1 Settlement of a circular footing on sand

1.1 Introduction

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

Objectives:

- Starting a new project
- Creating an axisymmetric model
- Creating soil stratigraphy using the *Borehole* feature
- Creating and assigning of material data sets for soil (*Mohr-Coulomb model*)
- Defining prescribed displacements
- Creation of footing using the *Plate* feature
- Creating and assigning material data sets for plates
- Creating loads
- Generating the mesh

- Generating initial stresses using the K0 procedure
- Defining a *Plastic* calculation
- Activating and modifying the values of loads in calculation phases
- Viewing the calculation results
- Selecting points for curves
- Creating a 'Load - displacement' curve

1.2 Geometry

A circular footing with a radius of 1.0 m is placed on a sand layer of 4.0m thickness as shown in Figure 1–1 (p. 11). Under the sand layer there is a stiff rock layer that extends to a large depth. The purpose of the exercise is to find the displacements and stresses in the soil caused by the load applied to the footing. Calculations are performed for both rigid and flexible footings. The geometry of the finite element model for these two situations is similar. The rock layer is not included in the model; instead, an appropriate boundary condition is applied at the bottom of the sand layer. To enable any possible mechanism in the sand and to avoid any influence of the outer boundary, the model as shown in Figure 1–1 (p. 11) is extended in horizontal direction to a total radius of 5.0 m.

