1

Foundation in overconsolidated clay

1.1 Introduction

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.

1.2 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.

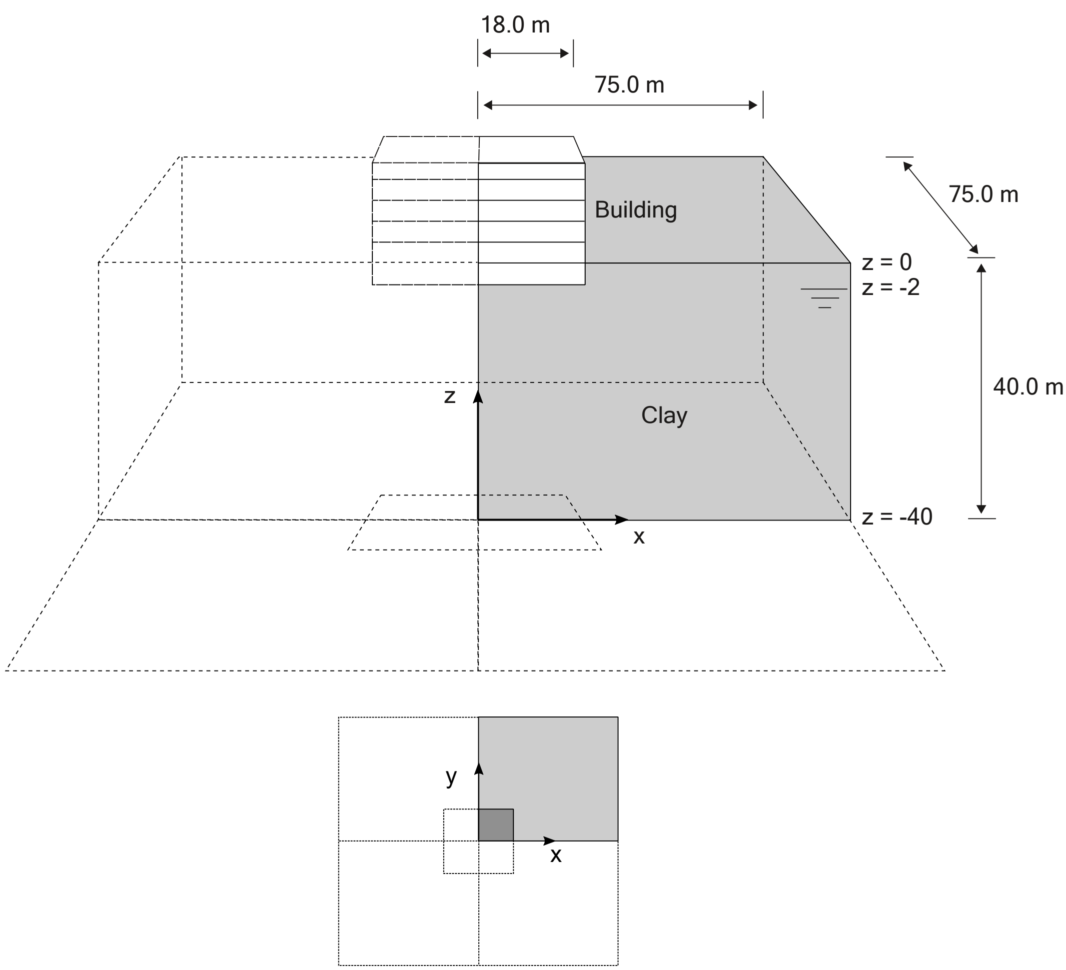


Figure 1–1: Geometry of a square building on a raft foundation

1.3 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.

1.3.1 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 1–2 (p. 9)).

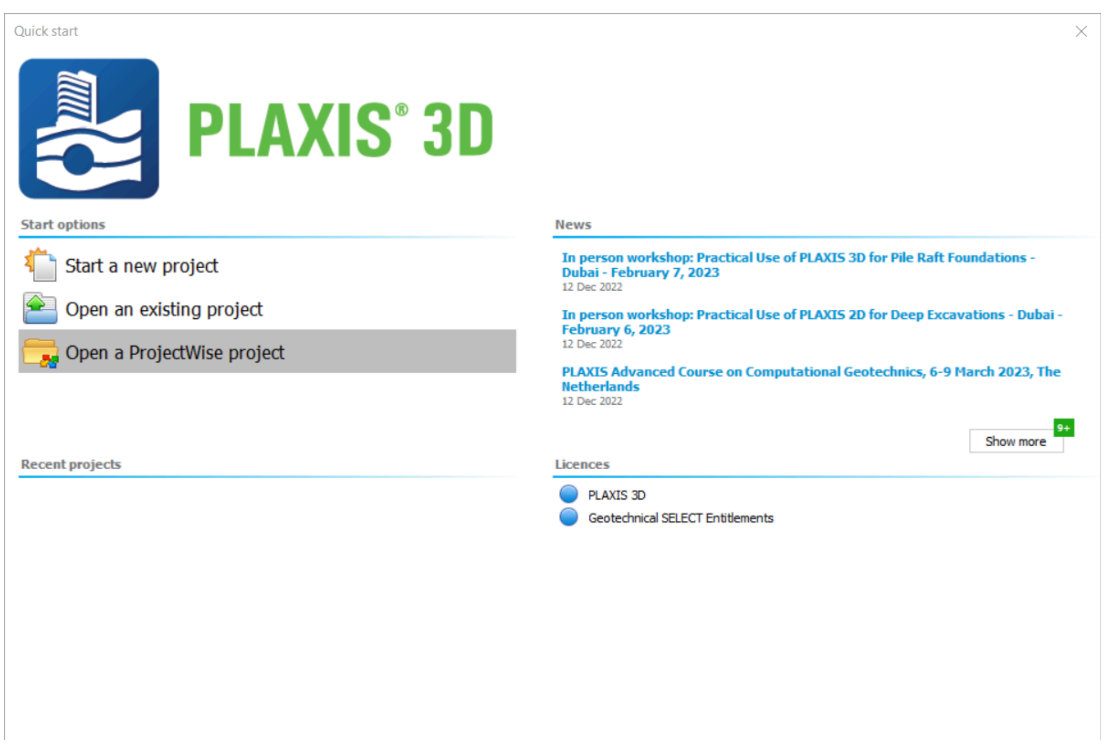


Figure 1–2: Quick Start window

2 Click **Start a new project**.

The **Project properties** window (see Figure 1–3 (p. 10)) appears with the tabsheets: **Project**, **Model** and **Cloud services**.

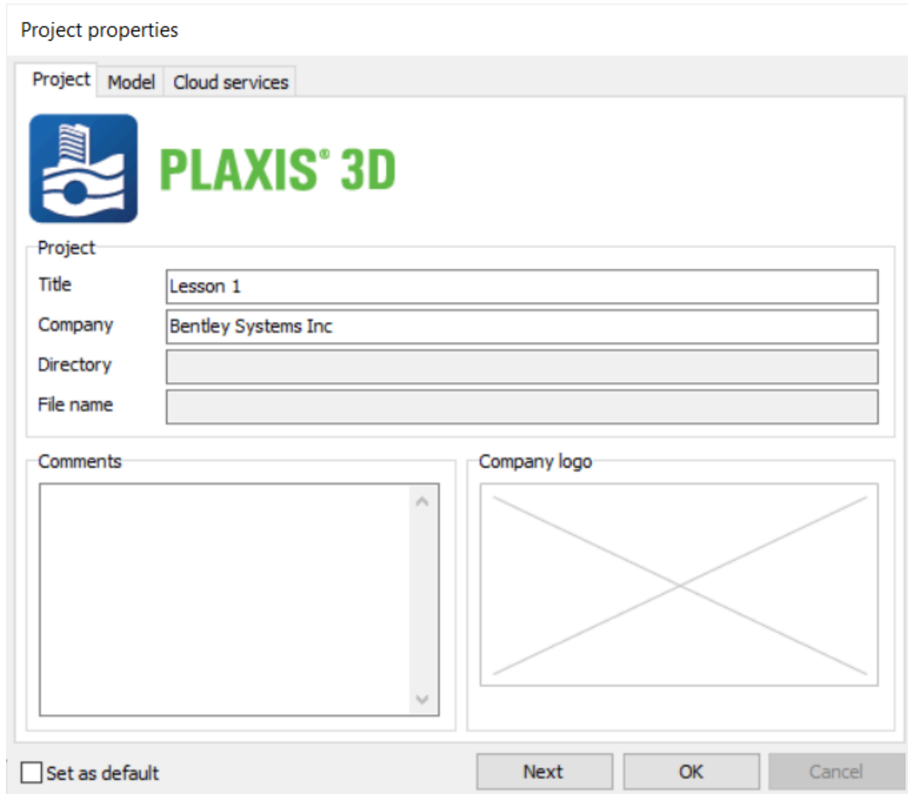


Figure 1–3: Project properties window

> **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 1–4 (p. 11):

