safety factors for the four situations, 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** > **ΣMsf**. 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.

Figure 110 (on page 134) displays the safety factor for the calculated phases.

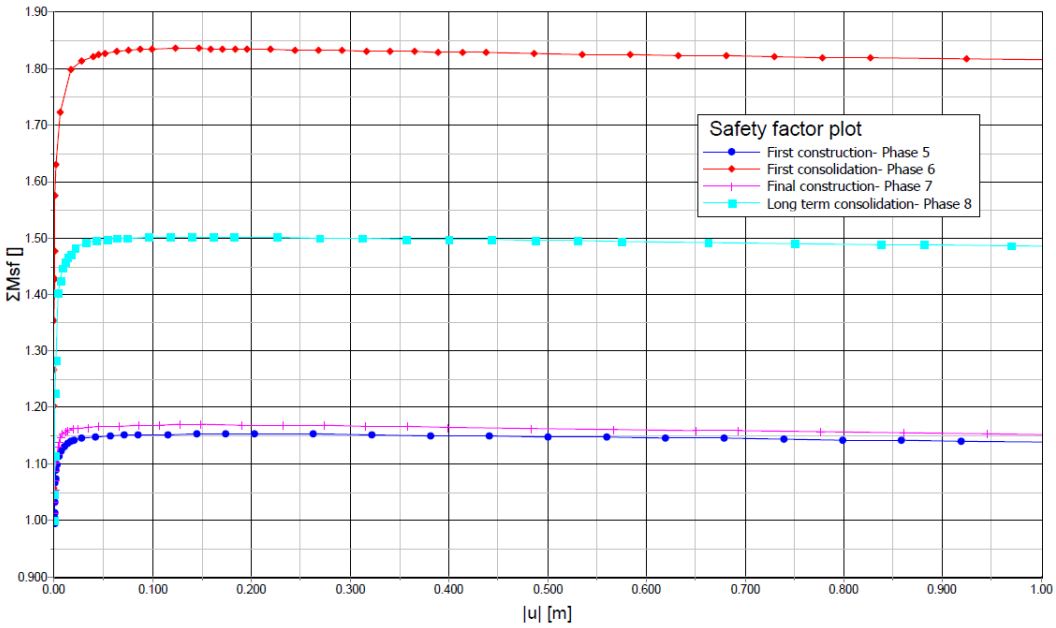


Figure 110: Evaluation of safety factor

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

6.9 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-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_{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. Figure 111 (on page 135) gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate.

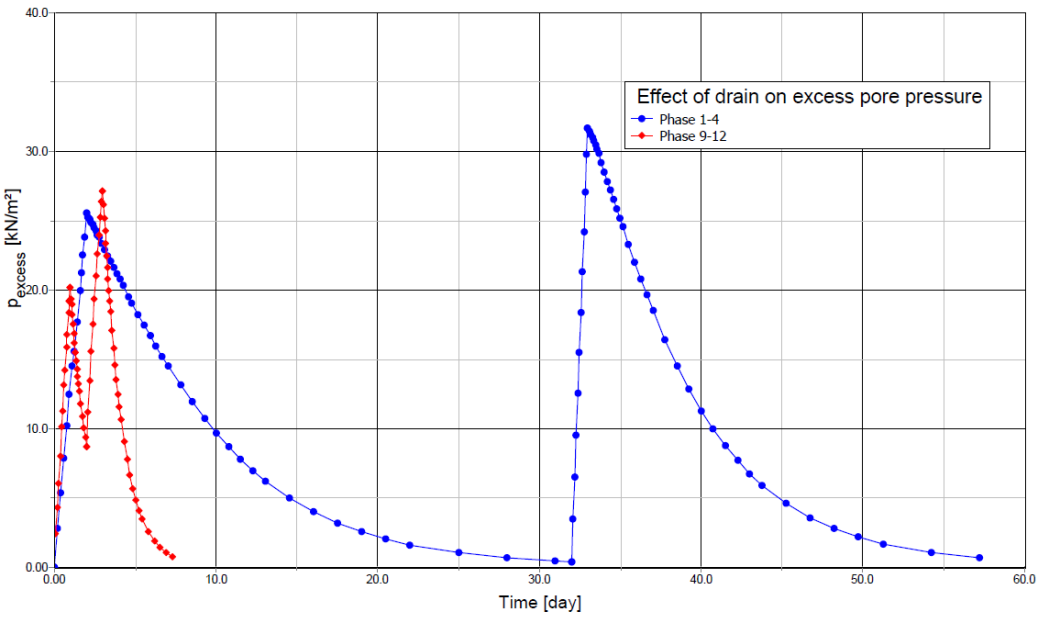


Figure 111: Effect of drains

7 Rapid drawdown analysis [ULT]

This example concerns the stability of a reservoir dam under conditions of drawdown. Fast reduction of the reservoir level may lead to instability of the dam due to high pore water pressures that remain inside the dam. 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 30m high. The top width and the base width of the dam are 5m and 172.5m respectively. The geometry of the dam is depicted below. The normal water level behind the dam is 25m high. A situation is considered where the water level drops 20m. The normal phreatic level at the right hand side of the dam is 10m below ground surface.

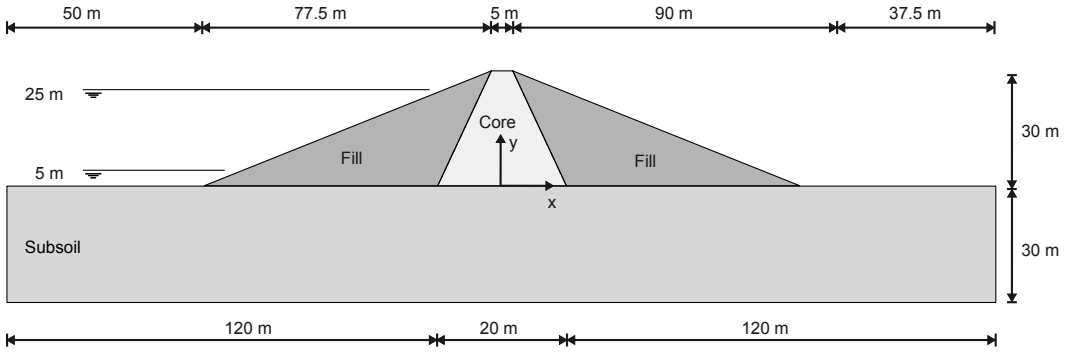


Figure 112: Geometry of the project

7.1 Create a new project

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_{min} = -130.0m, x_{max} = 130.0m
   b. y_{min} = 0m and y_{max} = 50.0m

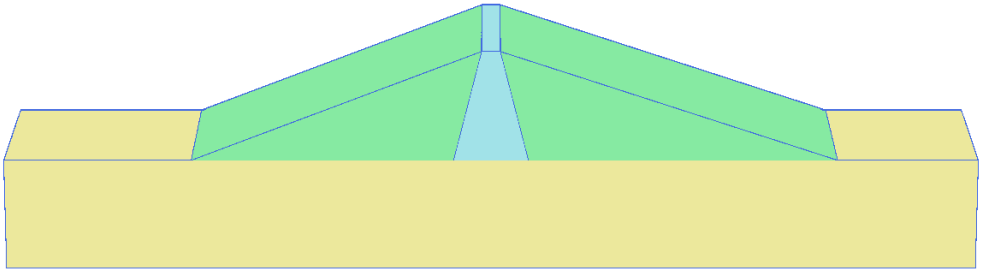


Figure 113: The geometry of the model

7.2 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 30m 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 30m (z= -30).
3. Set the **Head** in the borehole to -10m. 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.

7.3 Create and assign material data sets

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

The layers displayed on Table 17 (on page 137) have the properties:

Table 17: Material properties of the dam and subsoil

Property	Name	Core	Fill	Subsoil	Unit
General					
Soil model	Model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	-

Property	Name	Core	Fill	Subsoil	Unit
General					
Draining type	Type	Undrained (B)	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	16.0	16.0	17.0	kN/m^3
Saturated unit weight	γ_{sat}	18.0	20.0	21.0	kN/m^3
Mechanical					
Young's modulus	E'_{ref}	$1.5 \cdot 10^3$	$2.0 \cdot 10^4$	$5.0 \cdot 10^4$	kN/m^2
Poisson's ratio	$\nu(nu)$	0.35	0.33	0.3	-
Young's modulus inc.	E'_{inc}	300	-	-	kN/m^2/m
Reference level	z_{ref}	30	-	-	m
Undrained shear strength	$s_{u,ref}$	5.0	-	-	kN/m^2
Cohesion	c'_{ref}	-	5.0	1.0	kN/m^2
Friction angle	φ'	-	31	35.0	°
Dilatancy angle	ψ	-	1.0	5.0	°
Undrained shear strength inc.	$s_{u,inc}$	3.0	-	-	kN/m^3
Groundwater					
Classification type	Model	Hypres	Hypres	Hypres	-
SWCC fitting method	-	Van Genuchten	Van Genuchten	Van Genuchten	-
Subsoil/Topsoil	-	Subsoil	Subsoil	Subsoil	-
Soil class	-	Very fine	Coarse	Coarse	-
Permeability in horizontal direction	k_x	$1.0 \cdot 10^{-4}$	0.25	0.01	m/day
Permeability in horizontal direction	k_y	$1.0 \cdot 10^{-4}$	0.25	0.01	m/day
Permeability in vertical direction	k_z	$1.0 \cdot 10^{-4}$	0.25	0.01	m/day

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 137). 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.

7.4 Define the dam

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

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 139).

Table 18: Surface groundwater flow boundary conditions

Surface	Points
1	(-130 0 0), (-80 0 0), (-80 50 0), (-130 50 0)
2	(-80 0 0), (-2.5 0 30), (-2.5 50 30), (-80 50 0)
3	(-130 0 0), (-130 0 -30), (-130 50 -30), (-130 50 0)

7.5 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 (See Figure 114 (on page 140)):

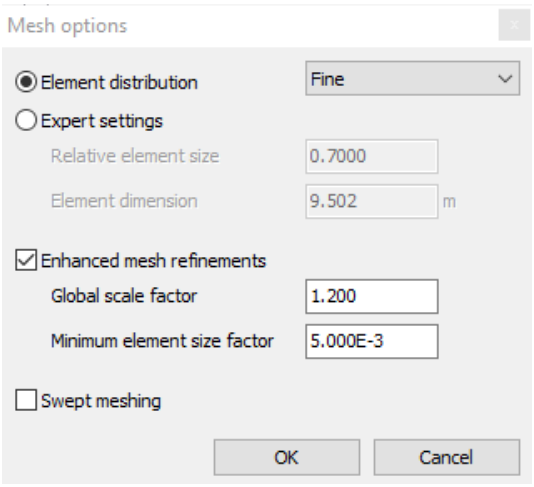


Figure 114: Modification of the Global coarseness

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 on Figure 115 (on page 140):

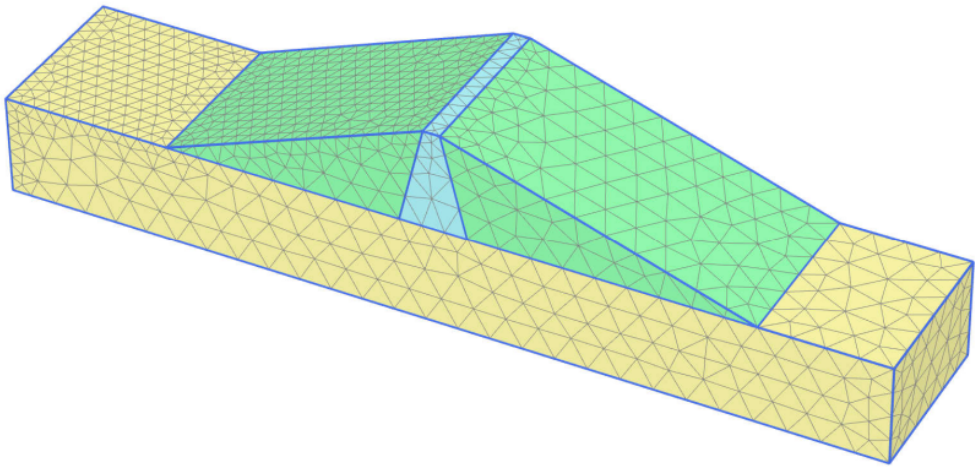


Figure 115: The generated mesh

7.6 Define and perform the calculation

In the calculation process the following cases will be considered:

- Initial state (high reservoir) - A long term situation with water level at 25m.
- The rapid drawdown case - A quick drop of the water level from 25 to 5m.
- The slow drawdown case - A slow drop of the water from 25 to 5m.

- The low water level case - A long term situation with water level at 5m.

In addition to **Initial phase**, the calculation consists of eight phases. In the initial phase, initial stresses and initial pore water pressures of the dam under normal working conditions are calculated using **Gravity loading**. For this situation the water pressure distribution is calculated using a steady-state groundwater flow calculation. The first and second phases both start from the initial phase (i.e. a dam with a reservoir level at 25m) and the water level is lowered to 5m. A distinction is made in the time interval at which this is done (i.e. different speeds of water level reduction; rapid drawdown and slow drawdown). In both cases the water pressure distribution is calculated using a fully coupled flow-deformation analysis. The third calculation phase also starts from the initial phase and considers the long-term behaviour of the dam at the low reservoir level of 5m, which involves a steady-state groundwater flow calculation to calculate the water pressure distribution. Finally, for all the water pressure situations the safety factor of the dam is calculated by means of phi-c reduction.