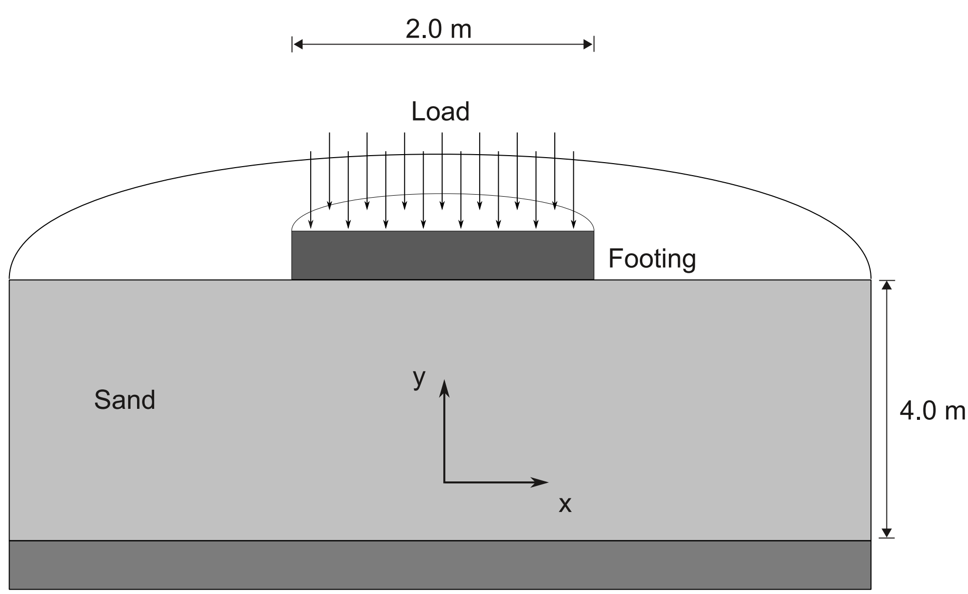

Figure 1–1: Geometry of a circular footing on a sand layer

1.3 Case A: Rigid footing

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

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

1.3.1 Create a new project

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

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

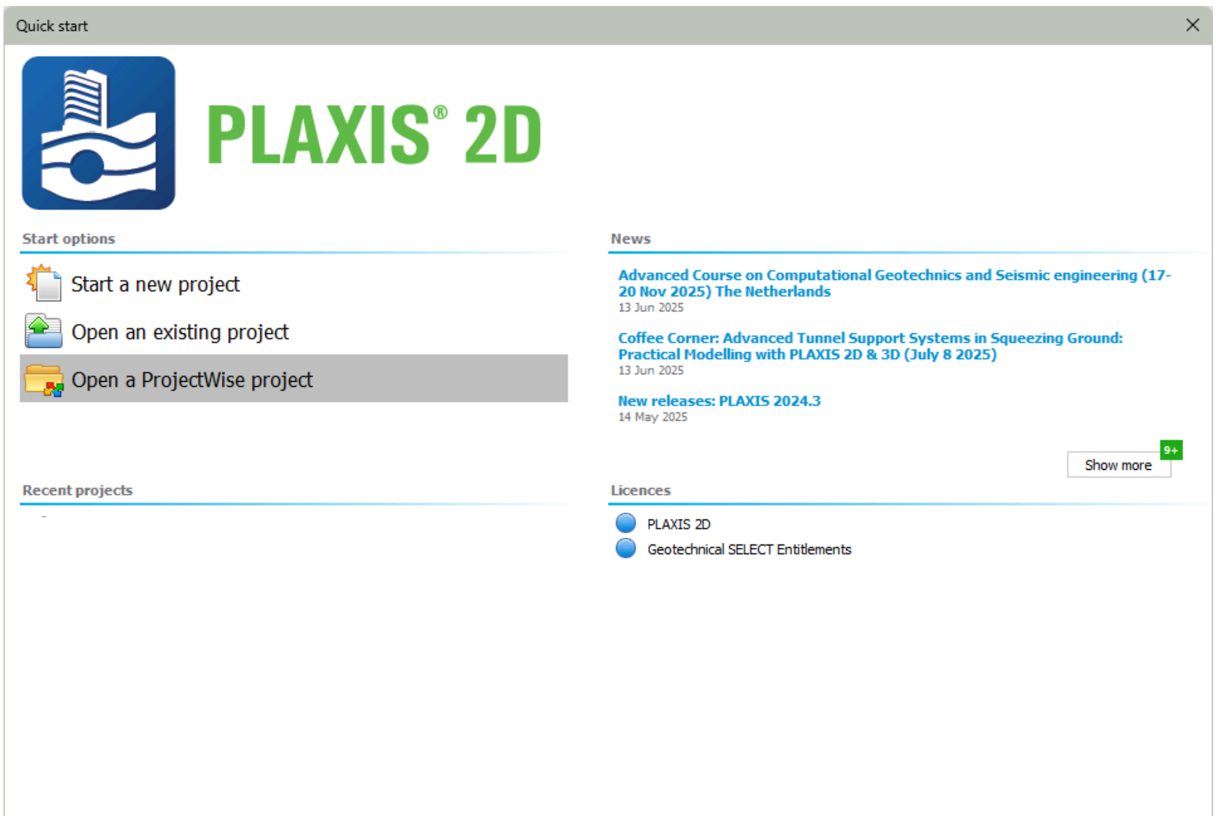

Figure 1–2: Quick start - PLAXIS 2D

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

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

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

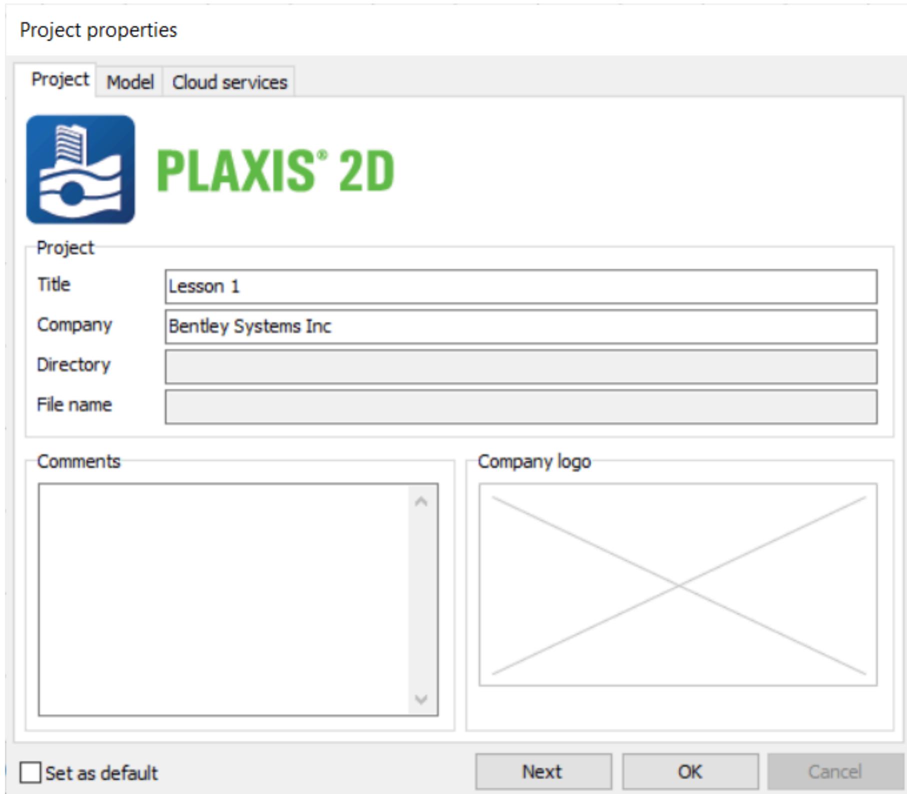

Figure 1–3: Project properties window - PLAXIS 2D

> **Note:**
>
> The first step in every analysis is to set the basic parameters of the finite element model. This is done in the **Project properties** window. These settings include the description of the problem, the type of model, the basic type of elements, the basic units and the size of the drawing area.
>
> To enter the appropriate settings for the footing calculation follow the steps below.

3. In the **Project** tabsheet, enter `Lesson 1` in the **Title** box and type `Settlement of a circular footing` in the **Comments** box.
4. Click the **Next** button at the bottom or click the **Model** tab.

The **Model** properties are shown in Figure 1–4 (p. 14):

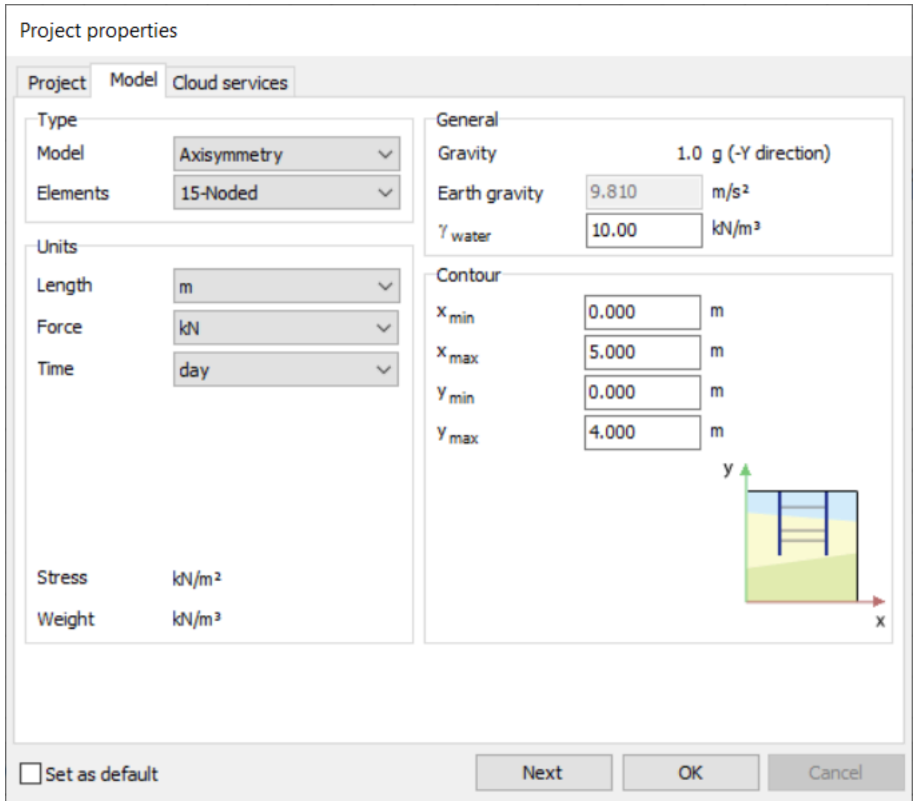

Figure 1–4: Model properties tabsheet

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

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

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

1.3.2 Define the soil stratigraphy

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

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

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

In order to construct the soil stratigraphy follow these steps:

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

   The **Modify soil layers** window will appear as shown in Figure 1–5 (p. 15) .
3. Add a soil layer by clicking the **Add** button in the **Modify soil layers** window.
4. Set the top boundary of the soil layer at y = 4 and keep the bottom boundary at y = 0 m.
5. Set the Head to 2.0 m.