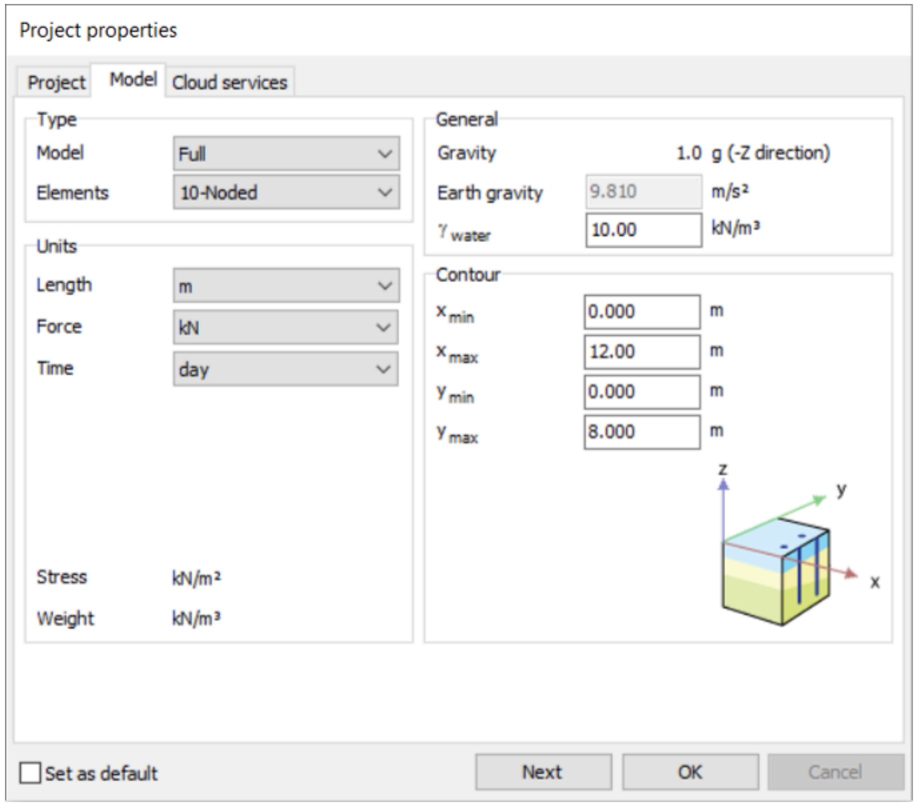


Figure 1–4: Project properties - Model tab

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 ...**.

1.3.2 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 1–5 (p. 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.

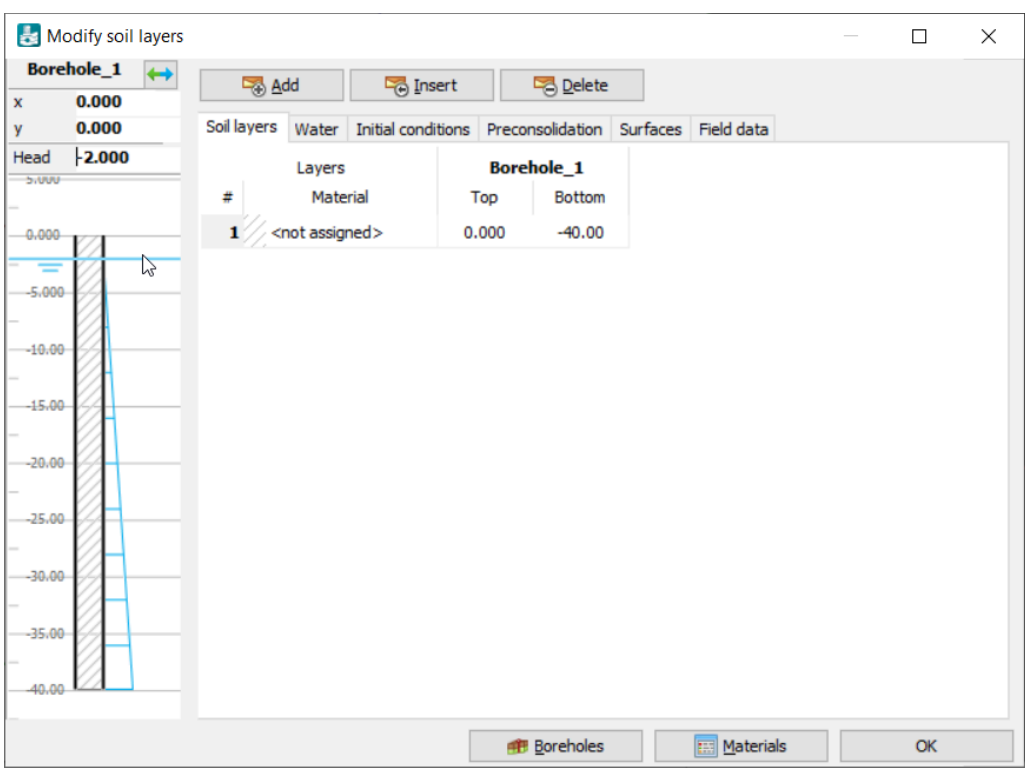


Figure 1–5: Modify soil layers window

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

1.3.3 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 1–1 (p. 13), and they are used as material data sets for **Soils and interfaces**.

Table 1–1: 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	ν *(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:

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 1–6 (p. 14).

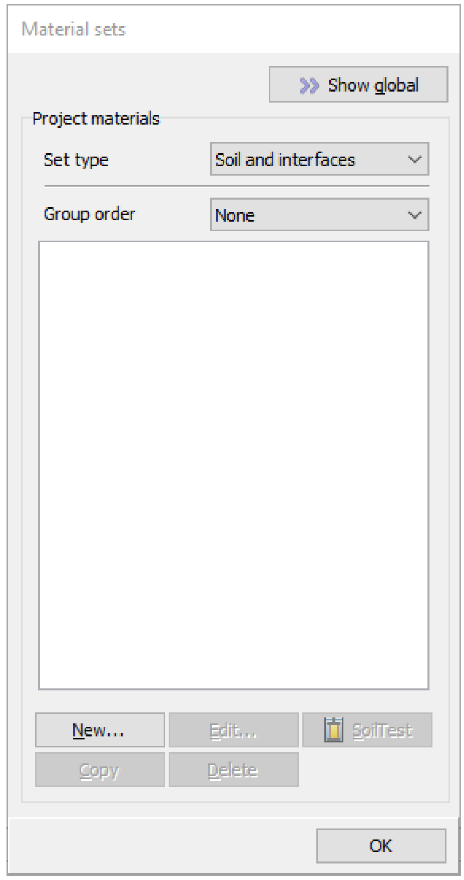


Figure 1–6: Material sets window

> **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**.

1.3.3.1 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 1–7 (p. 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 1–1 (p. 13). Keep the unmentioned **Void ratio** parameters as their default values.