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 141).

Table 19: Water levels

Level	Points
High reservoir	(-130 0 25), (-10 0 25), (93 0 -10), (130 0 -10), (130 50 -10), (93 50 -10), (-10 50 25), (-130 50 25)
Low reservoir	(-130 0 5), (-10 0 5), (93 0 -10), (130 0 -10), (130 50 -10), (93 50 -10), (-10 50 5), (-130 50 5)

3. In the **Model explorer** under **Attributes library** 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**.

7.6.1 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. as **Calculation type** select the **Gravity loading** option .
   Note that **Staged construction** is the only option available for **Loading type**.
5. As **pore pressure calculation type** select the **Steady state groundwater flow** option.

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** (See Figure 116 (on page 142)).
11. In the **Water** subtree select the high reservoir water level (High_Reservoir) as **GlobalWaterLevel**.

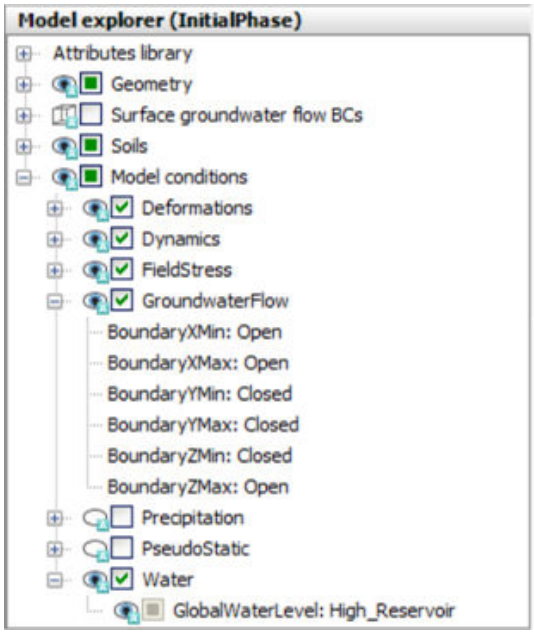


Figure 116: Boundary conditions for groundwater flow

7.6.2 Phase 1: Rapid drawdown

In the rapid drawdown phase the water level in the reservoir will be lowered from z = 25m to z = 5m 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. In the **Include reference flow analysis** option keep the checkbox unselected. This sets the non-steady state boundary conditions.
6. Set the **Time interval** to 5 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 (see Figure 117 (on page 143)).

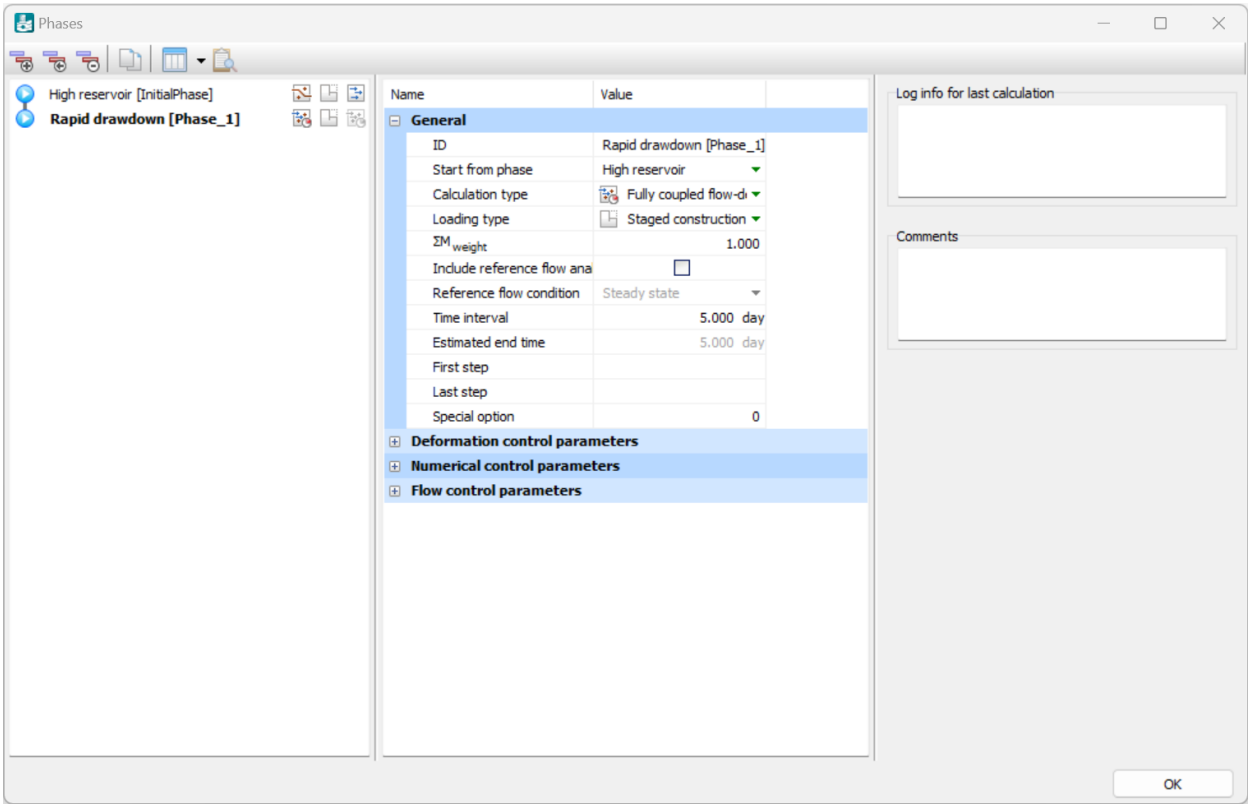


Figure 117: Phases window for the Rapid drawdown phase with - fully coupled flow-deformation calculation

9. Expand the **Attributes library** in the **Model explorer**.
10. Right-click on **Flow functions** and select the **Edit** option in the appearing menu.
The **Flow functions** window is displayed.
11.  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 -20m to **ΔHead**, representing the amount of the head decrease.
    d. Specify a time interval of 5 days. Figure 118 (on page 144) shows the defined function.

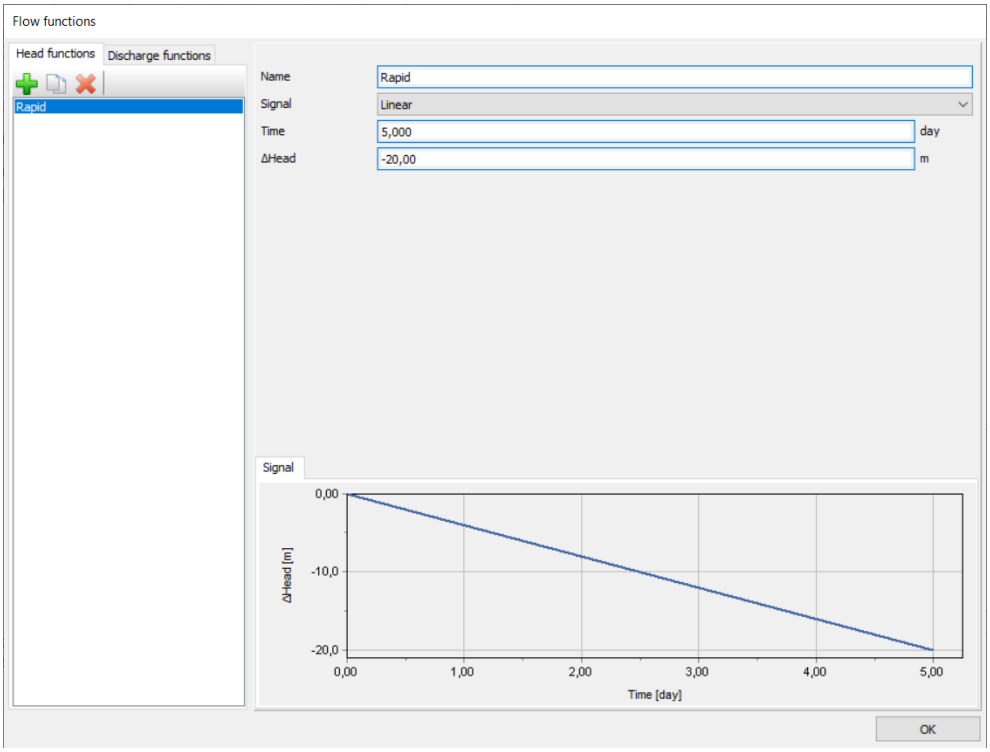


Figure 118: The flow function for the rapid drawdown case

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

12. Activate all the surface groundwater flow boundary conditions.
13. Multi-select the surface groundwater flow BCs in the drawing area.
14. In the **Selection explorer** as **behaviour** select the **Head** option. The distribution of the head is **Uniform**. Assign a value of 25m to h_{ref}.
15. Set the time dependency to **Time dependent** and as **Head function** select the **Rapid** option. Information related to the head function is displayed in the **Object explorers** as well.
16. In the **Model explorer** > **Model Conditions** > **Water** as **GlobalWaterLevel** select the BoreholeWaterLevel_1 option.

7.6.3 Phase 2: Slow drawdown

In the slow drawdown phase the water level in the reservoir will be lowered from z = 25m to z = 5m 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 (see Figure 119 (on page 145)) 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 -20m to **ΔHead**, representing the amount of the head decrease.
   d. Specify a time interval of 50 days.

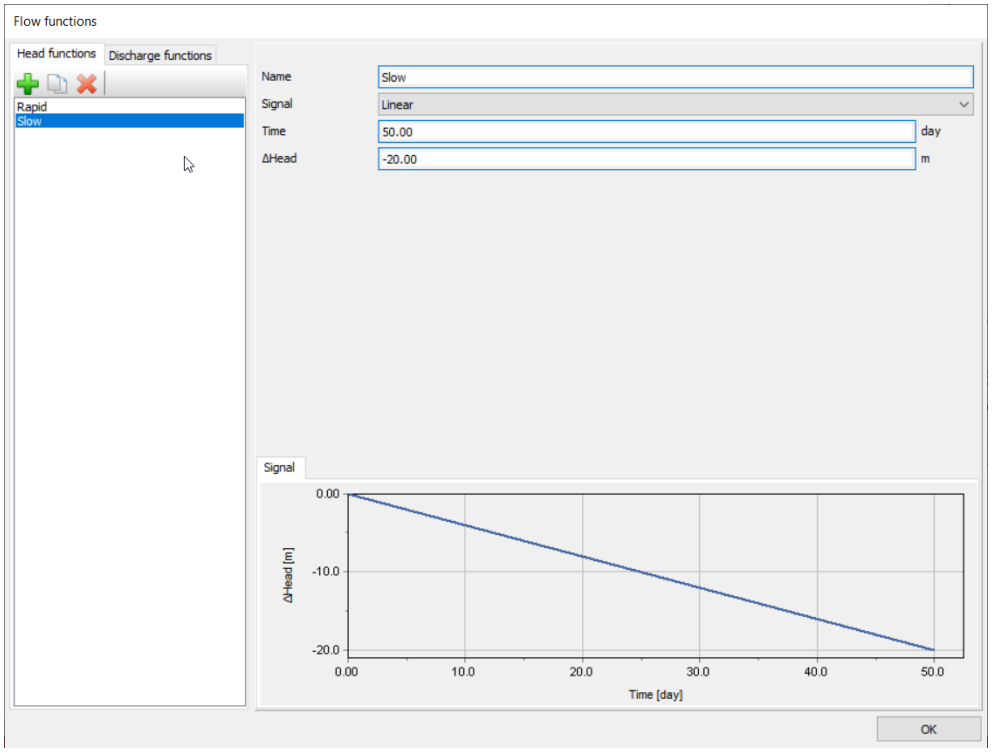


Figure 119: The flow function for the slow drawdown case

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 25m to h_{ref}.
12. Set the time dependency to **Time dependent** and as **Head function** select the **Slow** option.
13. In the **Water** subtree in the **Model explorer** as **GlobalWaterLevel** select the **BoreholeWaterLevel_1** option.

7.6.4 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**.

7.6.5 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**. As **Loading type** select **Incremental multipliers** option.
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 as shown in Figure 120 (on page 146).

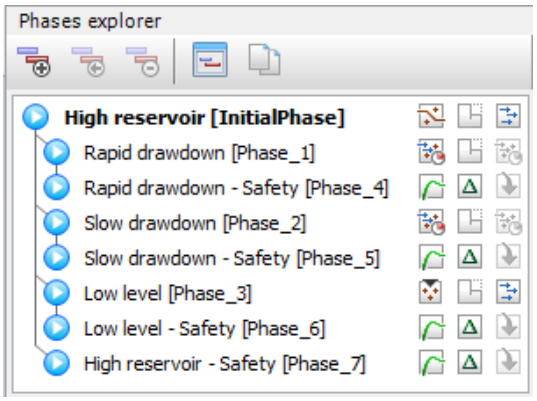


Figure 120: The final view of the Phases explorer

7.6.6 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.

7.7 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_{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 displayed for four conditions as follows:

- The situation with a high (standard) reservoir level - See Figure 121 (on page 148).
- The situation after rapid drawdown of the reservoir level - See Figure 122 (on page 148).
- The situation after slow drawdown of the reservoir level - See Figure 123 (on page 149).
- The situation with a low reservoir level - See Figure 124 (on page 149).