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

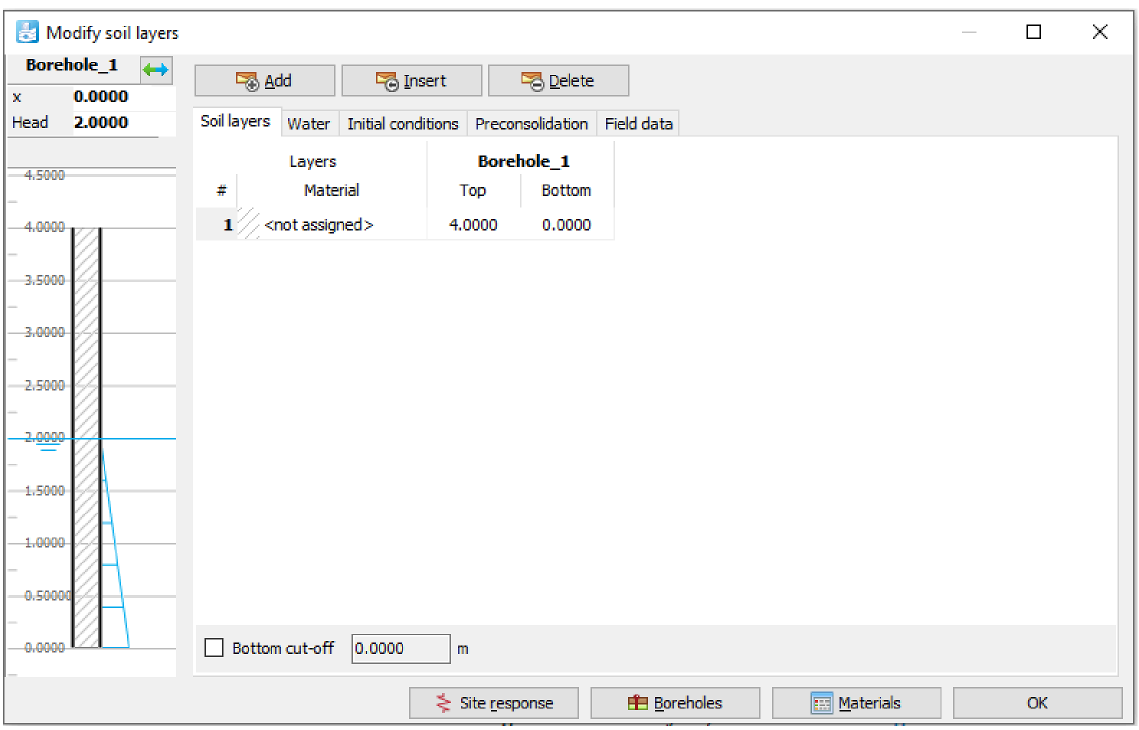

Figure 1–5: Modify soil layers window

Next the material data sets are defined and assigned to the soil layers, see 1.3.3 Create and assign material data sets (p. 15).

1.3.3 Create and assign material data sets

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

The sand layer that is used in this tutorial has the following properties as shown in Table 1–1 (p. 16) :

Table 1–1: Material properties of the sand layer

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

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

1. Open the **Material sets** window by clicking the **Materials** button  in the **Modify soil layers** window or in the side toolbar.

   The **Material sets** window pops up as shown in Figure 1–6 (p. 17).

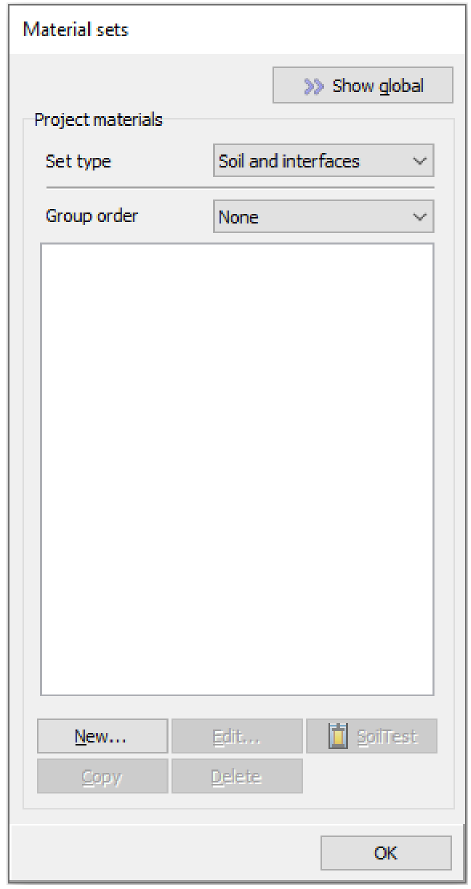

Figure 1–6: Material sets window

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

   A new window will appear with the tabsheets: **General**, **Mechanical**, **Groundwater**, **Interfaces** and **Initial**.

3. In the **Material set** box of the **General** tabsheet, write Sand in the **Identification** box.

   The default material model (Mohr-Coulomb) and drainage type (Drained) are valid for this example.

4. Enter the proper values in the **General properties** box (Figure 1–7 (p. 18)) according to the material properties listed in Table 1–1 (p. 16). Keep parameters that are not mentioned in the table at their default values.

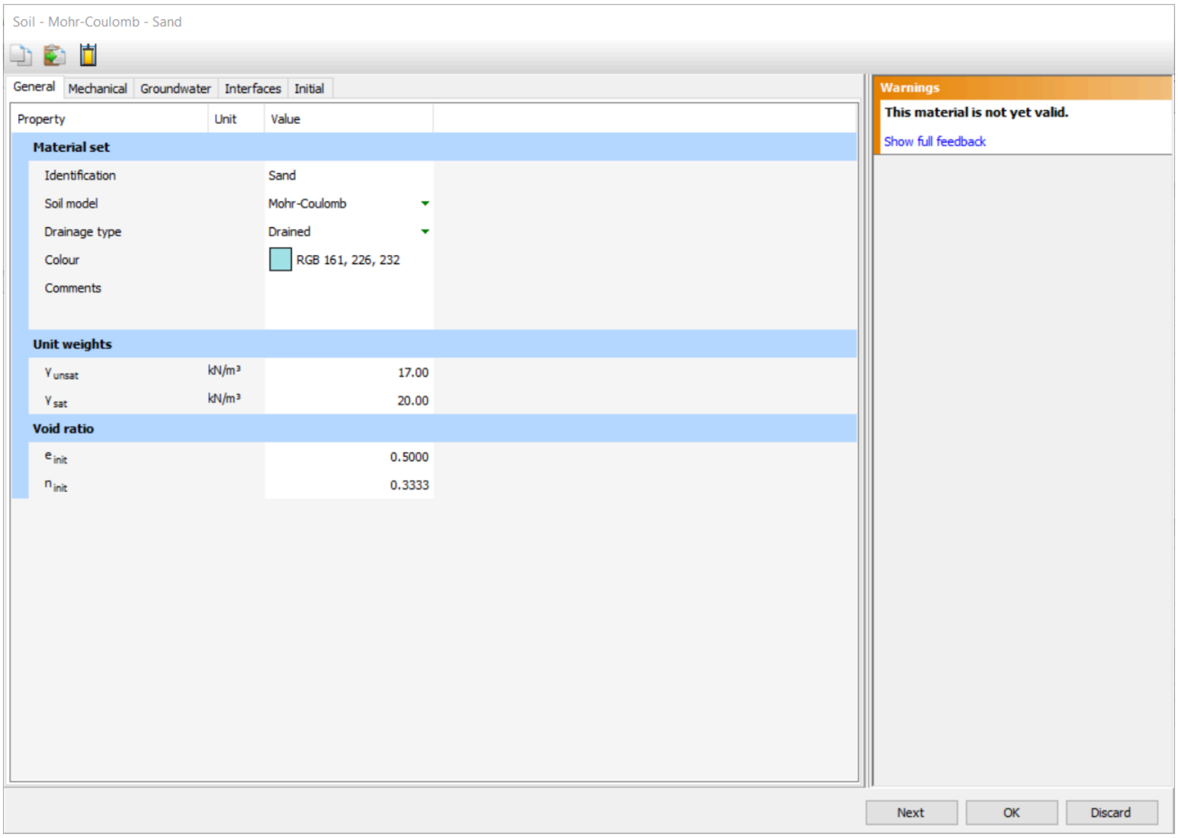

Figure 1–7: The General tabsheet of the Soil window

Note:

a. As displayed in Figure 1–7 (p. 18) a **Feedback side panel** is included in the **Material** window. This panel prevents the definition of an invalid material data set. To display the list of detailed messages please select *Show full feedback*. Three types of messages are possible:

- **Errors:** the parameter value or combination of parameter values must be changed, otherwise the material set could be invalid and calculation of the project will be blocked.
- **Warnings:** the parameter value seems to deviate from a recommended parameter value or parameter range. Generally the material set will not be considered invalid and calculating the project will not be blocked. The chosen parameter could however cause unexpected results.
- **Hints:** the entered parameter can be defined under certain circumstances or options.