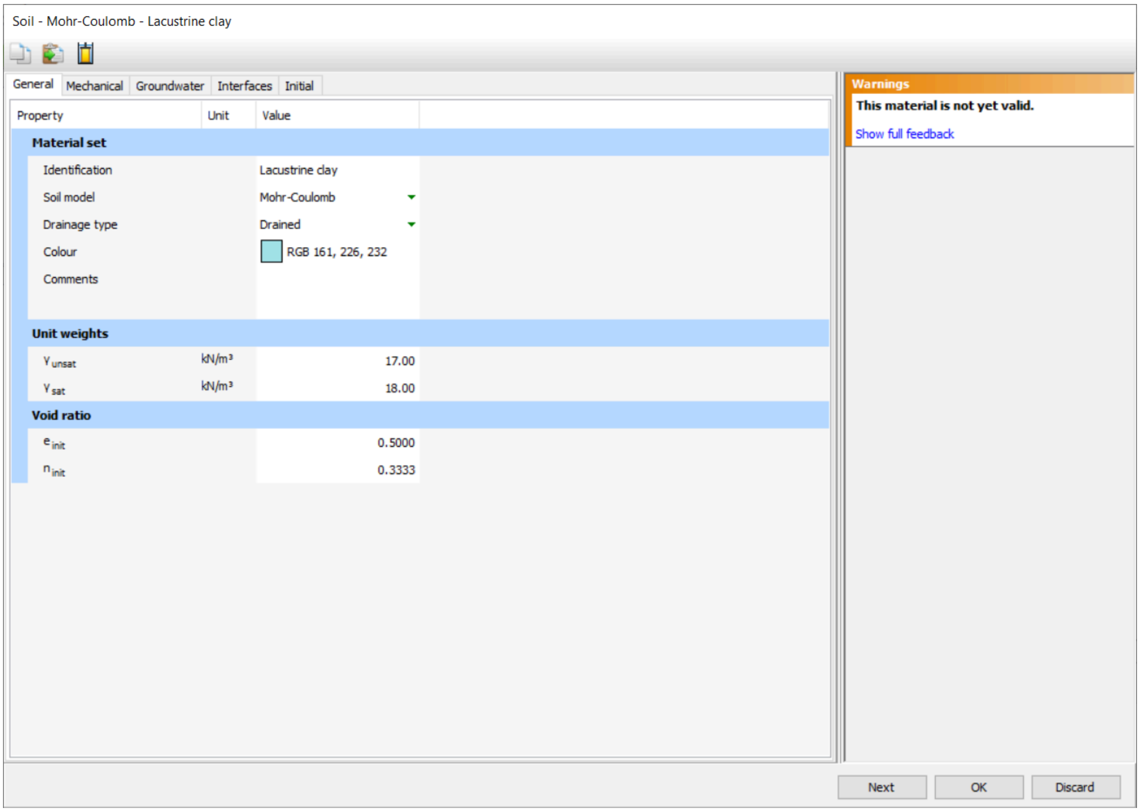


Figure 1–7: General tabsheet of the Soil and interfaces data set window

Note:

a. As displayed in Figure 1–7 (p. 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.

b. 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 1–1 (p. 13) in the corresponding boxes of the **Mechanical** tabsheet.

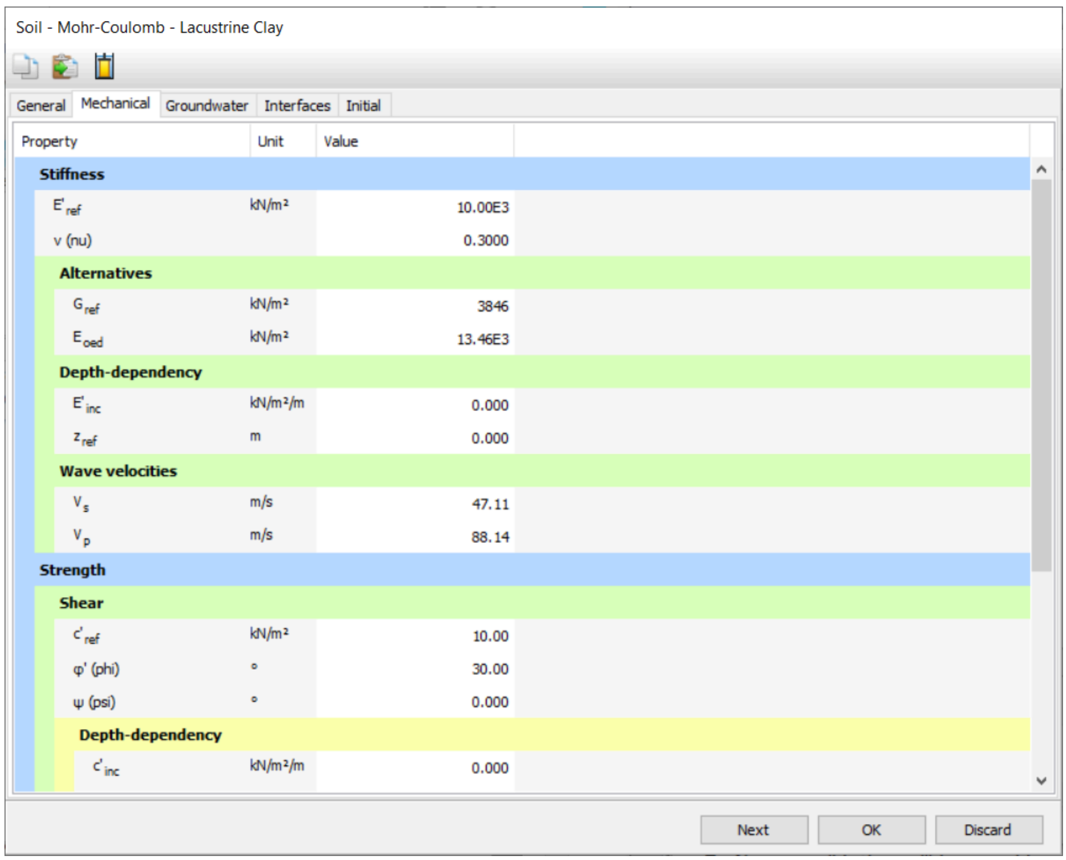


Figure 1–8: Mechanical tabsheet of the Soil and interfaces data set window

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.

1.3.3.2 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 1–9 (p. 17).