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

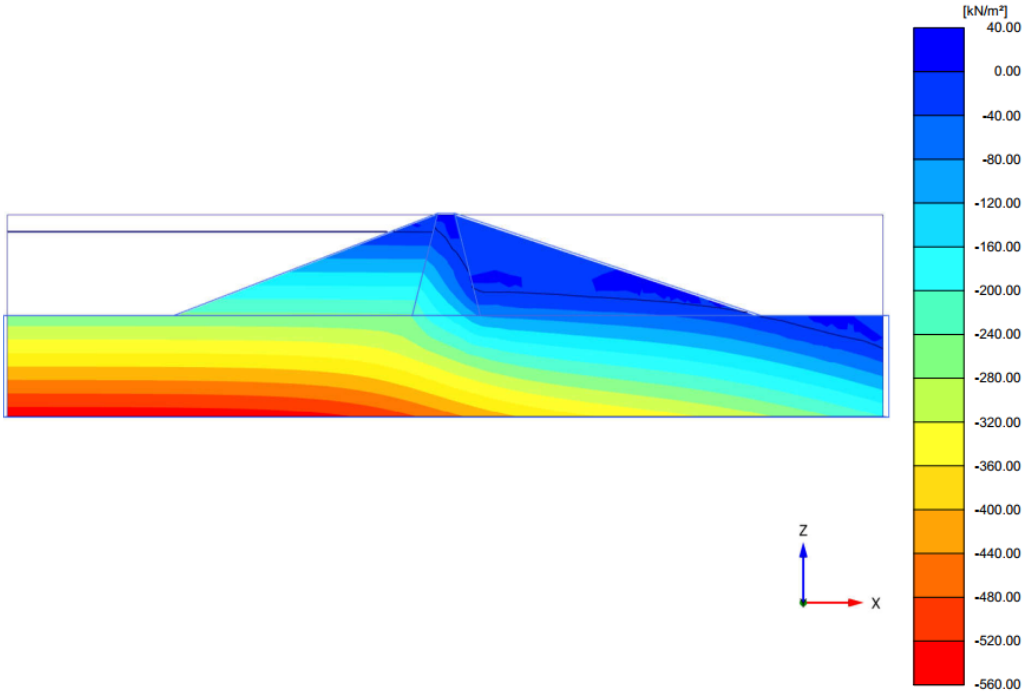


Figure 121: Pore water pressure distribution for high reservoir level (Initial phase)

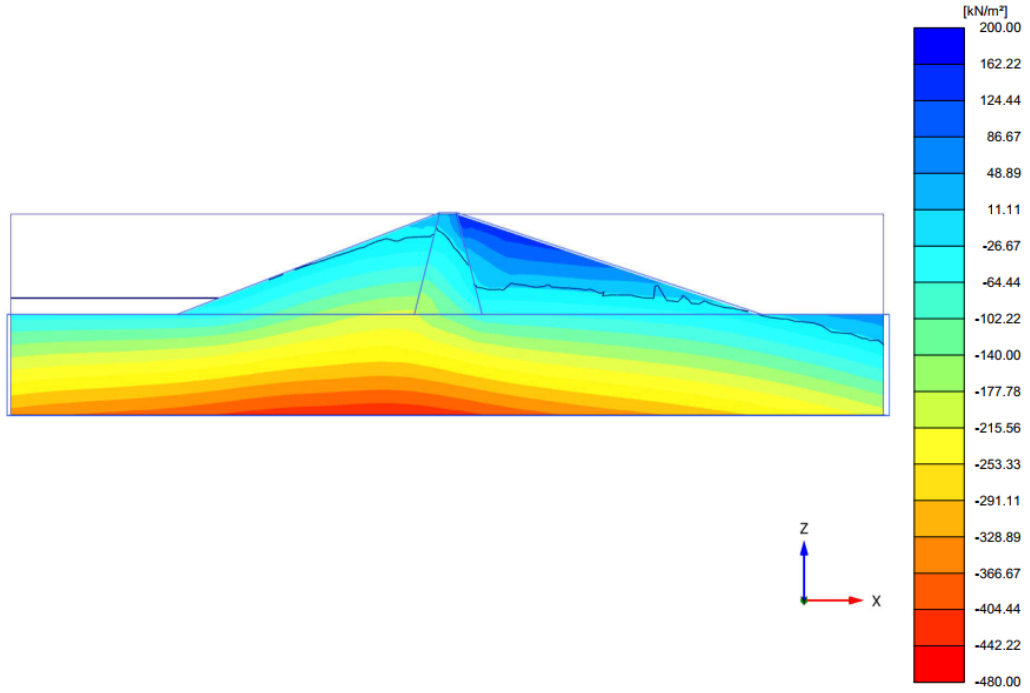


Figure 122: Pore water pressure distribution after rapid drawdown (Phase_1)

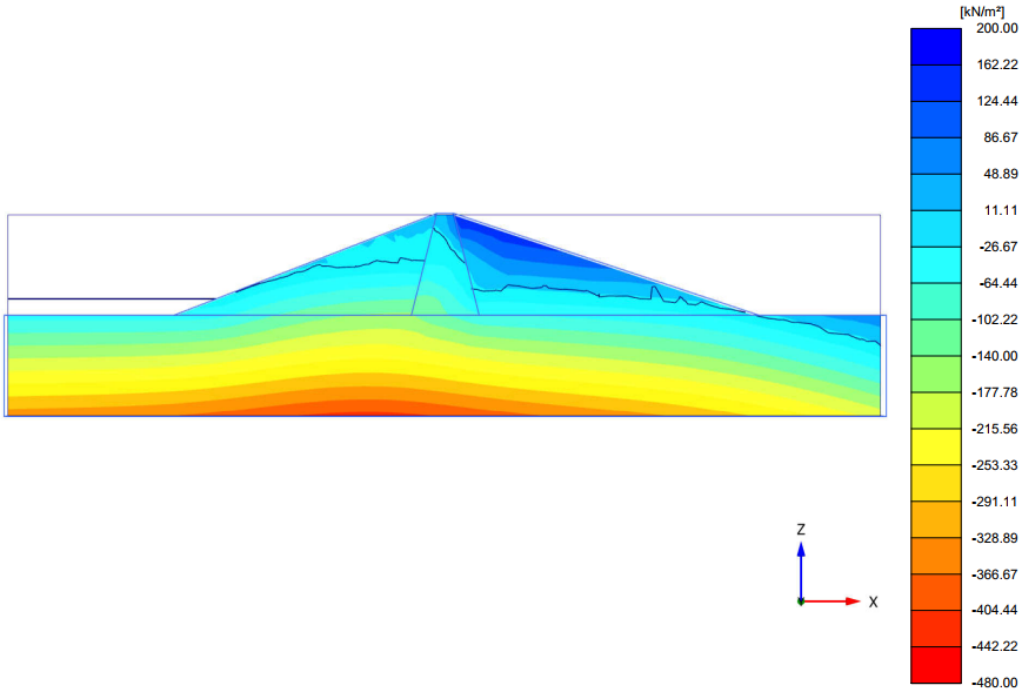


Figure 123: Pore water pressure distribution after slower drawdown (Phase_2)

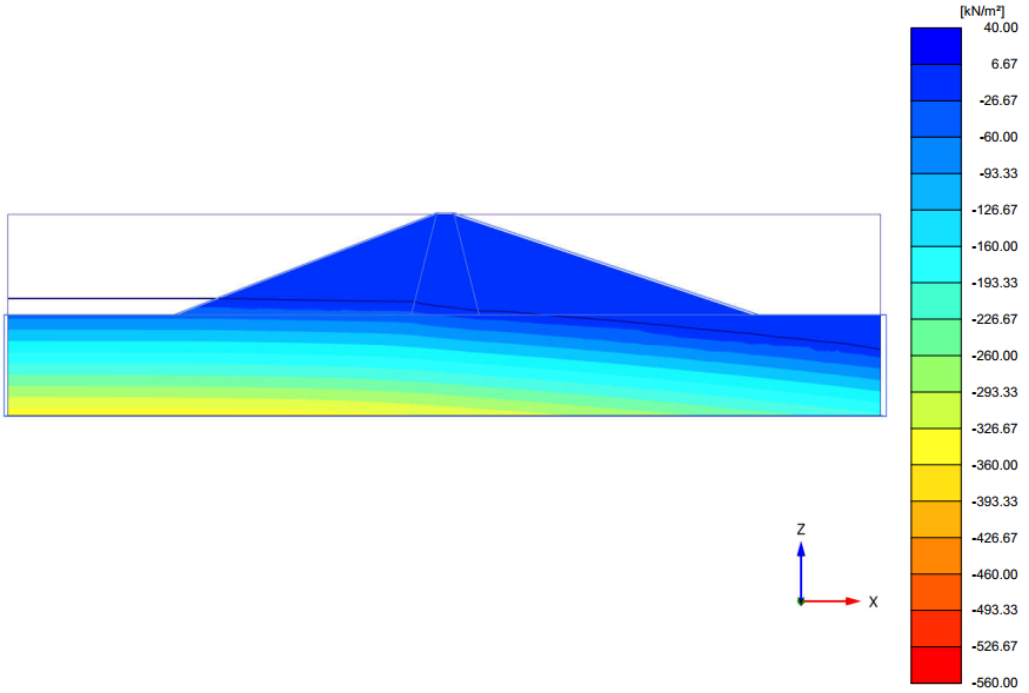


Figure 124: Pore water pressure distribution for low reservoir level (Phase_3)

In this tutorial attention is focused on the variation of the safety factor of the dam for the different situations. Therefore, the development of **ΣMsf** is plotted for the phases 4 to 7 as a function of the displacement of the dam crest point (see Figure 125 (on page 150)).

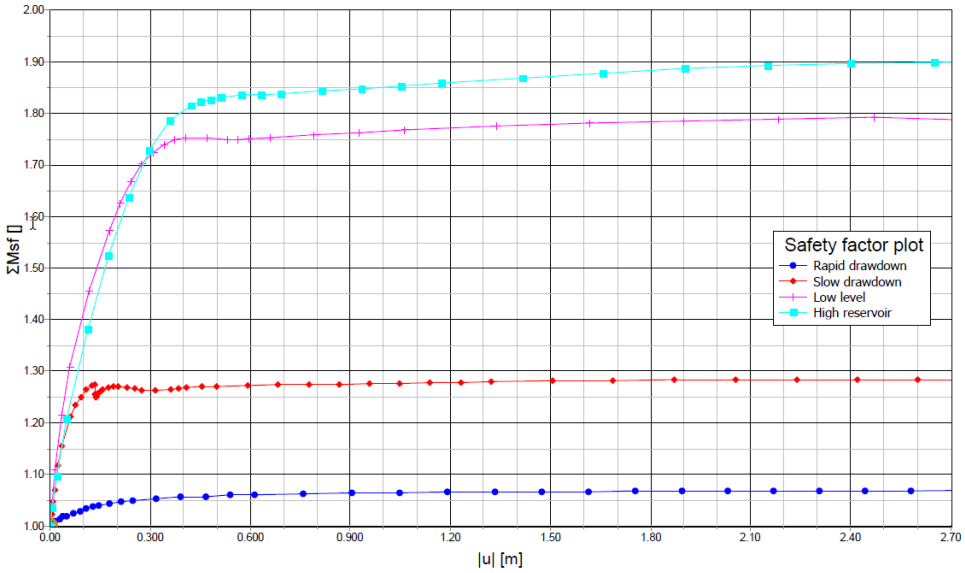


Figure 125: Safety factors for different situations

Rapid drawdown of a reservoir level can reduce the stability of a dam significantly. Fully coupled flow-deformation and stability analysis can be performed with PLAXIS 3D to effectively analyse such situations.

8

Dynamics analysis of a generator on an elastic foundation [ULT]

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 dynamics 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 0.2m thick concrete footing of 1m in diameter. Oscillations caused by the generator are transmitted through the footing into the subsoil (See Figure 126 (on page 152)). These oscillations are simulated as a uniform harmonic loading, with a frequency of 10Hz and amplitude of $10 kN/m^2$. In addition to the weight of the footing, the weight of the generator is modelled as a uniformly distributed load of $8 kN/m^2$.

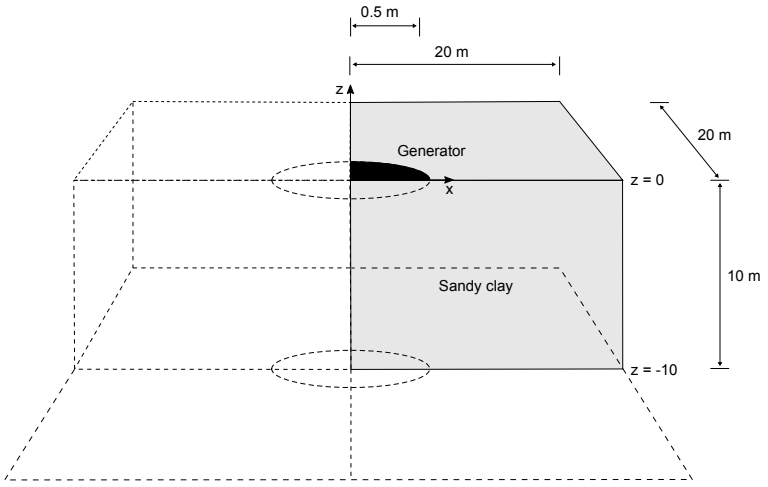


Figure 126: Generator founded on elastic subsoil

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 dynamics analysis, model boundaries are generally taken further away than in a static analysis.