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 model parameters.

The parameters appearing on the **Mechanical** tabsheet depend on the selected material model (in this case the Mohr-Coulomb model).

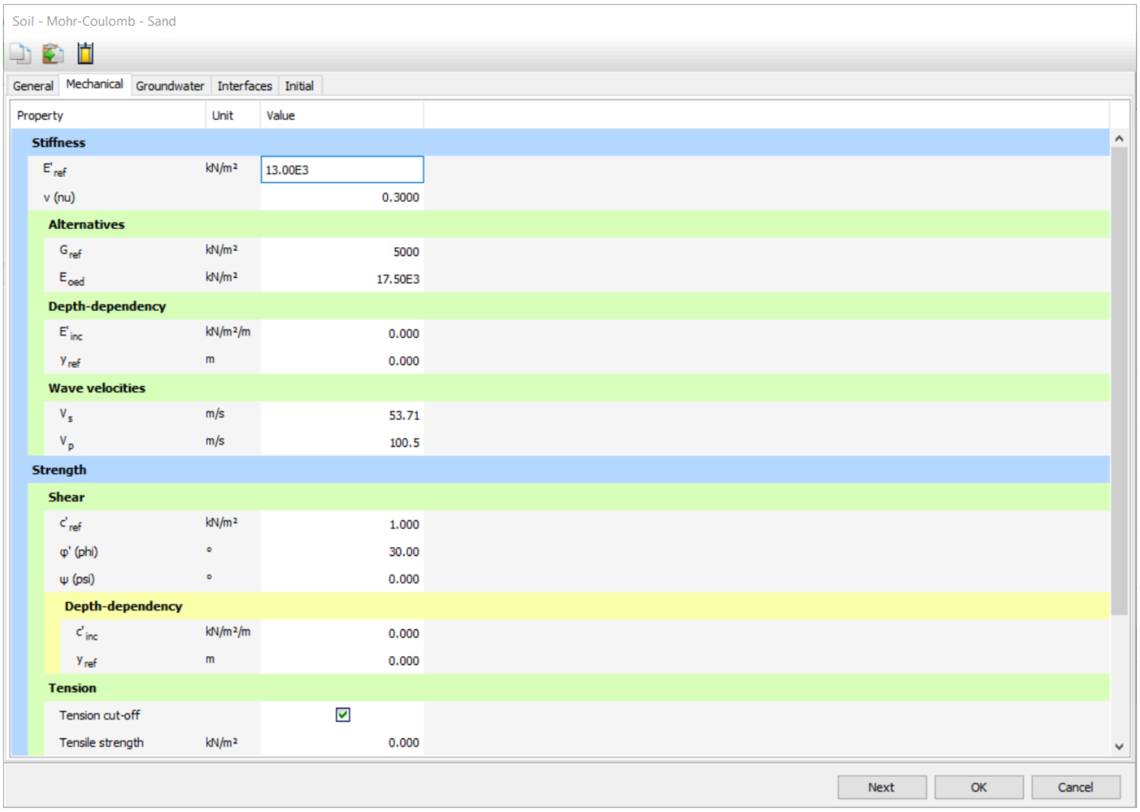

Figure 1–8: Mechanical tabsheet of the Soil window - Soil and interfaces set type

6 Enter the model parameters of Table 1–1 (p. 16) in the corresponding edit boxes of the **Mechanical** tabsheet (Figure 1–8 (p. 19)) and keep the other parameters as their default values. A detailed description of different soil models and their corresponding parameters can be found in the Material Models Manual.

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

7 The soil material is drained, the geometry model does not include interfaces and the default thermal and initial conditions are valid for this case, therefore the remaining tabsheets can be skipped. Click **OK** to confirm the input of the current material data set.

Now the created data set will appear in the tree view of the **Material sets** window.

8 Drag the set **Sand** from the **Material sets** window (select it and hold down the left mouse button while moving) to the graph of the soil column on the left hand side of the **Modify soil layers** window and drop it there (release the left mouse button).

9 Click **OK** in the **Material sets** window to close the database.

10 Click **OK** to close the **Modify soil layers** window.

Tip:

- Existing data sets may be changed by opening the **Material sets** window, selecting the data set to be changed from the tree view and clicking the **Edit** button. As an alternative, the **Material sets** window can be opened by clicking the corresponding button in the side toolbar.
- PLAXIS 2D distinguishes between a project database and a global database of material sets. Data sets may be exchanged from one project to another using the global database. The global database can be shown in the **Material sets** window by clicking the *Show global* button. The data sets of all tutorials in the Tutorial Manual are stored in the global database during the installation of the program.
- The material assigned to a selected entity in the model can be changed in the **Material** drop-down menu in the **Selection explorer**. Note that all the material datasets assignable to the entity are listed in the drop-down menu. However, only the materials listed under **Project materials** are listed, and not the ones listed under **Global materials**.
- The program performs a consistency check on the material parameters and will give a warning message in the case of a detected inconsistency in the data.

1.3.4 Define the footing

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

Tip:

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

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

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .
2. Click the **Create prescribed displacement** button in the side toolbar.
3. Select the **Create line displacement** option in the expanded menu.
4. In the drawing area move the cursor to point (0 4) and click the left mouse button.

5. Move along the upper boundary of the soil to point (1 4) and click the left mouse button again.
6. Click the right mouse button to stop drawing.
7. In the **Selection explorer** set the x-component of the prescribed displacement (Displacement $_x$) to `Fixed`.
8. Specify a uniform prescribed displacement in the vertical direction by assigning a value of -0.05 to $u_{y,start,ref}$, signifying a downward displacement of 0.05 m as shown in Figure 1–9 (p. 21).

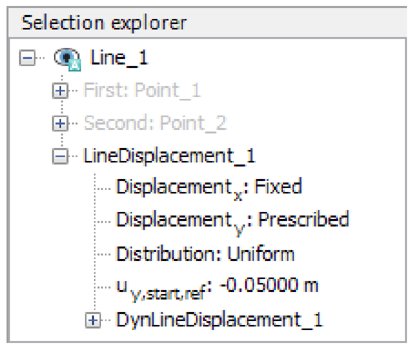

Figure 1–9: Prescribed displacement in the Selection explorer

The geometry of the model is complete.

When the geometry model is complete, the finite element mesh can be generated. Proceed to 1.3.5 Generate the mesh (p. 21).

1.3.5 Generate the mesh

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

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

In order to generate the mesh, follow these steps:

1. Proceed to the **Mesh mode** by clicking the corresponding tab.
2. Click the **Generate mesh** button in the side toolbar.

   The **Mesh options** window pops up as shown in Figure 1–10 (p. 22). The **Medium** option is by default selected as element distribution.

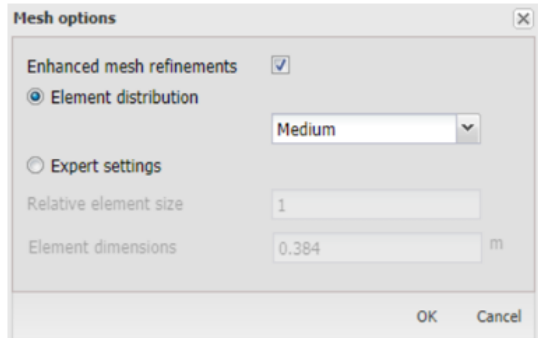

Figure 1–10: The Mesh options window

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

   A new window is opened displaying the generated mesh as shown in Figure 1–11 (p. 22). Note that the mesh is automatically refined under the footing.