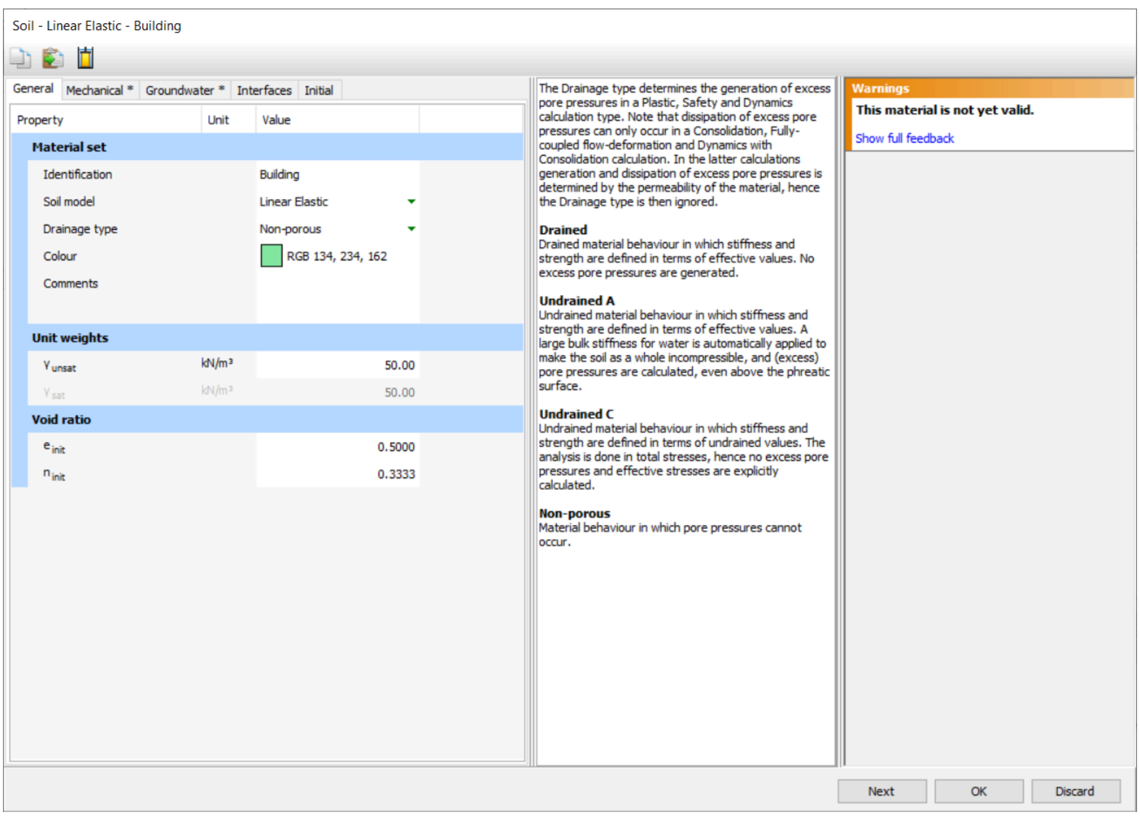


Figure 1–9: Drainage type - General tabsheet of the Soil and interfaces data set window

4. Enter the unit weight in the **General properties** box according to the material data as listed in Table 1–1 (p. 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 1–1 (p. 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. .

1.3.4 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 1–10 (p. 18)).

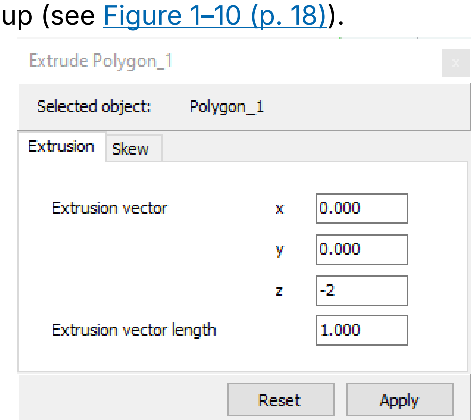


Figure 1–10: Extrude window

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.

1.3.5 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 1–11 (p. 19)).

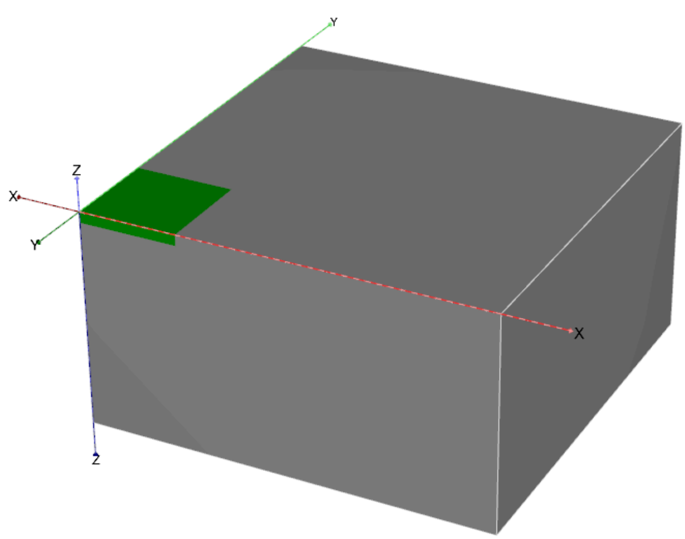


Figure 1–11: The indication of the local refinement in the model

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 1–12 (p. 20).

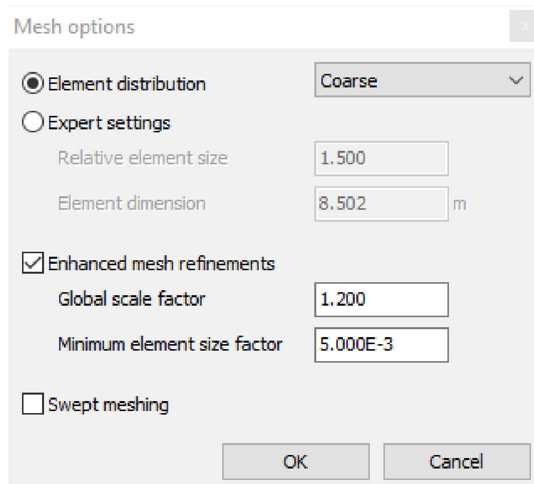


Figure 1–12: Mesh options window

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 1–13 (p. 20)).

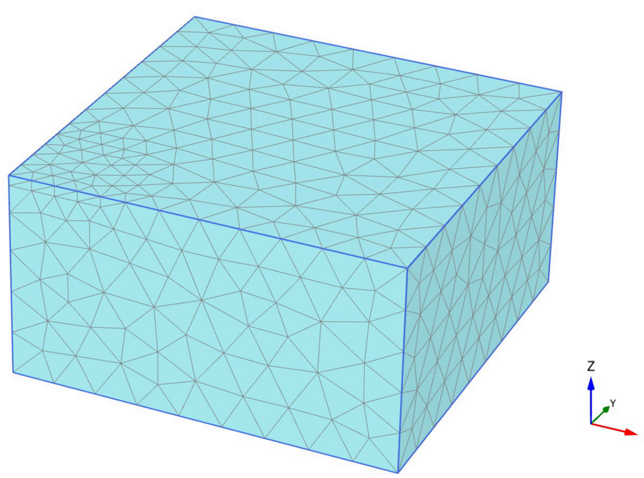


Figure 1–13: Generated mesh in the Output window

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

1.3.6 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.

1.3.6.1 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 1–14 (p. 21)), is automatically created and selected in the **Phases explorer**:

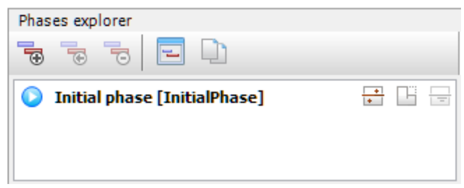


Figure 1–14: ***Phases explorer***

   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 1–15 (p. 21).

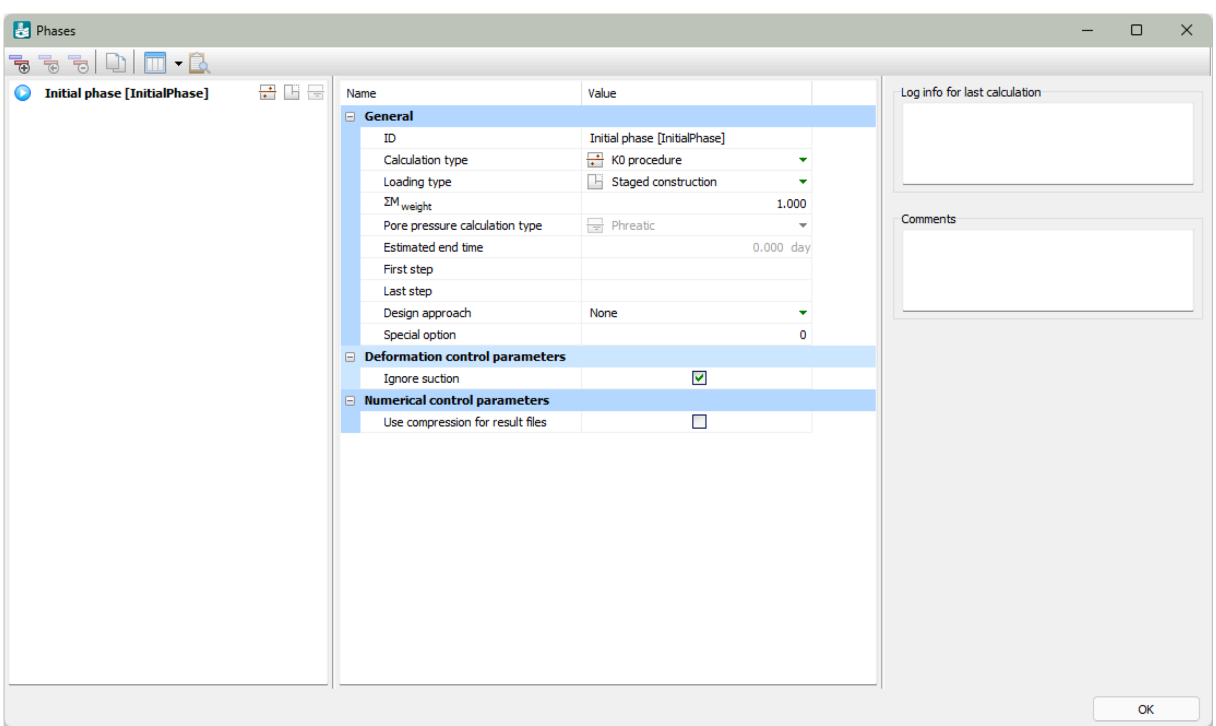


Figure 1–15: The ***Phases*** **window for** ***Initial phase***

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**.