8.1 Create a new project

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. $x_{min} = 0$ and $x_{max} = 20$.
   b. $y_{min} = 0$ and $y_{max} = 20$.

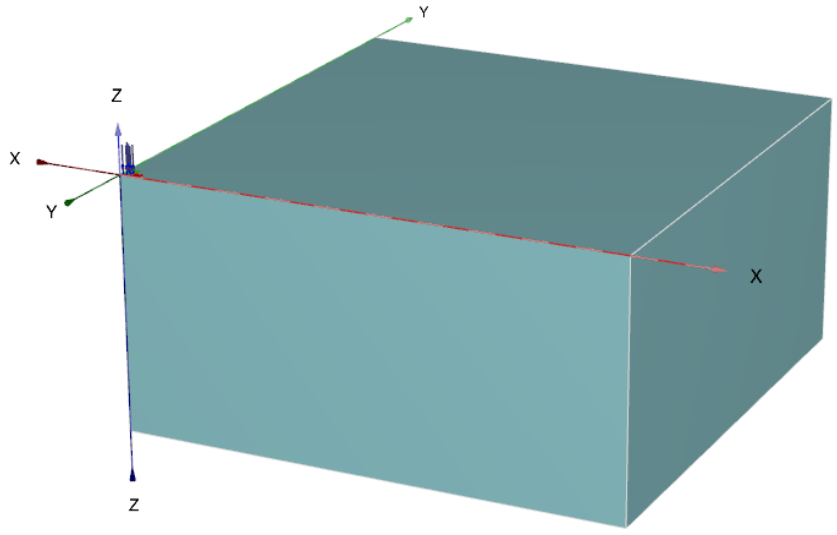


Figure 127: The geometry of the model

8.2 Define the soil stratigraphy

1. The subsoil consists of one layer with a depth of 10m. 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.

8.3 Create and assign material data sets

Create the material data set according to Table 20 (on page 153) and assign it to the soil layer.

Table 20: Material properties

Property	Name	Sandy clay	Unit
General			
Soil model	Model	Linear elastic	-
Drainage type	Type	Drained	-
Unsaturated unit weight	γ_{unsat}	20.0	kN/m^3
Saturated unit weight	γ_{sat}	20.0	kN/m^3

Mechanical			
Young's modulus	E'_{ref}	$5 \cdot 10^4$	kN/m^2
Poisson's ratio	*v(nu)*	0.3	-
Interfaces			
Strength determination	-	Rigid	-
Initial			
K_0 determination	-	Automatic	-
Lateral earth pressure coefficient	$K_{0,x}$, $K_{0,y}$	0.5	-

8.4 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 as insertion point.
2. In the **General** tabsheet 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 Table 21 (on page 154).

Table 21: Segment properties

Segment	Segment 1	Segment 2	Segment 3
Segment type	Line	Arc	Line
Segment properties	Relative start angle = 0° Length = 0.5m	Relative start angle = 90° Radius = 0.5m Segment angle = 90°	Relative start angle = 90° Length = 0.5m

4. Once the segments are created close the polycurve designer.
5. Right-click the polycurve and select the **Create** > **Create surface** option from the appearing menu.
6. Right-click the created surface and select the **Create** > **Create surface load** option in the appearing menu.
7. In the **Selection explorer**, for the **surface load** the **Uniform** distribution is valid. Assign (0 0 -8) to the pressure components.

8.4.1 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 in Figure 128 (on page 155).

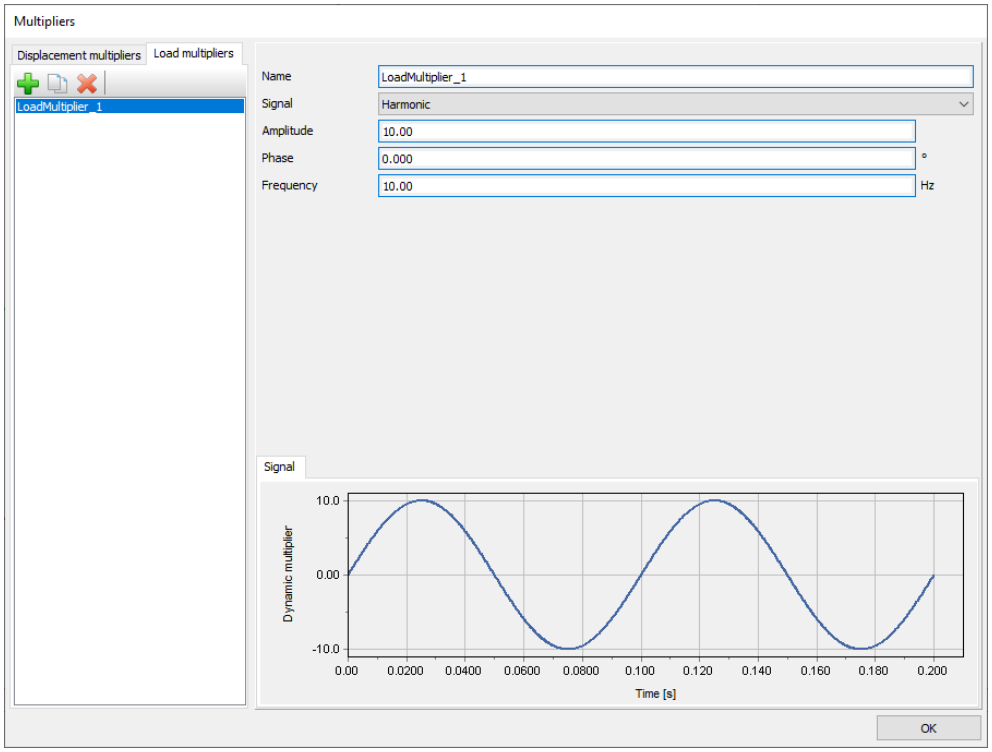


Figure 128: Definition of a Harmonic multiplier

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 **Structures mode** as well as in the **Stage construction mode**.

8.5 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. For **Element distribution** the **Medium** option will be used.
4. View the generated mesh (See Figure 129 (on page 156)).

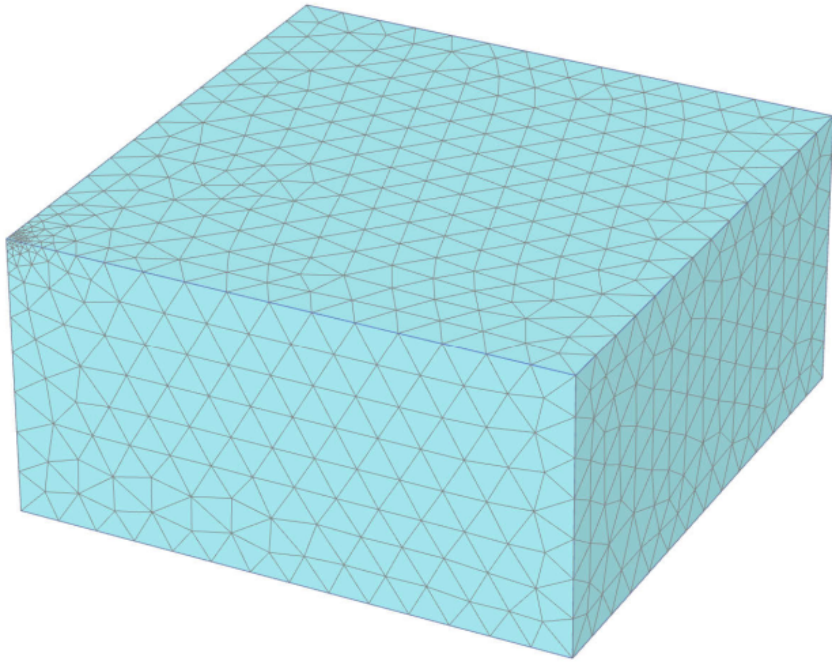


Figure 129: The generated mesh

Note: In all dynamics 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.

8.6 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 fisrt phase is a **Plastic** calculation where the static load is activated. The second phase is a

Dynamic calculation where the effect of the functioning generator is considered. The third and final phase are **Dynamic** calculations as well where the generator is turned off and the soil will vibrate freely.

8.6.1 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.

8.6.2 Phase 1: Footing

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**, as displayed on Figure 130 (on page 157), activate the static component of the surface load. Do not activate the dynamic load.

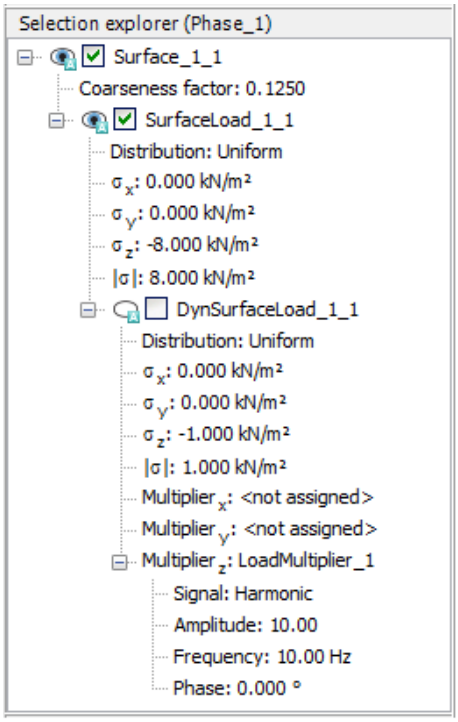


Figure 130: Applied load in the Phase_1

8.6.3 Phase 2: Start generator

In this phase, a vertical harmonic load, with a frequency of 10 Hz and amplitude of 10 kN/m2, is applied to simulate the vibrations transmitted by the generator. Five cycles with a total time interval of 0.5 sec are considered.

1. Add a new calculation phase (Phase_2).
2. In the **General** subtree in the **Phases** window, as calculation type select the **Dynamic** option.
3. Set the **Time interval** parameter to 0.5s.
4. In the **Deformation control parameters** subtree 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 (see Figure 131 (on page 158)).

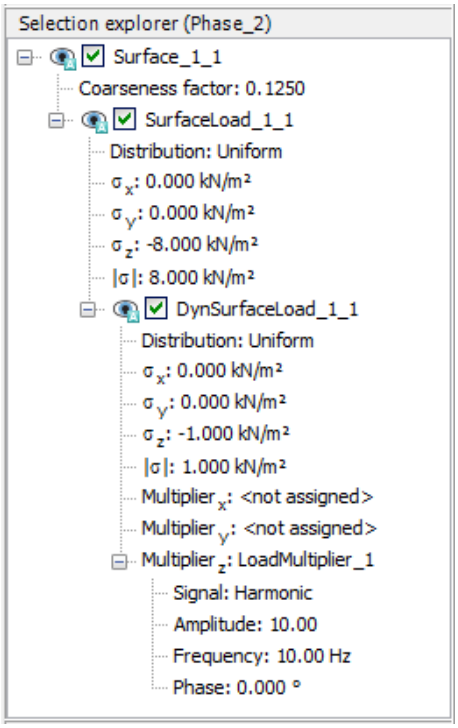


Figure 131: Applied load in the Phase_2

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 **Model Explorer** > **Model conditions** > **Dynamics** as displayed in Figure 132 (on page 159).

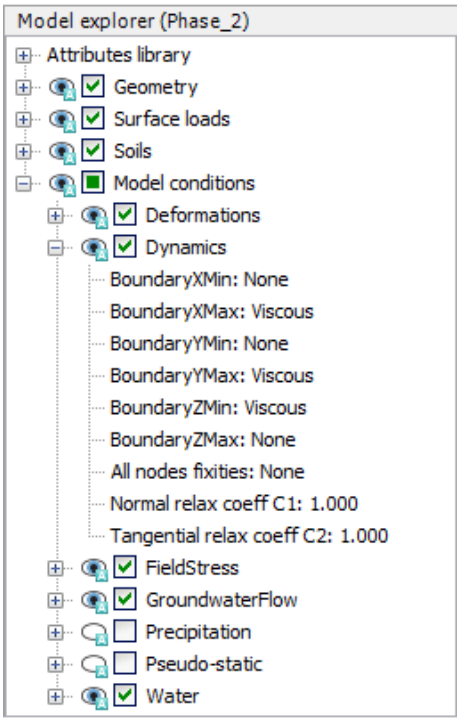


Figure 132: Boundary conditions for dynamics calculations

8.6.4 Phase 3: Stop generator

1. Add a new calculation phase (Phase_3).
2. In the **General** subtree in the **Phases** window, as **Calculation type** select the **Dynamic** option.
3. Set the **Dynamic time interval** parameter to 0.5s.
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.

Figure 133 (on page 160) shows the **Phases** explorer of this tutorial.

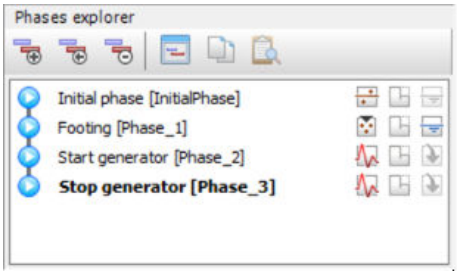


Figure 133: Phases explorer

8.6.5 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.

8.6.6 Additional calculation with damping

In a second calculation, material damping is introduced by means of Rayleigh damping. Rayleigh damping can be entered in the material data set. The following steps are necessary:

1. Save the project under another name.
2. Open the material data set of the soil.
3. In the **General** tabsheet under **Rayleigh damping** select as Input method the **SDOF equivalent** option (See Figure 134 (on page 161)).
4. Set a value of 5% for both ξ_1 and ξ_2. .
5. Set respectively values of 9 and 11 for the frequency targets, f_1 and f_2.
6. Notice that the values of α and β are automatically calculated by the program.