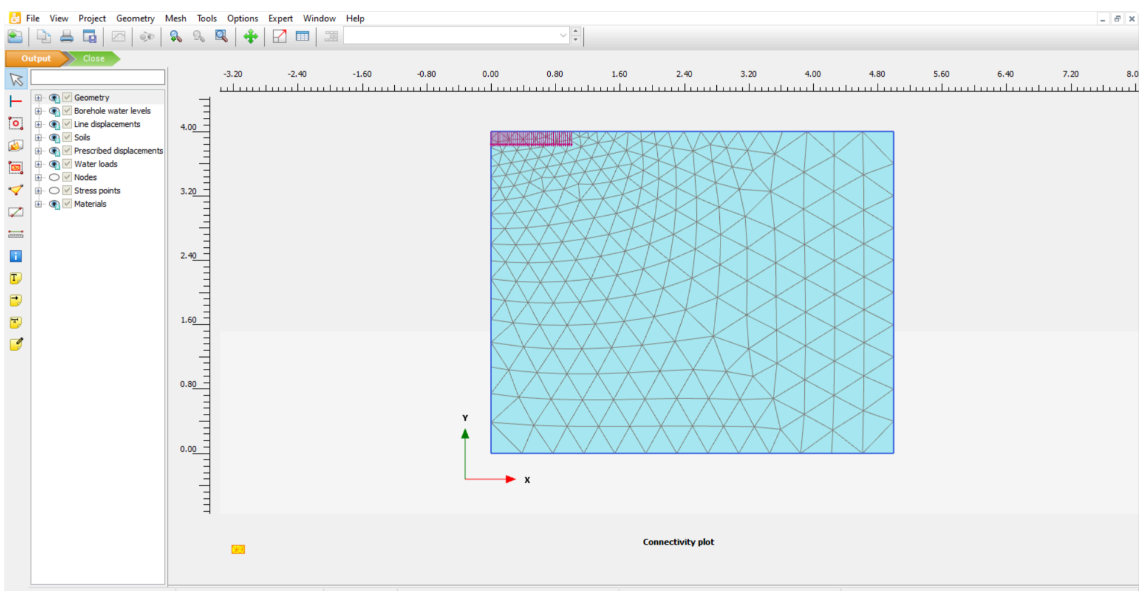

Figure 1–11: The generated mesh in the Output window

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

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

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

After the mesh was generated, the calculation phases are defined and the calculation is done, see 1.3.6.1 Initial phase (p. 23) for instructions.

1.3.6 Define and perform the calculation

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

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.

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

   When a new project has been defined, a first calculation phase named ' Initial phase', is automatically created and selected in the **Phases explorer** as shown in Figure 1–12 (p. 23):

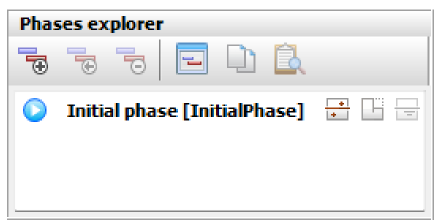

Figure 1–12: Phases explorer - Initial Phase

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

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

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

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

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

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

The **Phases** window is displayed in Figure 1–13 (p. 24).

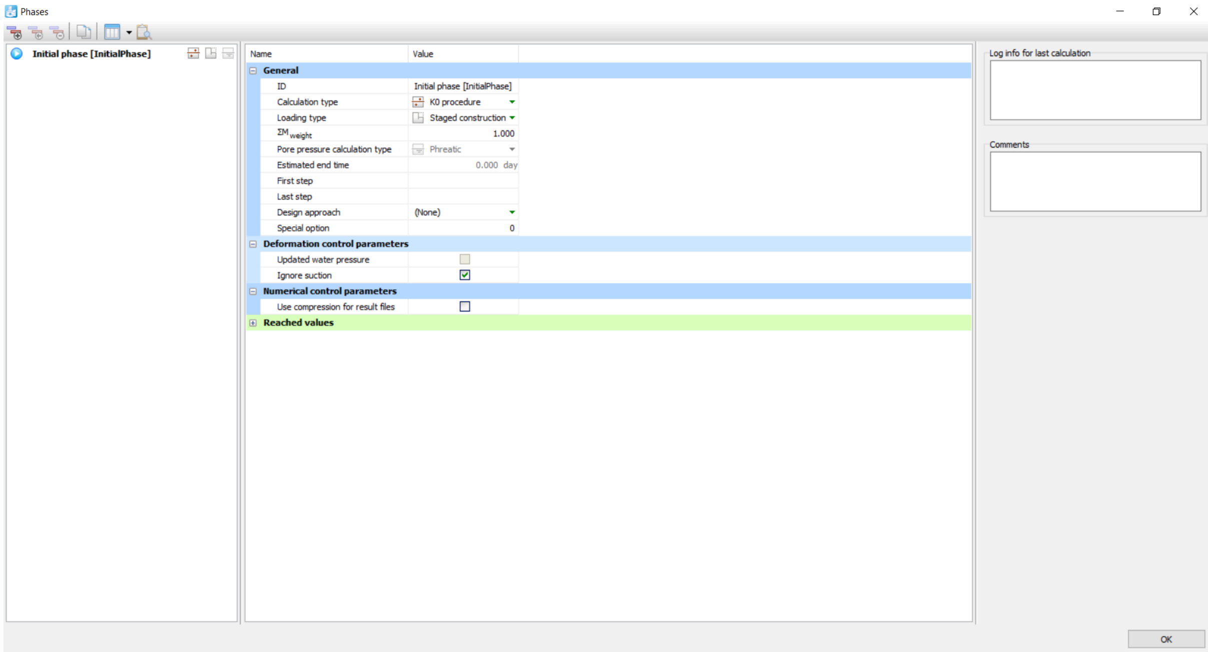

Figure 1–13: Phases window - Initial phase

3. Click **OK** to close the **Phases** window.
4. In the **Model explorer** expand the **Model conditions** subtree as shown in Figure 1–14 (p. 25).

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

To avoid the situation where the displacements of the geometry are undetermined, some points of the geometry must have prescribed displacements. The simplest form of a prescribed displacement is a fixity (zero displacement), but non-zero prescribed displacements may also be given.

5. Expand the **Deformations** subtree.

Note that the box is checked by default. By default, a full fixity is generated at the base of the geometry, whereas roller supports are assigned to the vertical boundaries (**BoundaryXMin** and **BoundaryXMax** are normally fixed, **BoundaryYMin** is fully fixed and **BoundaryYMax** is free).