1.3.6.2 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 1–16 (p. 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.

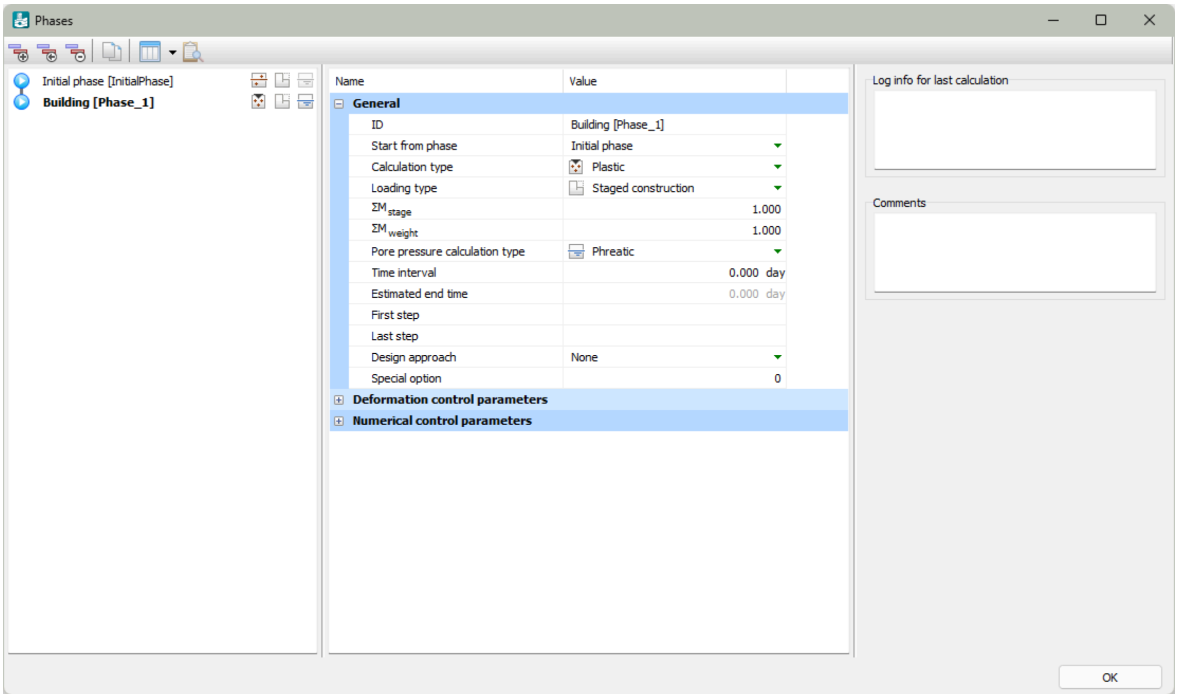


Figure 1–16: The *Phases* window for *Building* phase

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.

1.3.6.3 | 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 1–17 (p. 24)) which gives information about the progress of the actual calculation phase.

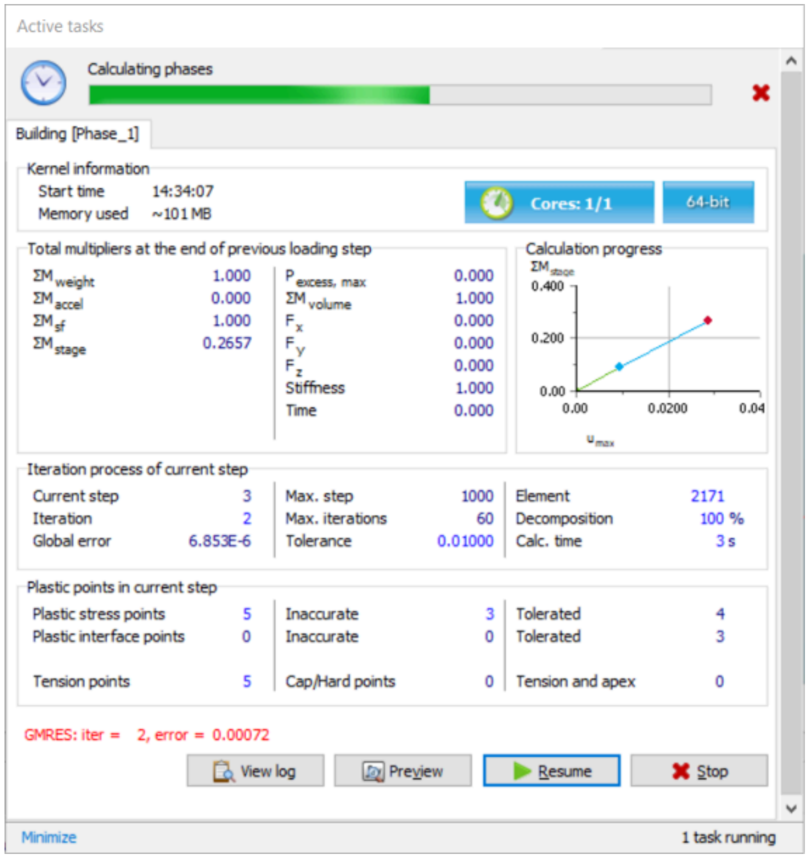


Figure 1–17: *Active task* window displaying the calculation progress

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.

1.3.7 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 1–18 (p. 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.

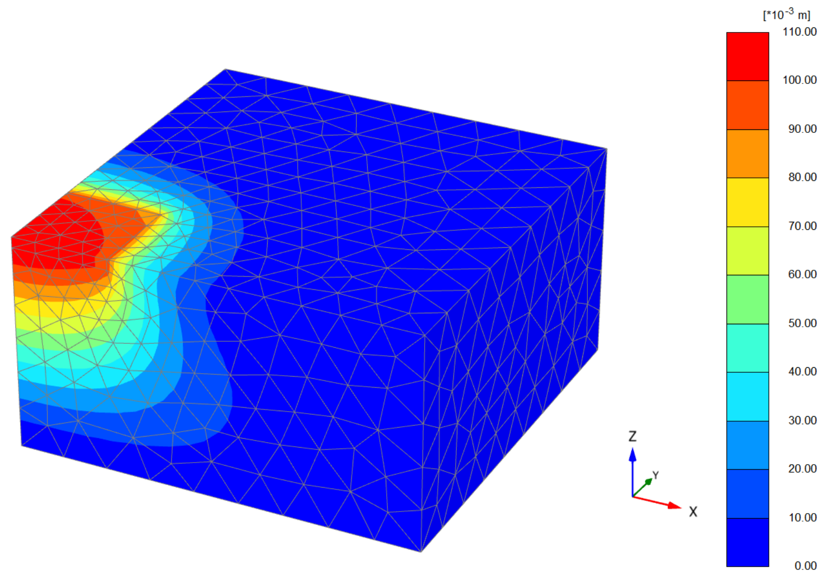


Figure 1–18: Shadings of *Total displacements* at the end of the last phase

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.