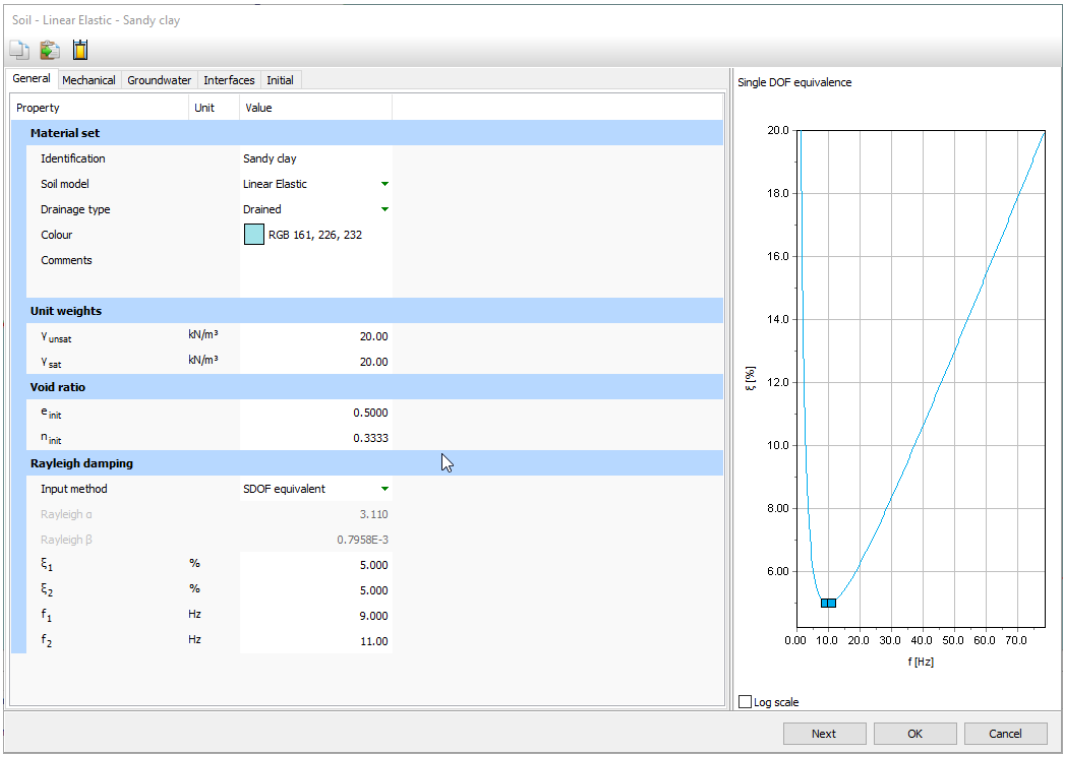


Figure 134: Input of Rayleigh damping

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.

8.6.7 Results

The **Curve manager** feature is particularly useful for dynamics analysis. You can easily display the actual loading versus time (input) and also displacements, velocities and accelerations of the pre-selected points versus time. The evolution of the defined multipliers with time can be plotted by assigning **Dynamic time** to x-axis and u_z to the y-axis. Figure 135 (on page 162) 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.

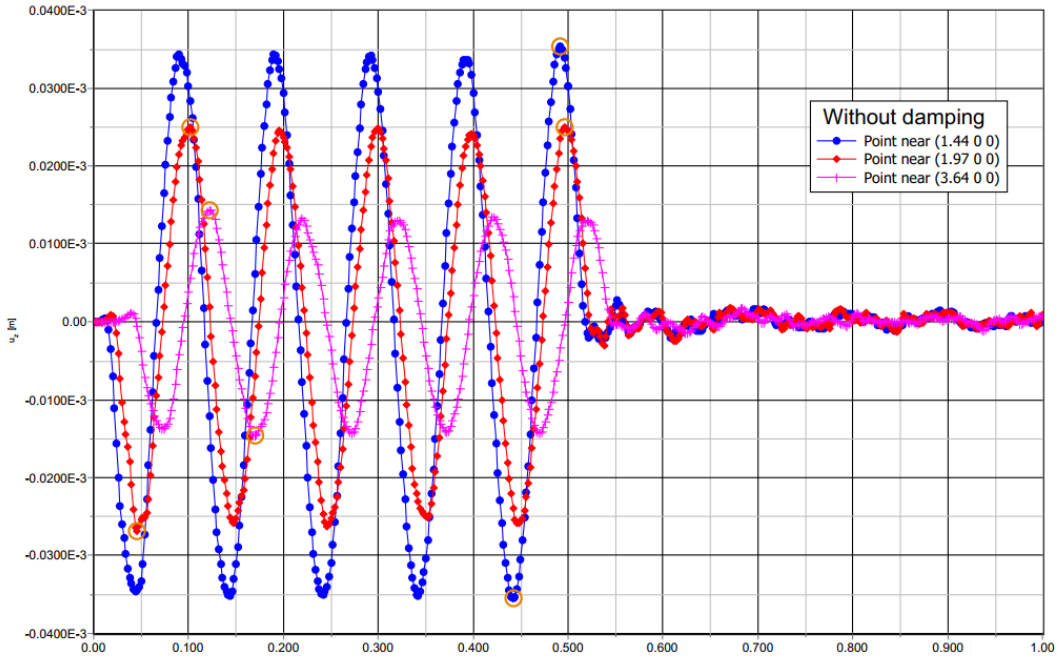


Figure 135: Vertical displ.- time on the surface at different distances to the vibrating source (without damping)

Figure 136 (on page 162) 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.

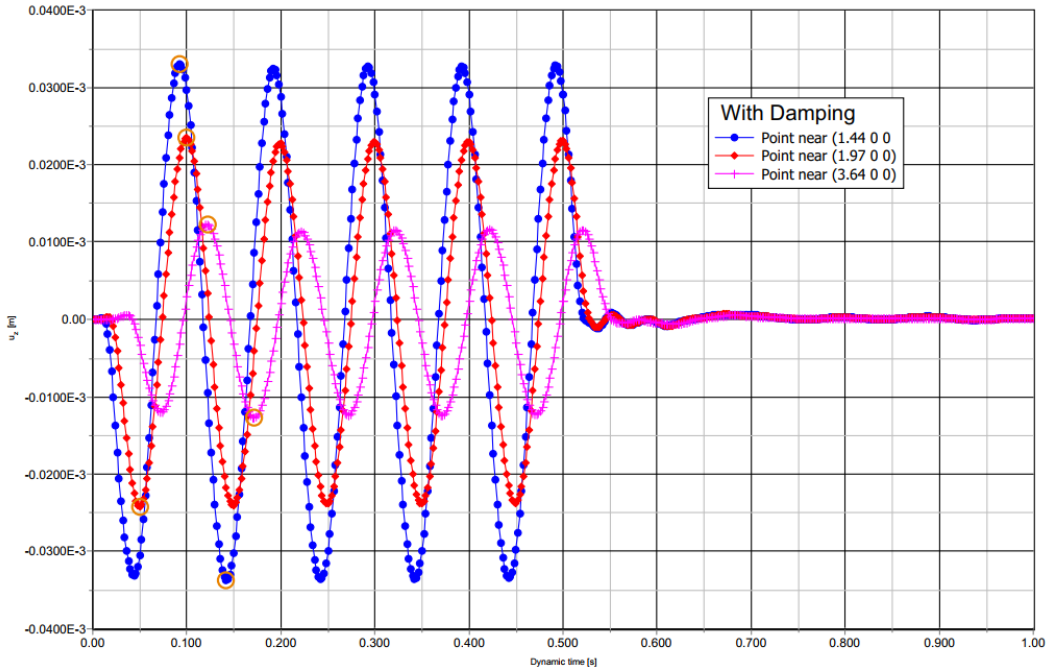


Figure 136: Vertical displ.- time (with damping)

It is possible in the Output program to display displacements, velocities and accelerations at a particular time, by choosing the appropriate option in the **Deformations** menu. Figure 137 (on page 163) shows the total accelerations in the soil at the end of phase 2 (t = 0.5s).

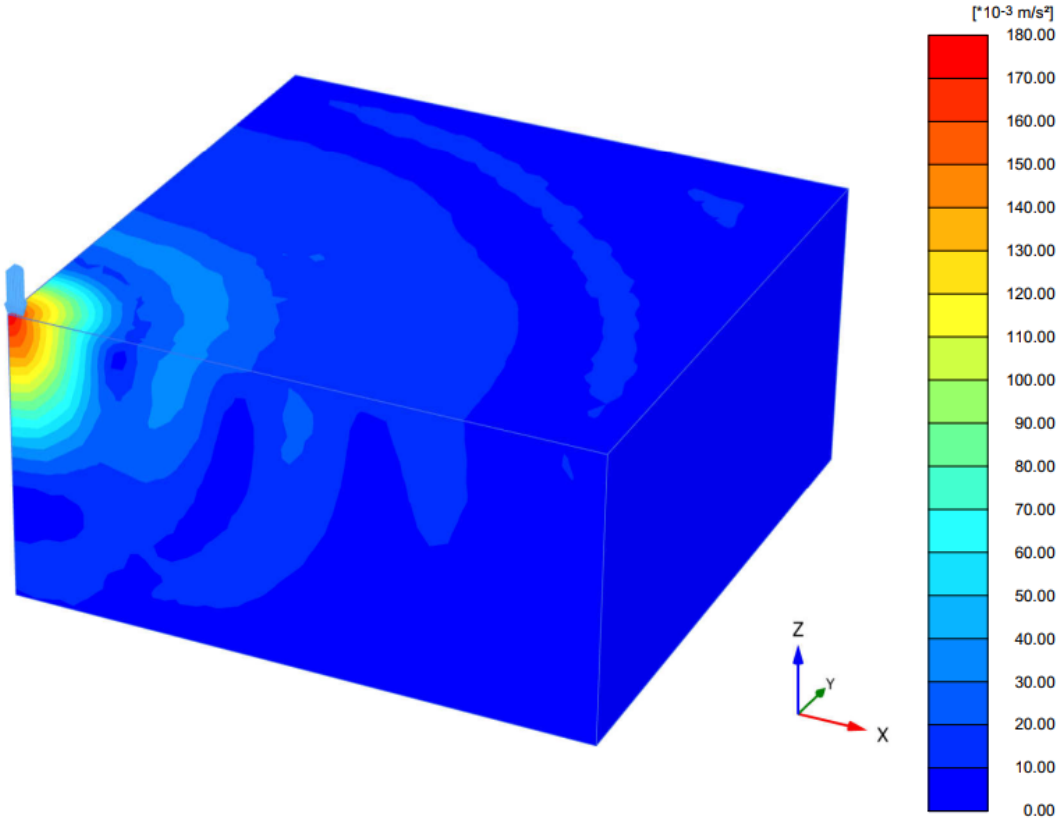


Figure 137: Total accelerations in the soil at the end of Phase 2 (with damping)

9 Free vibration and earthquake analysis of a building [ULT]

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 10m wide and 17m high including the basement. The total height from the ground level is 5 x 3m = 15m and the basement is 2m deep. A value of 5kN/m^2 is taken as the weight of the floors and the walls. The building is constructed on a clay layer of 15m depth underlayed by a deep sand layer. In the model, 25m of the sand layer will be considered.

9.1 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 3m 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:
   a. x_{min} = -80 and x_{max} = 80
   b. y_{min} = 0 and y_{max} = 3

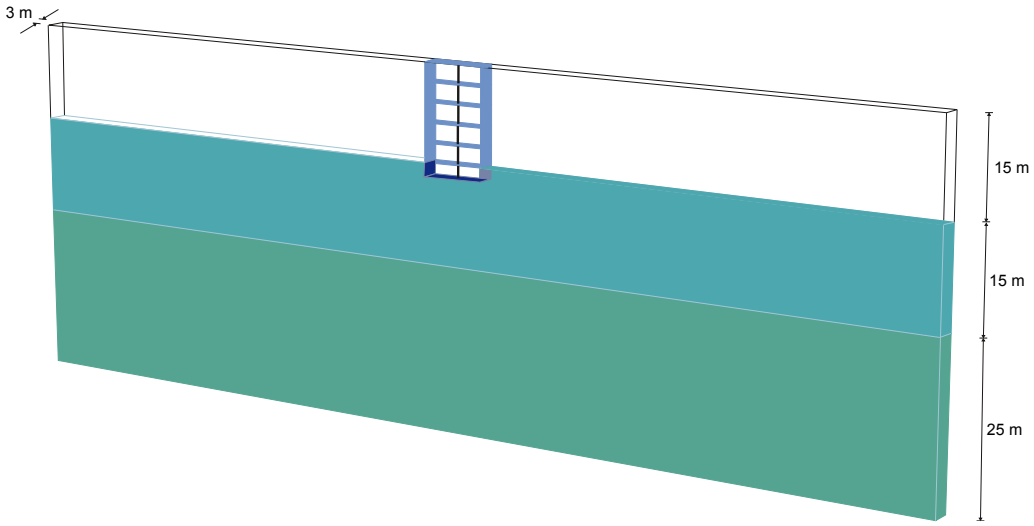


Figure 138: The geometry of the model

9.2 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.

9.3 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 displayed in Table 22 (on page 165).