6. Expand the **Water** subtree.

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

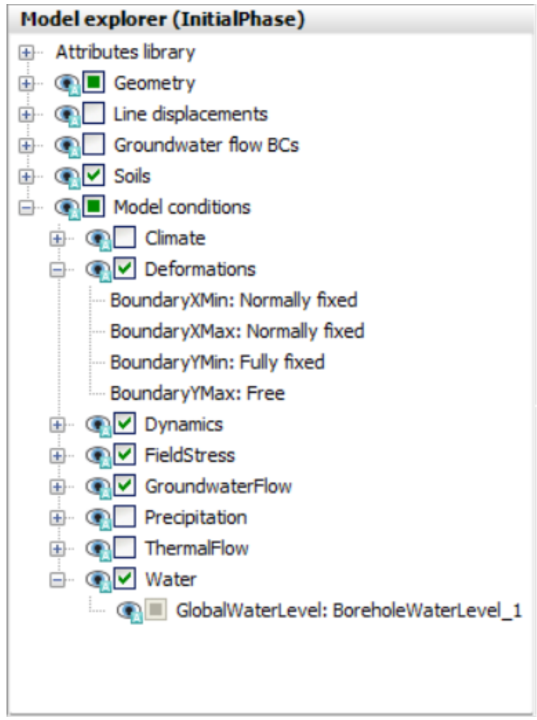

Figure 1–14: Model explorer showing model conditions and Deformations

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

The model of the project in the initial phase is shown in Figure 1–15 (p. 25).

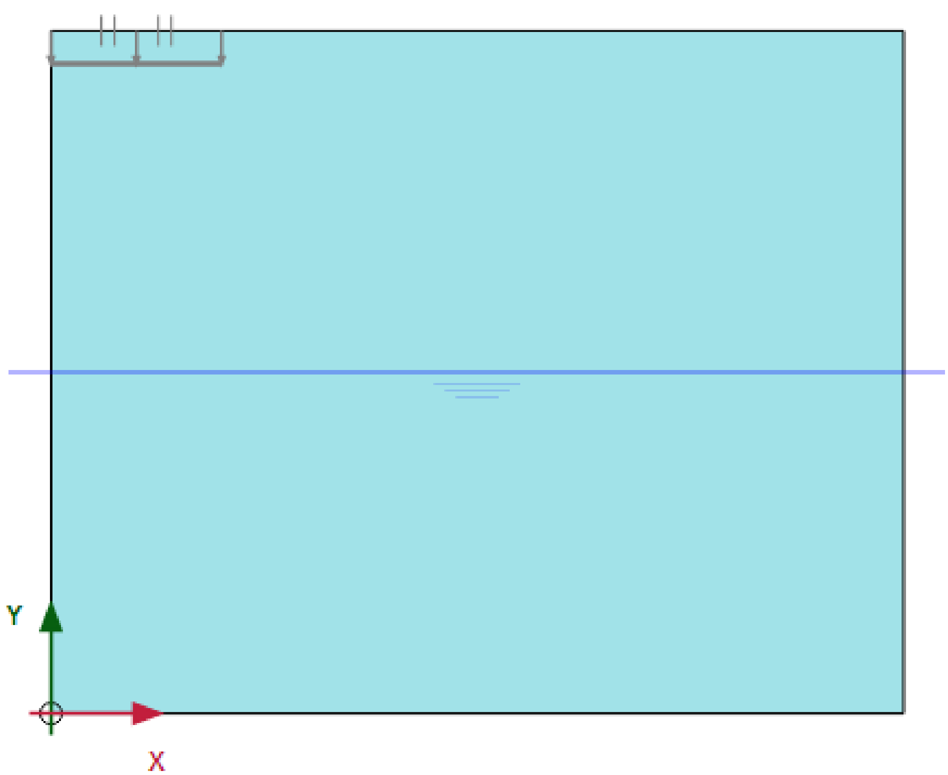

Figure 1–15: Initial phase in the Staged construction mode

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

1.3.6.2 | Phase 1: Footing

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

1. Click the **Add phase** button in the **Phases explorer**.

   A new phase, named `Phase_1` will be added in the **Phases explorer**.

2. Double-click **Phase_1** to open the **Phases** window. In the **ID** box of the **General** section, write (optionally) an appropriate name for the new phase (for example `Indentation`).

   The current phase starts from the **Initial phase**, which contains the initial stress state. The default options and values assigned are valid for this phase as shown in Figure 1–16 (p. 26).

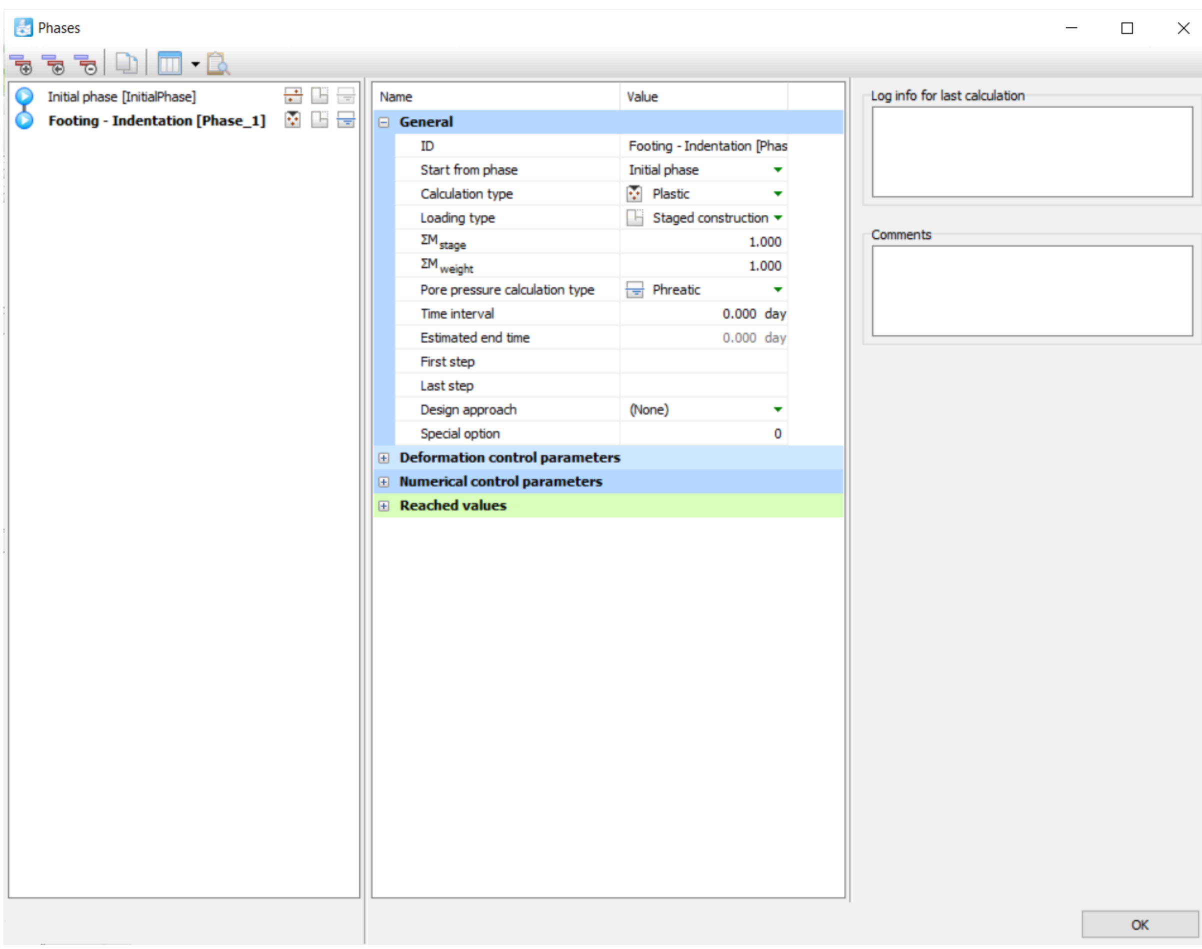

Figure 1–16: The Phases window for the Indentation phase

3. Click **OK** to close the **Phases** window.

4. Click the **Staged construction** tab to enter the corresponding mode.

5 In the drawing area right-click the prescribed displacement and select the **Activate** option in the appearing menu as shown in Figure 1–17 (p. 27).