1.4 Case B: Raft foundation

1.4.1 Introduction

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.

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

1.4.2 Geometry

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 1–19 (p. 26).

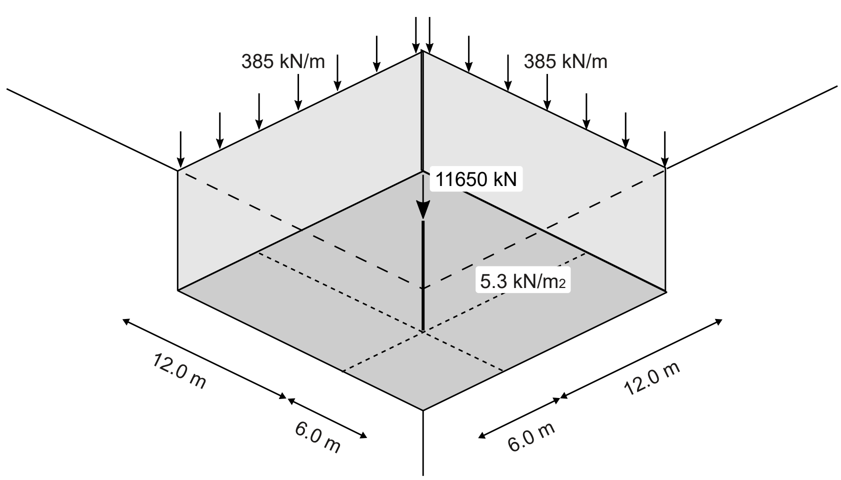


Figure 1–19: Geometry of the basement

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.

1.4.1 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`).

1.4.2 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 1–20 (p. 28)).

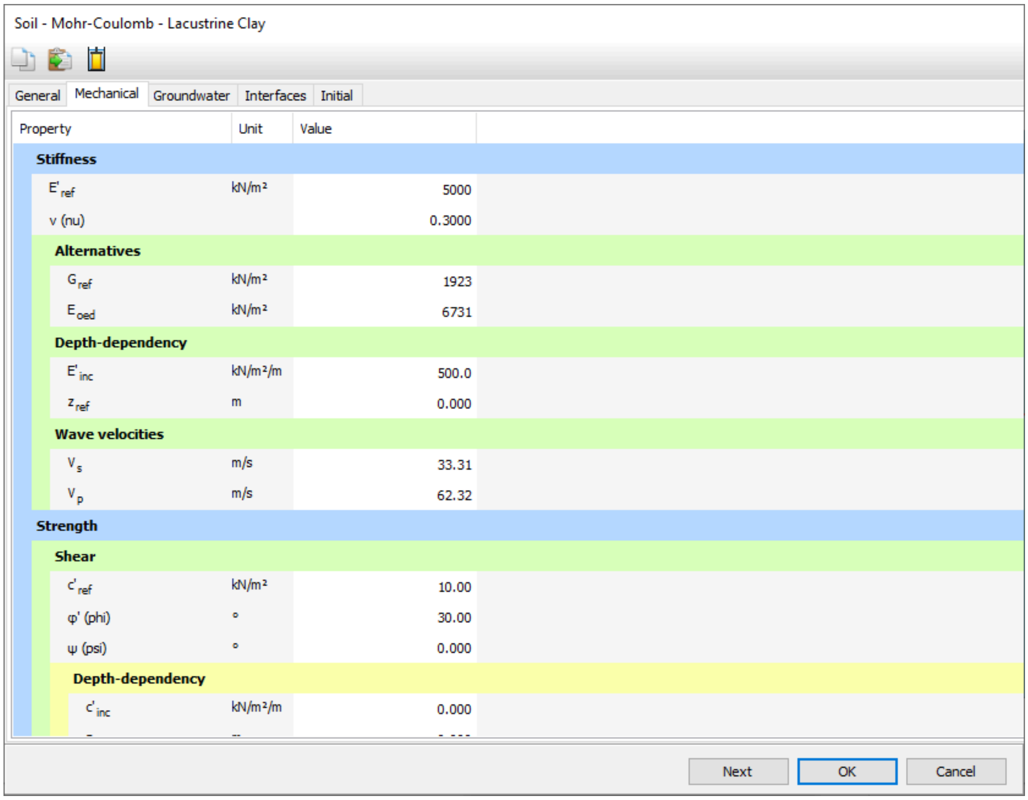


Figure 1–20: Stiffness Depth-dependency parameters

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

1.4.3 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 1–2: 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 1–3: 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

1. Creation of basement floor and basement walls

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 1–21 (p. 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.

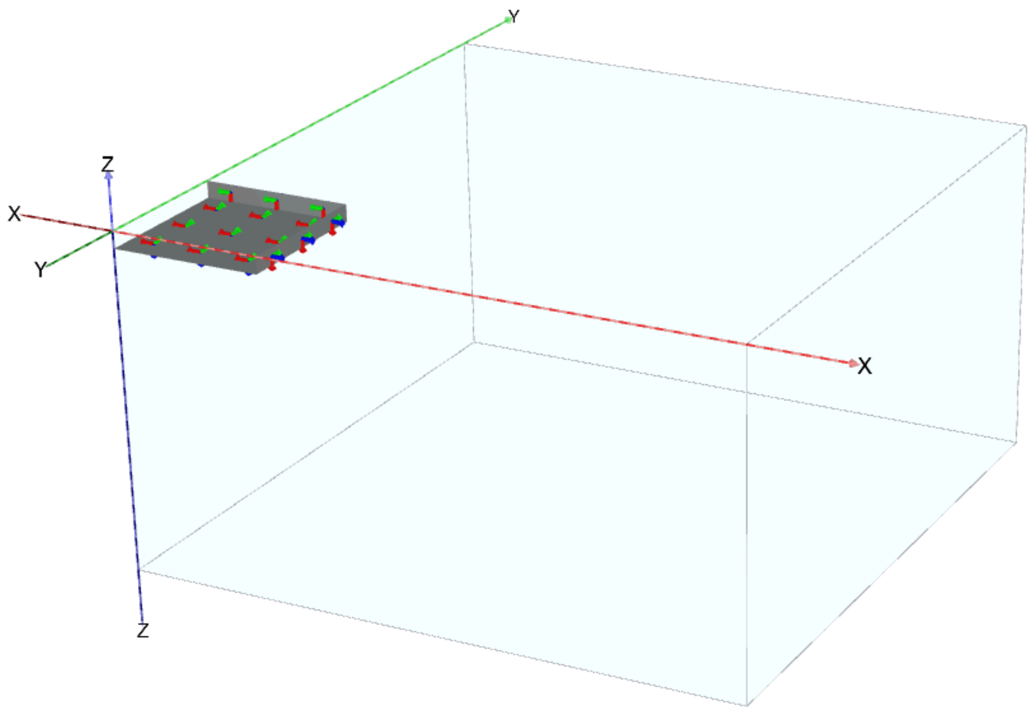


Figure 1–21: Location of plates in the project

Note: To visualise the local axes displayed in Figure 1–21 (p. 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 1–2 (p. 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 1–22 (p. 31).