Table 22: Material properties

Property	Name	Upper clayey layer	Lower sandy layer	Unit
General				
Soil model	Model	HS small	HS small	-
Drainage type	Type	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	16	20	kN/m^3

Property	Name	Upper clayey layer	Lower sandy layer	Unit
General				
Saturated unit weight	γ_{sat}	20	20	kN/m^3
Mechanical				
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.0 \cdot 10^4$	$3.0 \cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$2.561 \cdot 10^4$	$3.601 \cdot 10^4$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$9.484 \cdot 10^4$	$1.108 \cdot 10^5$	kN/m^2
Poisson's ratio	ν_{ur}	0.2	0.2	-
Power for stress-level dependency of stiffness	m	0.5	0.5	-
Shear modulus at very small strains	G_0^{ref}	$2.7 \cdot 10^5$	$1.0 \cdot 10^5$	kN/m^2
Shear strain at which $G_s = 0.722\ G_0$	$\gamma_{0.7}$	$1.2 \cdot 10^{-4}$	$1.5 \cdot 10^{-4}$	-
Cohesion	c'_{ref}	10	5	kN/m^2
Friction angle	φ'	18.0	28.0	°
Dilatancy angle	ψ	0.0	0.0	°

When subjected to cyclic shear loading, the Hardening Soil model with small-strain stiffness will show typical hysteretic behaviour. Starting from the small-strain shear stiffness, G_0^{ref}, the actual stiffness will decrease with increasing shear. Figure 139 (on page 167) and Figure 140 (on page 167) display the Modulus reduction curves, i.e. the decay of the shear modulus with strain.

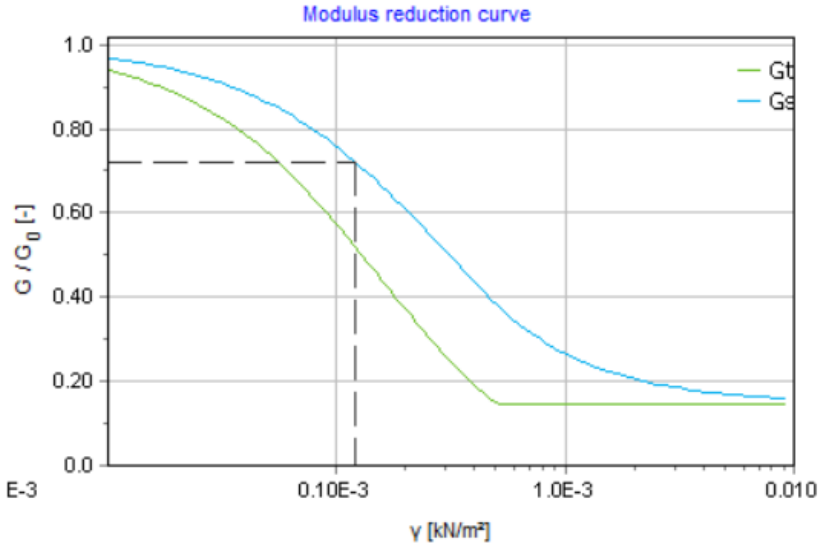


Figure 139: Modulus reduction curves for the upper clayey layer

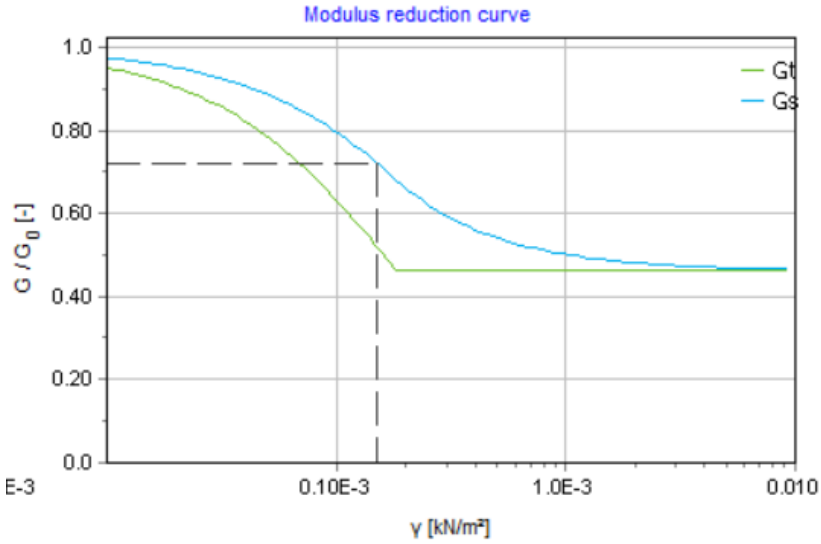


Figure 140: Modulus reduction curve for the lower sandy layer

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

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

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 168). This ratio determines the maximum damping ratio that can be obtained.

Table 23: G_{ur} values and ratio to G_0^{ref}

Parameter	Unit	Upper clayey layer	Lower sandy layer
G_{ur}	kN/m^2	39517	41167
G_0^{ref}/G_{ur}	-	6.83	2.43

Figure 141 (on page 168) and Figure 142 (on page 169) 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.

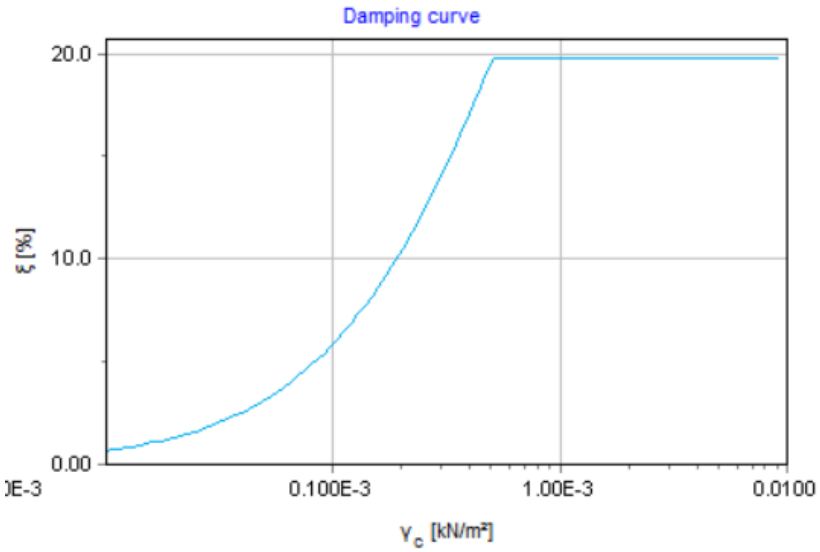


Figure 141: Damping curve for the upper clayey layer

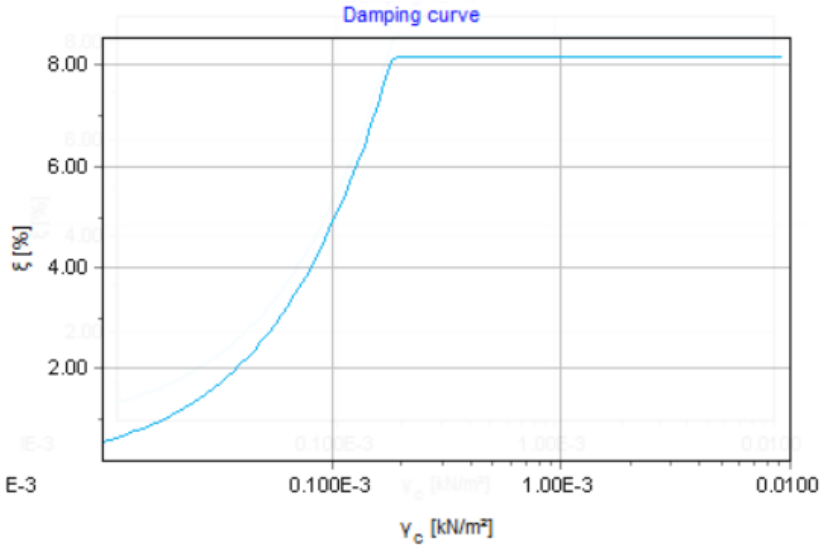


Figure 142: Damping curve for the lower sandy layer

1. Create the material data set according to Table 22 (on page 165) 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.

9.4 Definition of structural elements

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

9.4.1 Create a building

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

For the building two material data sets are needed and their corresponding material properties are described in Table 24 (on page 169):

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

Parameter	Name	Rest of building	Basement	Unit
General				
Material type	Type	Elastic	Elastic	-

Parameter	Name	Rest of building	Basement	Unit
General				
Unit weight	γ	33.33	50	kN/m^3
Rayleigh damping (Direct)	α	0.2320	0.2320	-
	β	$8 \cdot 10^{-3}$	$8 \cdot 10^{-3}$	-
Mechanical				
Isotropic	-	Yes	Yes	-
Young's modulus	E_1	$3 \cdot 10^7$	$3 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{12}	0	0	-
Thickness	d	0.3	0.3	m

Structure definition

To create the floors and walls 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 2m.
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 -10m.
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 **Create** > **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 169). Note that two different material data sets are used for the basement and the rest of the building respectively.
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.

Central column

The central column of the structure is modelled using the **Node-to-node anchor** feature. The modelling procedure is described as follows:

1. Create a **Line** through points (0 1.5 -2) and (0 1.5 0) corresponding to the column in the basement floor.
2. Create a **Line** through points (0 1.5 0) and (0 1.5 3) corresponding to the column in the first floor.
3. 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 3m.
4. Select the created lines, right-click and select the **Create** > **Create node-to-node anchor** option from the appearing menu.
5. Create the material data set according to Table 25 (on page 171) and assign it to the anchors.

Table 25: Material properties of the node-to-node anchor

Parameter	Name	Column	Unit
Material type	Type	Elastic	-
Normal stiffness	EA	$2.5 \cdot 10^6$	kN

9.4.2 Create the loads

Creation of the static load

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

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).

Earthquake definition

The earthquake is modelled by imposing a prescribed displacement at the bottom boundary and assigning dynamic multipliers to the prescribed displacements:

- **To define the prescribed displacement:**
  1. Create a surface prescribed displacement passing through (-80 0 -40), (80 0 -40), (80 3 -40) and (-80 3 -40).
  2. 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.

- **To define the dynamic multipliers for the prescribed displacement:**

1. 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.
2.  To add a multiplier click the corresponding button in the **Multipliers** window.
3. From the **Signal** drop-down menu select the **Table** option.
4. The file containing the earthquake data is available on Bentley Communities .
5. 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.
6.  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.
7. Select the **Acceleration** option in the **Data type** drop-down menu.
8. Select the **Drift correction** options and click **OK** to finalize the definition of the multiplier.
9. In the **Dynamic multipliers** window the table and the plot of the data is displayed (See Figure 143 (on page 172)).

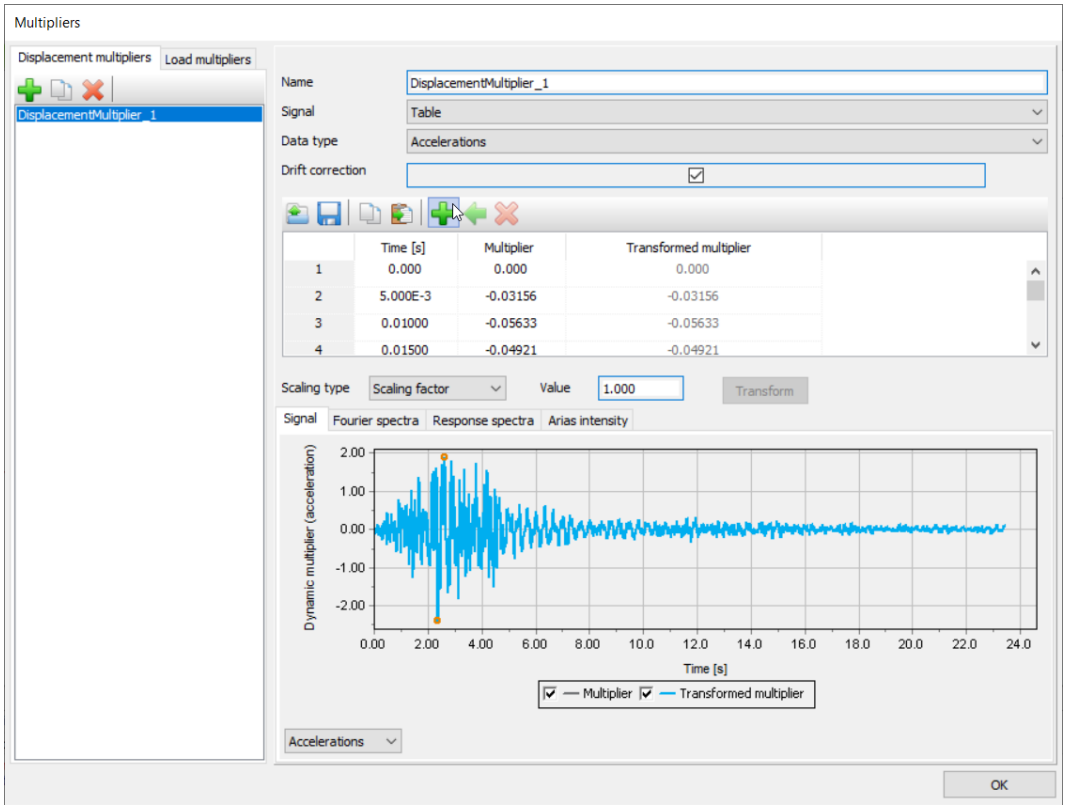


Figure 143: Dynamic multipliers window

10. In the **Model explorer** expand the **Surface displacements** subtree and in DynSurfaceDisplacement_1 assign the Multiplier $_x$ to the x- component by selecting the option in the drop-down menu.