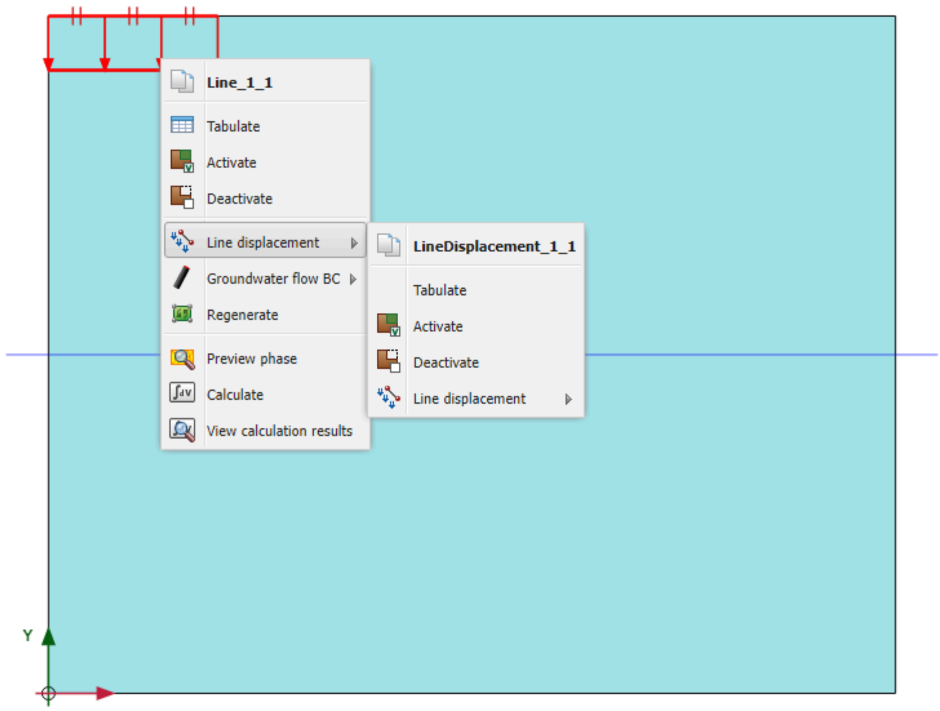

Figure 1–17: Activation of the prescribed displacement in the Staged construction mode

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

1.3.6.3 Execute the calculation

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

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

During the execution of a calculation, a window appears which gives information about the progress of the actual calculation phase as shown in Figure 1–18 (p. 28) .

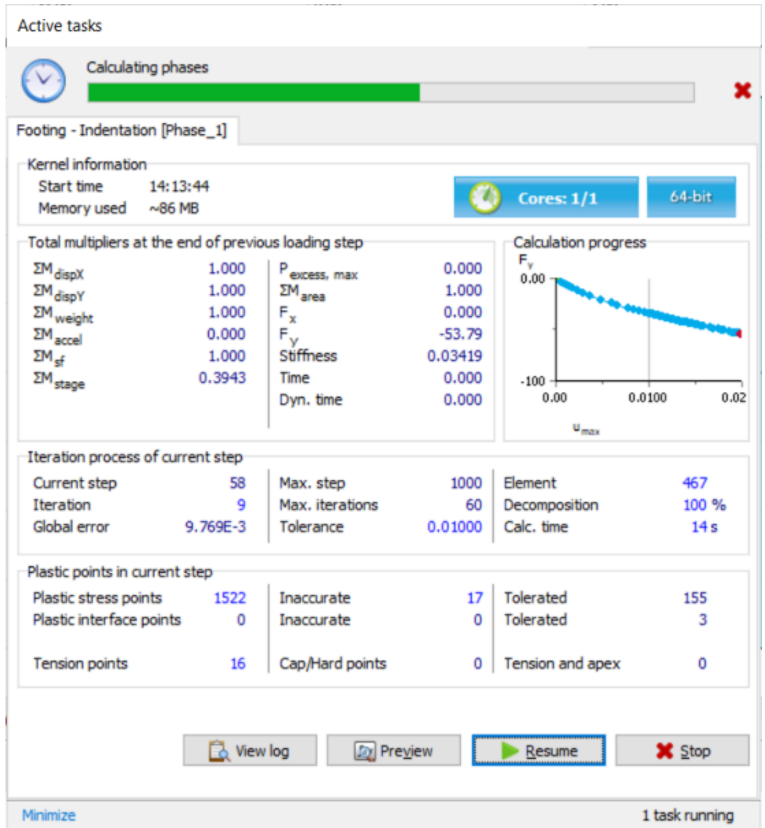

Figure 1–18: Calculation progress

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

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

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

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

1.3.6.4 View the calculation results

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

1 Open the **Phases** window.

2 From the **Reached values** subtree look for the **Force-Y** which is an important value of the current application. This value represents the total reaction force corresponding to the applied prescribed vertical displacement, which corresponds to the total force under 1.0 radian of the footing (note that the analysis is axisymmetric). In order to obtain the total footing force, the value of **Force-Y** should be multiplied by 2π (this gives a value of about 588 kN).

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

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

3. Select the last calculation phase in the **Phases** explorer.

4. Click the **View calculation results** button in the side toolbar.

   As a result, the **Output** program is started, showing the deformed mesh at the end of the selected calculation phase as shown in Figure 1–19 (p. 29):