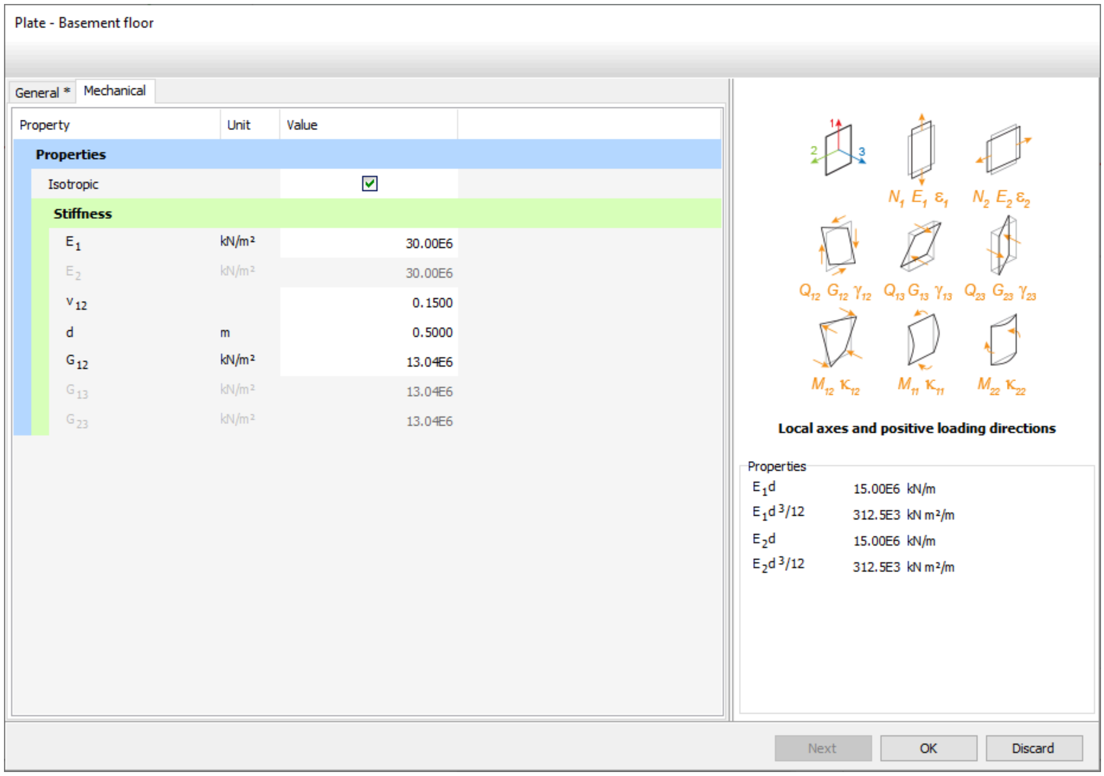


Figure 1–22: Mechanical properties of basement floor plate

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.

2. Creation basement columns and basement beams

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 1–3 (p. 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.

1.4.4 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.

1.4.5 Define and perform the calculation

Proceed to the **Staged construction mode**.

1.4.5.1 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**.

1.4.5.2 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.

1.4.5.3 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.

1.4.6 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 1–23 (p. 34). If the title is not visible, select this option from the **View** menu.

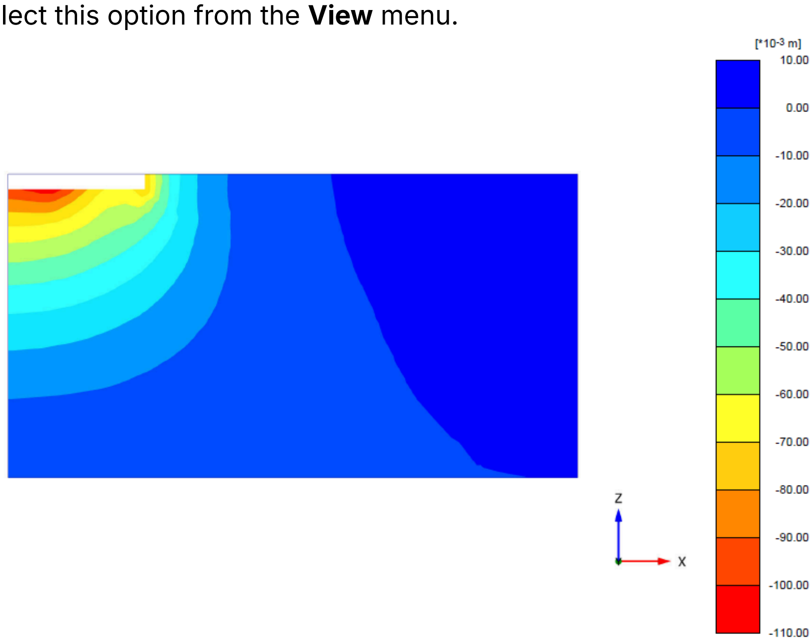


Figure 1–23: Cross section showing the total vertical displacement

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 1–24 (p. 35).

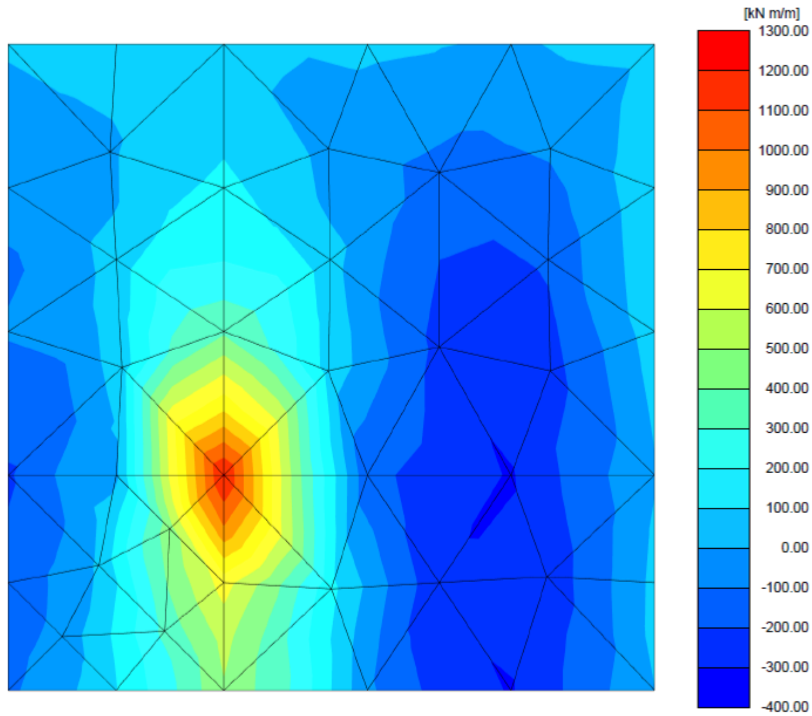


Figure 1–24: Bending moments in the basement floor

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.

1.5 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.

1.5.1 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`).

1.5.2 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 1–4: 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
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 1–4 (p. 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.

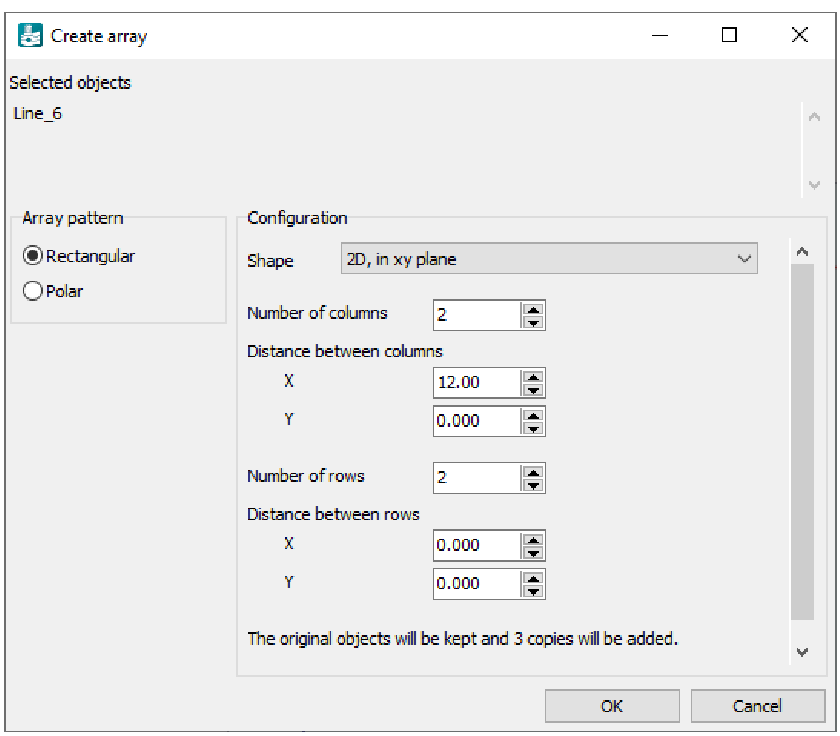


Figure 1–25: *Create array* window

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

1.5.3 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 1–26 (p. 38):