9.4.3 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 click **Create** > **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 click **Create** > **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** > **Create positive interface** to add an interface inside the model.

9.5 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.

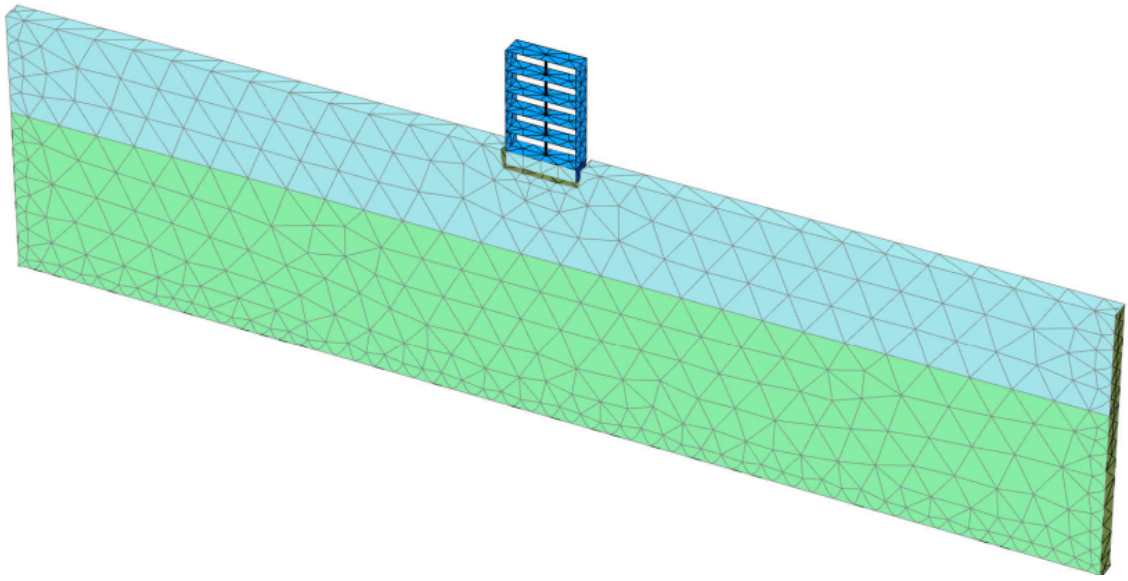


Figure 144: The generated mesh

9.6 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.

9.6.1 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.

9.6.2 Phase 1 - Building construction

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.

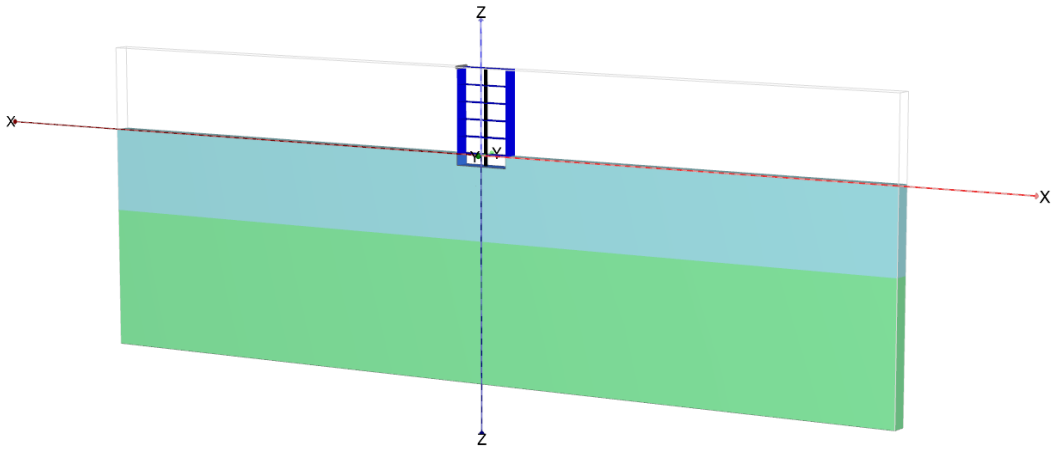


Figure 145: Construction of the building

9.6.3 Phase 2 -Excitation

1. Add a new calculation phase (Phase_2).
2. In the **Phases** window in the **Deformation control parameters** subtree select the **Reset displacement to zero**. 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** .

9.6.4 Phase 3 - Free Vibration

1. Add a new calculation phase (Phase_3).
2. In the **Phases** window as **Calculation type** select the **Dynamic** option.
3. Set the **Time interval** parameter to 5sec.
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 directions are set to viscous. Select the **None** option for the boundaries in the y direction. Set the boundary Zmin to **Viscous** (See Figure 146 (on page 176)).

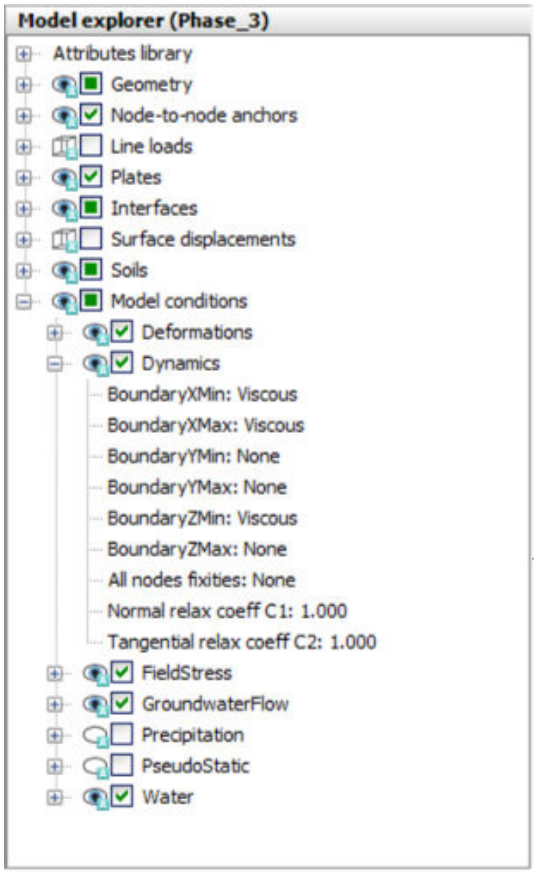


Figure 146: Boundary conditions for dynamics calculations (Phase_3)

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.

9.6.5 Phase 4 - Earthquake

1. Add a new phase (Phase_4).
2. In the **Phases** window set the **Start from phase** option to Phase 1 (construction of building).
3. As **Calculation type** select the **Dynamic** option.
4. Set the **Dynamic time interval** parameter to 20 sec.
5. In the **Deformation control parameters** subtree select the **Reset displacement to zero**. 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.
7. Set the **Max steps** to 1000.
8. Set the **Time step determination** to **Manual** and the **Number of sub steps** to 4.

Note:

- The dynamic time interval is set to 20 seconds with a time interval of 0.005 seconds which gives 20/0.005=4000. So 4000 steps are required for the calculation. Therefore, the **Max steps** is set to 1000 and the **Number of sub steps** is set to 4. However, the automatic time stepping suggests smaller time steps due to one or two relatively small elements, which requires a sensitivity analysis for the time step size. For more information on time step for dynamic calculations please visit Bentley communities.
- For more information on drift correction and input signal please visit Bentley communities.

9. In the **Model explorer** expand the **Model conditions** subtree.
10. 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** (see Figure 147 (on page 178)).

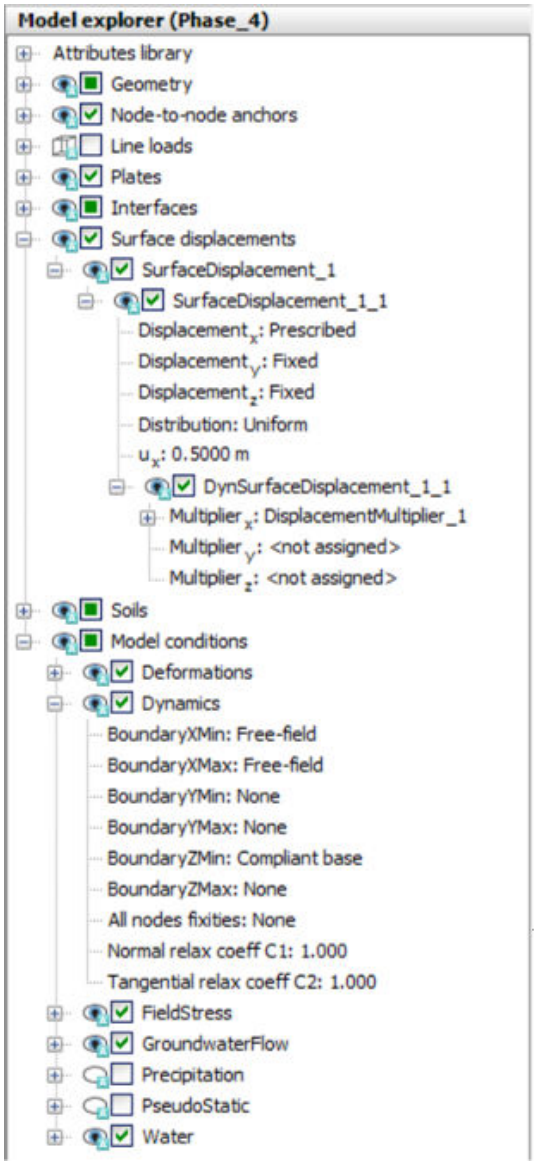


Figure 147: Boundary conditions for dynamics calculations (Phase_4)

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

Note:

- The dynamic time interval is set to 20 seconds with a time interval of 0.005 seconds which gives 20/0.005=4000. So 4000 steps are required for the calculation. Therefore, the **Max steps** is set to 1000 and the **Number of sub steps** is set to 4. However, the automatic time stepping suggests smaller time steps due to one or two relatively small elements, which requires a sensitivity analysis for the time step size. For more information on the topic of time step used in dynamic calculations please visit Bentley communities.
- For more information on drift correction and input signal please visit Bentley communities.

12. In the **Model explorer** activate the **Surface displacement** and its dynamic component. Set the value of u_x to 0.5m. 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.

9.6.6 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.

9.7 Results

Figure 148 (on page 179) shows the deformed structure at the end of the Phase 2 (application of horizontal load).

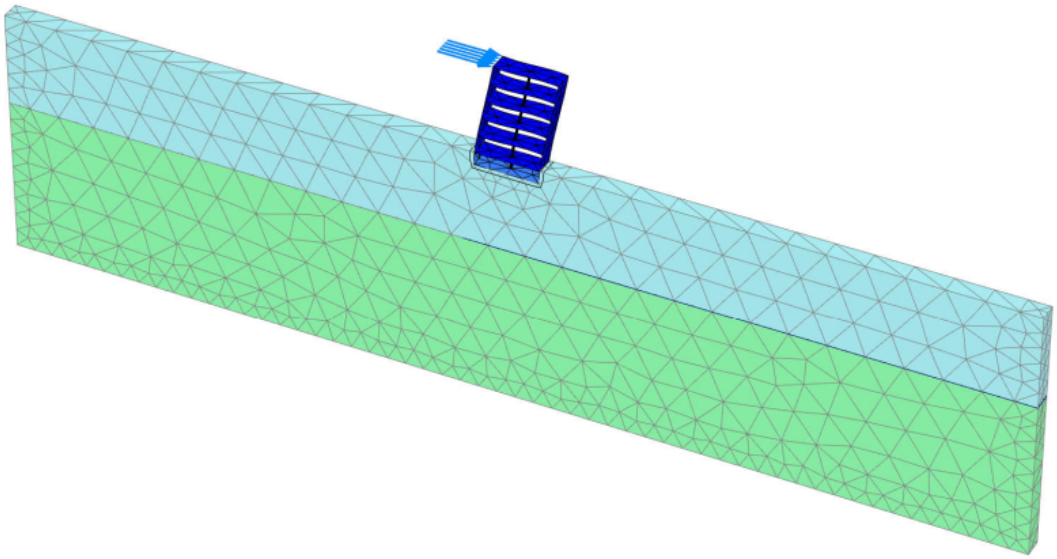


Figure 148: Deformed mesh of the system at the end of Phase_2

Figure 149 (on page 180) 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.