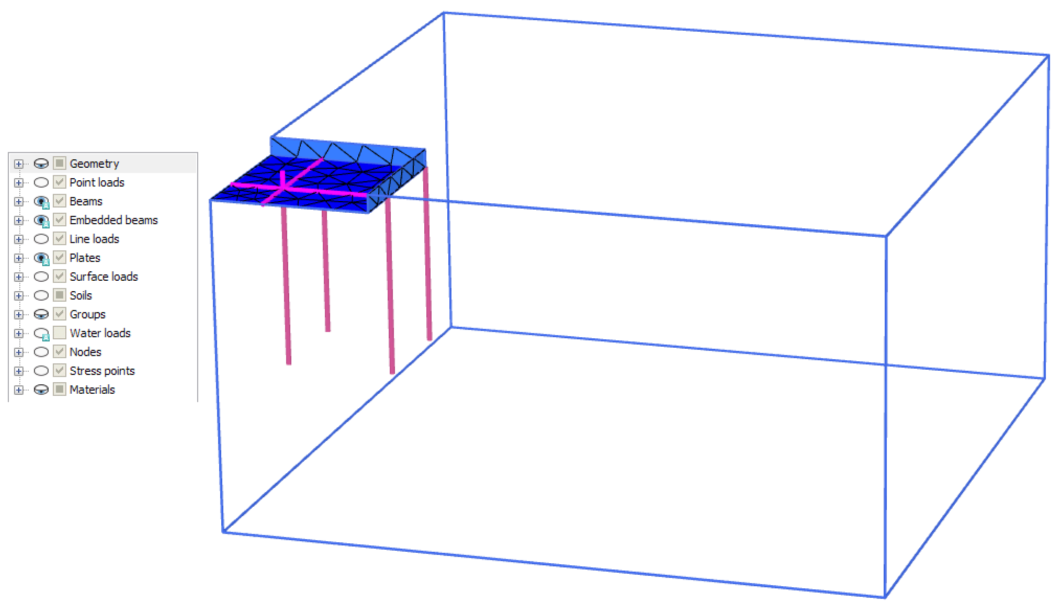


Figure 1–26: Partial geometry of the model in the Output

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

1.5.4 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.

1.5.5 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 1–27 (p. 39):

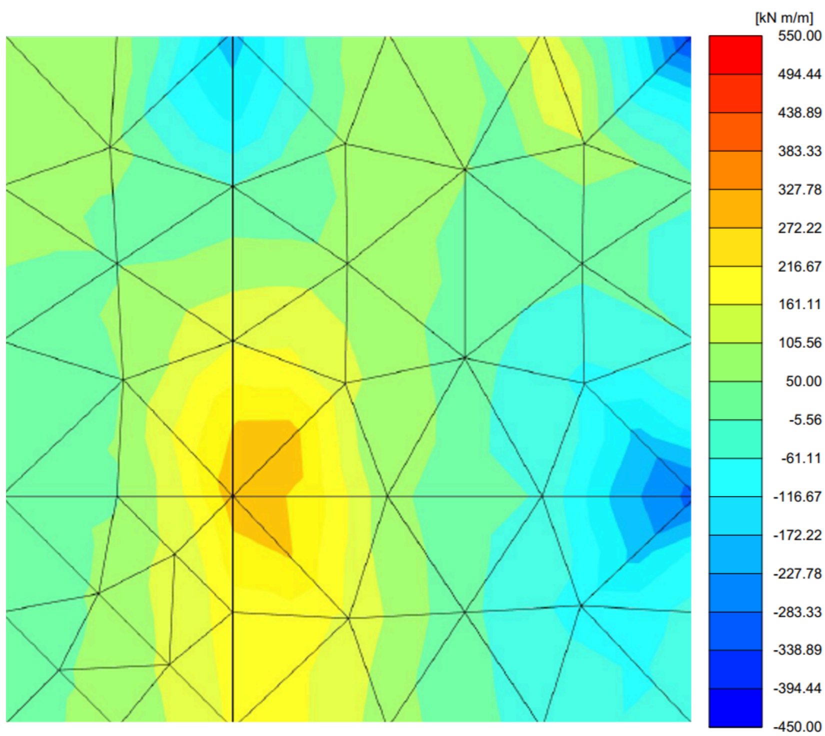


Figure 1–27: Bending moments in the basement floor

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.

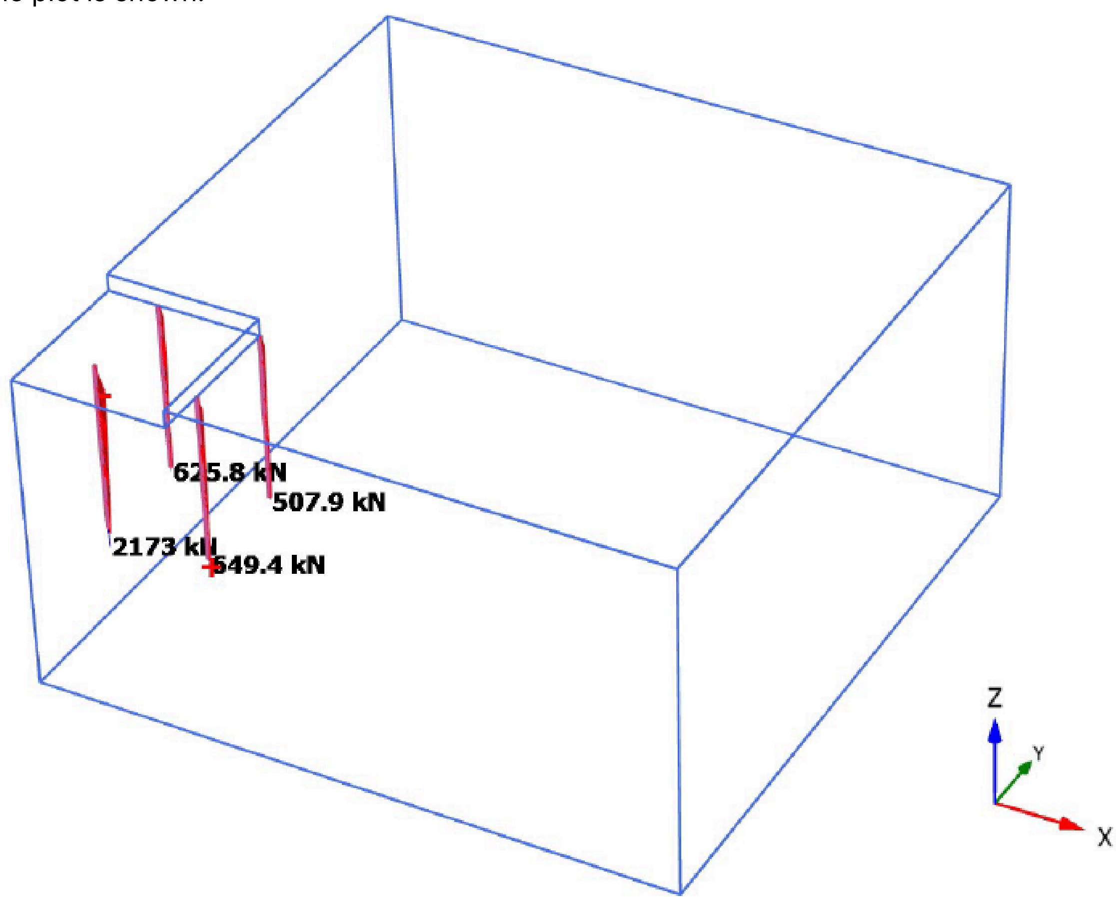
5 Click the **Toggle visibility** button in the side toolbar.

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 1–28: Resulting axial forces (N) in the embedded beams