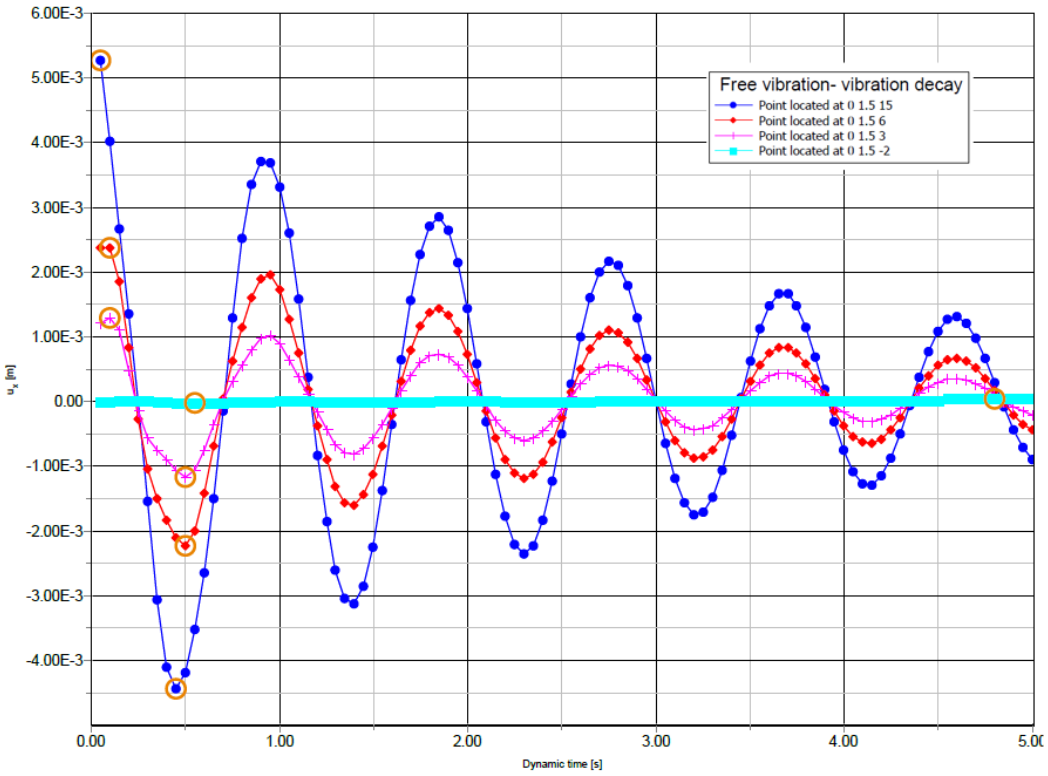


Figure 149: Time history of displacements (Free vibration)

In the **Curve generation** window under the **Fourier tabsheet** select **Power (spectrum)**, subsequently in the Total displacements subtree select u_x and click OK to create the plot. From Figure 150 (on page 181) 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.

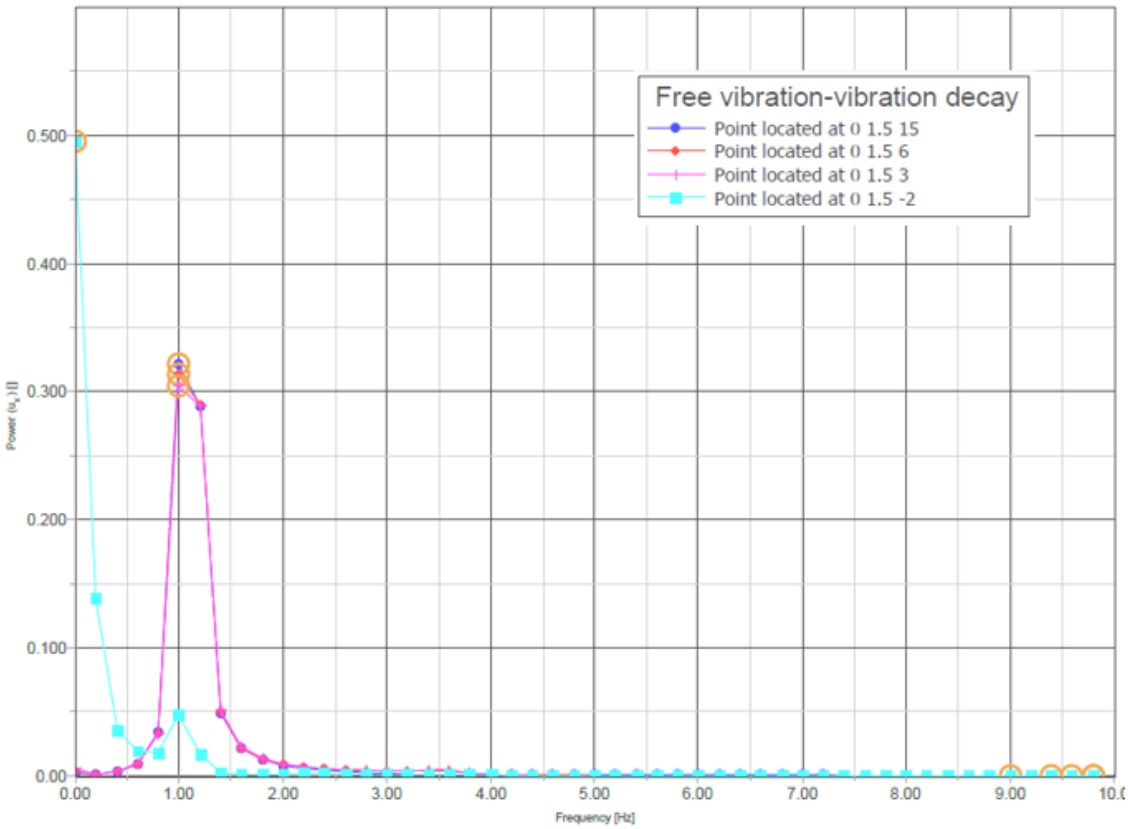


Figure 150: Frequency representation (spectrum - Free vibration)

Figure 151 (on page 182) 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.

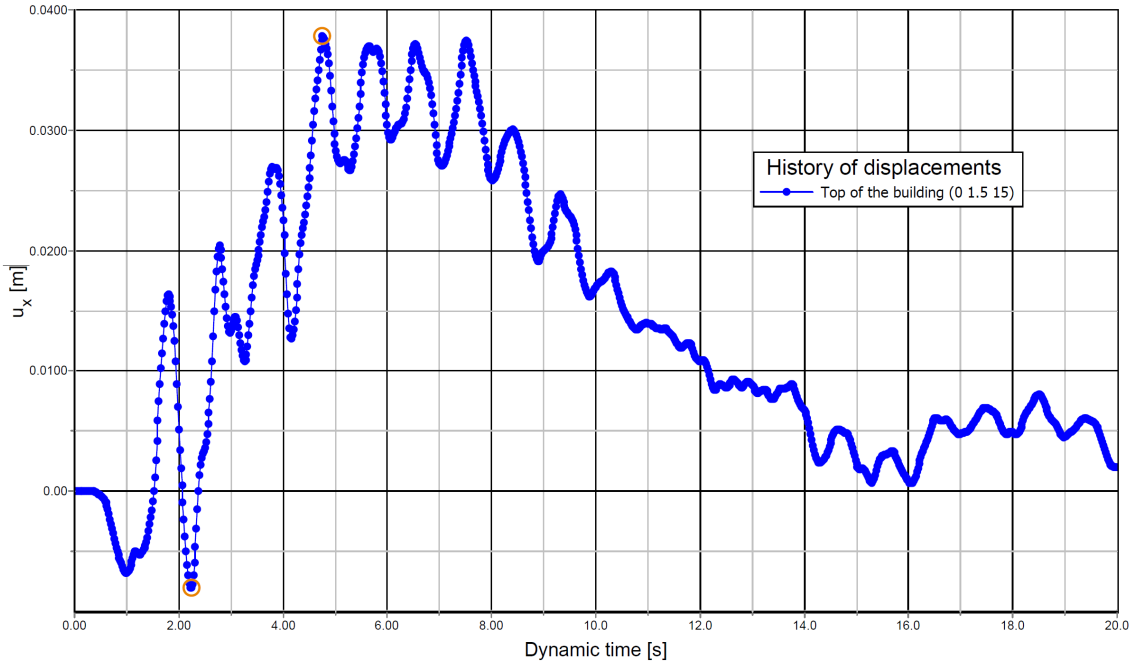


Figure 151: Time history of displacements of the top of the building (Earthquake)

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 Figure 152 (on page 183).

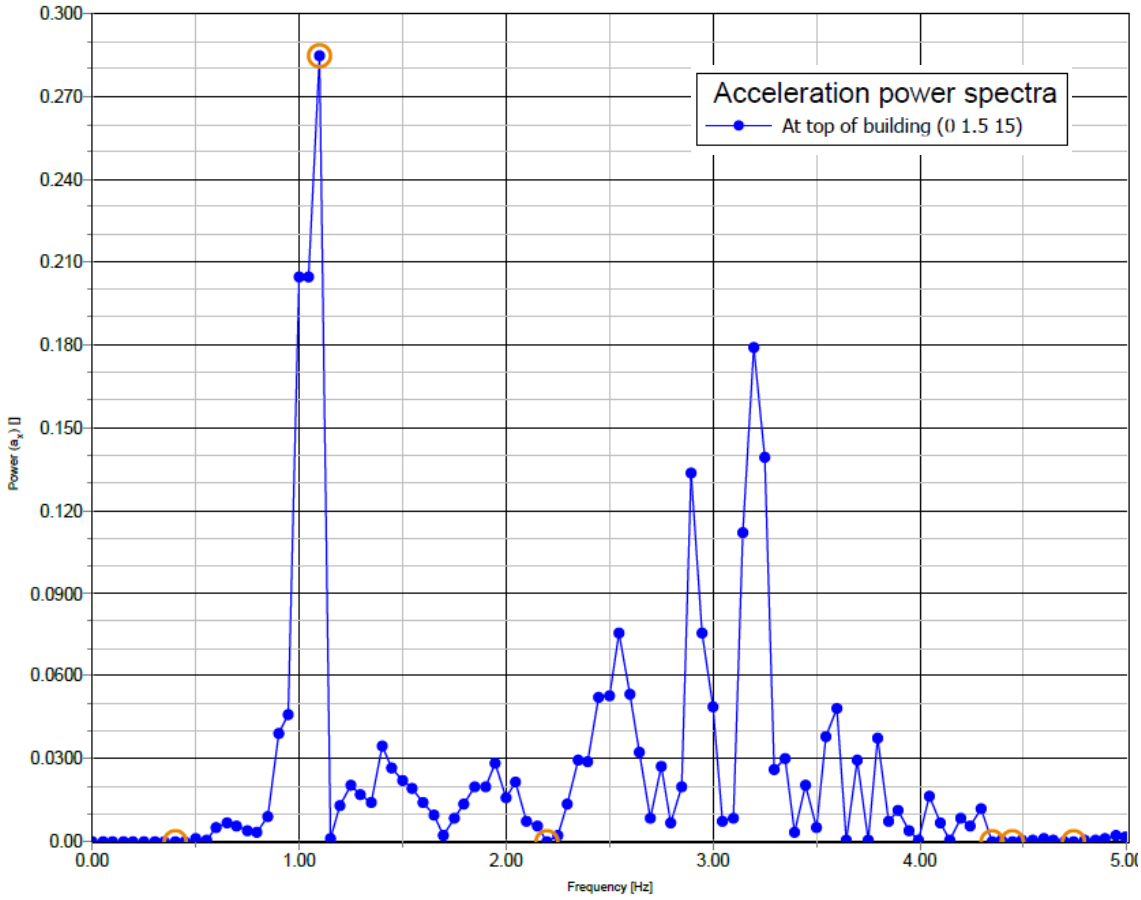
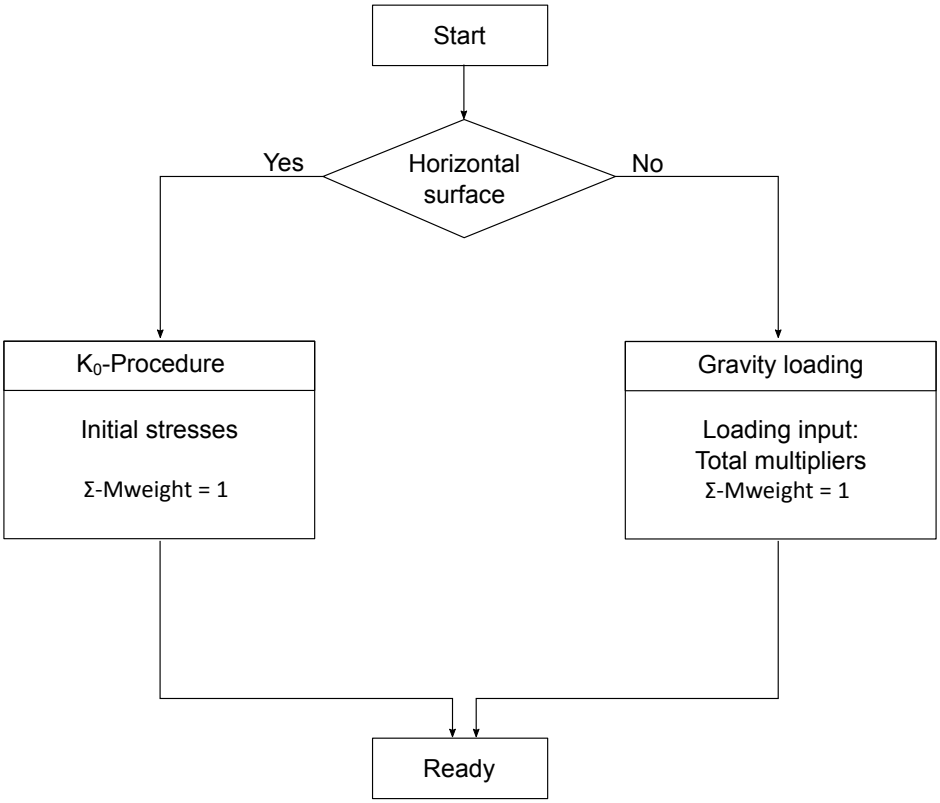


Figure 152: Acceleration power spectra at (0 1.5 15)

Appendices

A

Calculation scheme for initial stresses due to soil weight



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