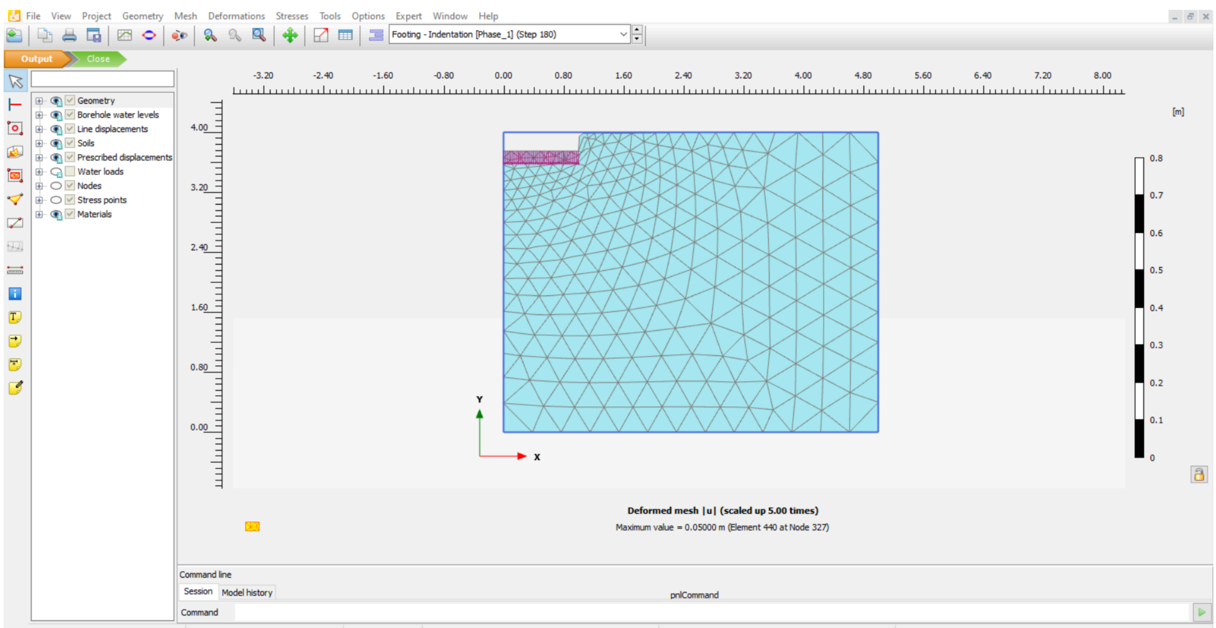

Figure 1–19: Deformed mesh

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

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

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

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

6. The total displacement distribution can be displayed in contours by clicking the Contour lines button in the toolbar.

   The plot shows contour lines of the total displacements, which are labelled. An index is presented with the displacement values corresponding to the labels.

7. Click the **Arrows** button .

The plot shows the total displacements of all nodes as arrows, with an indication of their relative magnitude.

8 Click the menu **Stresses** > **Principal effective stresses** > **Effective principal stresses**.

The plot shows the effective principal stresses at the stress points of each soil element with an indication of their direction and their relative magnitude as shown in Figure 1–20 (p. 30):

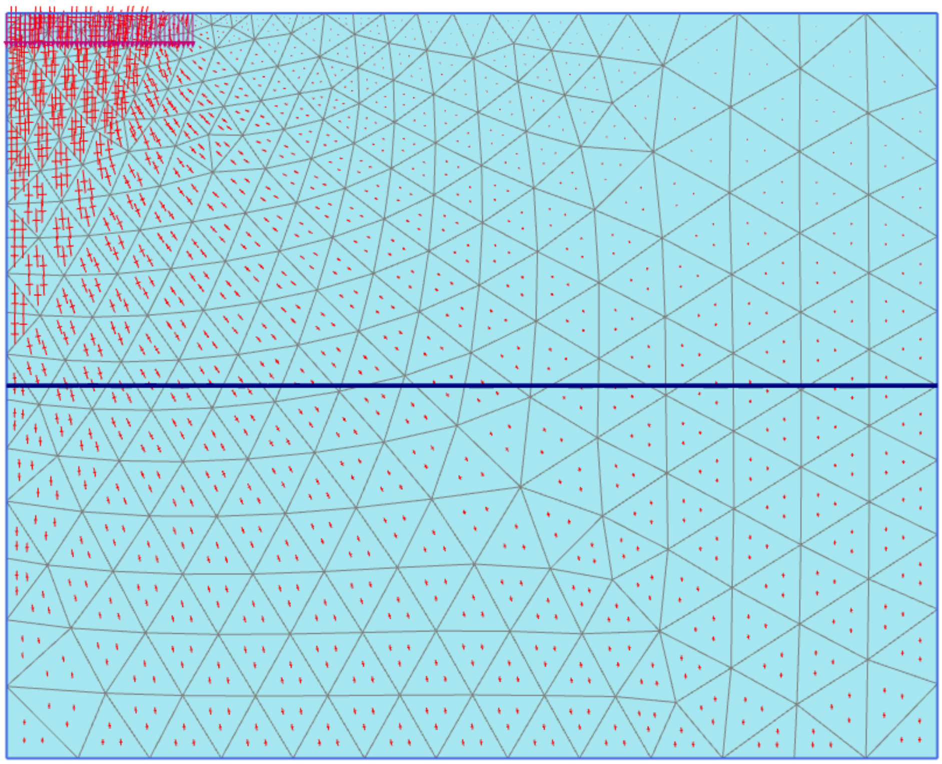

Figure 1–20: Effective principal stresses

9 Click the **Table** button on the toolbar.

A new window is opened in which a table is presented, showing the values of the principal stresses and other stress measures in each stress point of all elements.

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

1.4 Case B: Flexible footing

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

1.4.1 Modify the geometry

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

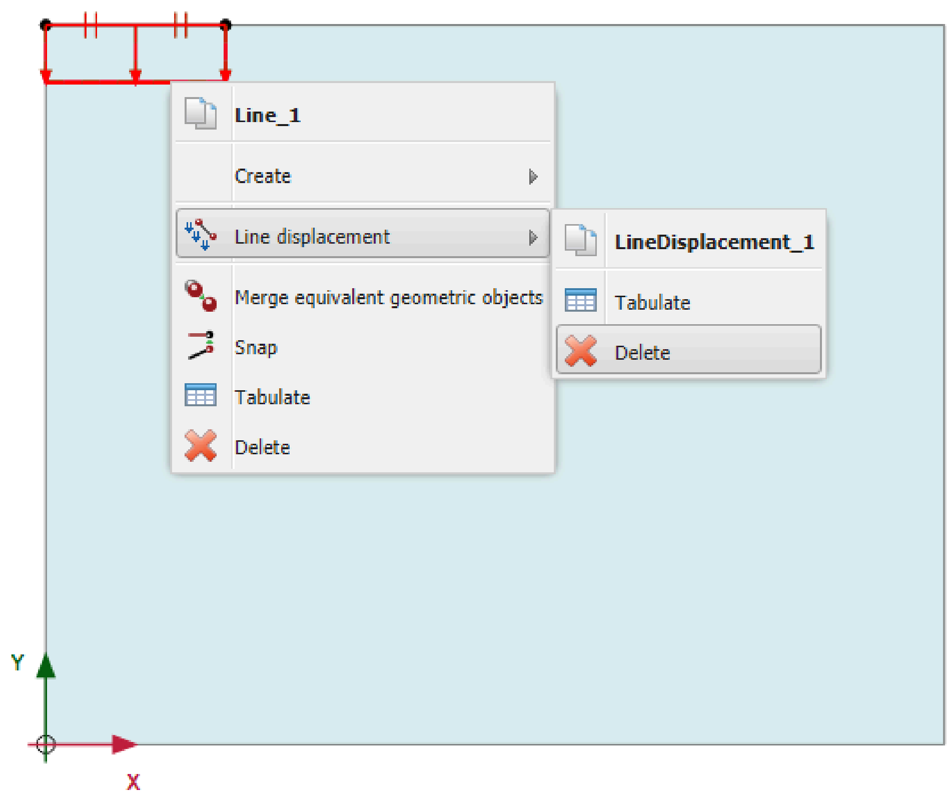

Figure 1–21: Delete the line displacement option

4. In the model right-click the line at the location of the footing. Select **Create** > **Create Plate** as shown in Figure 1–22 (p. 32).

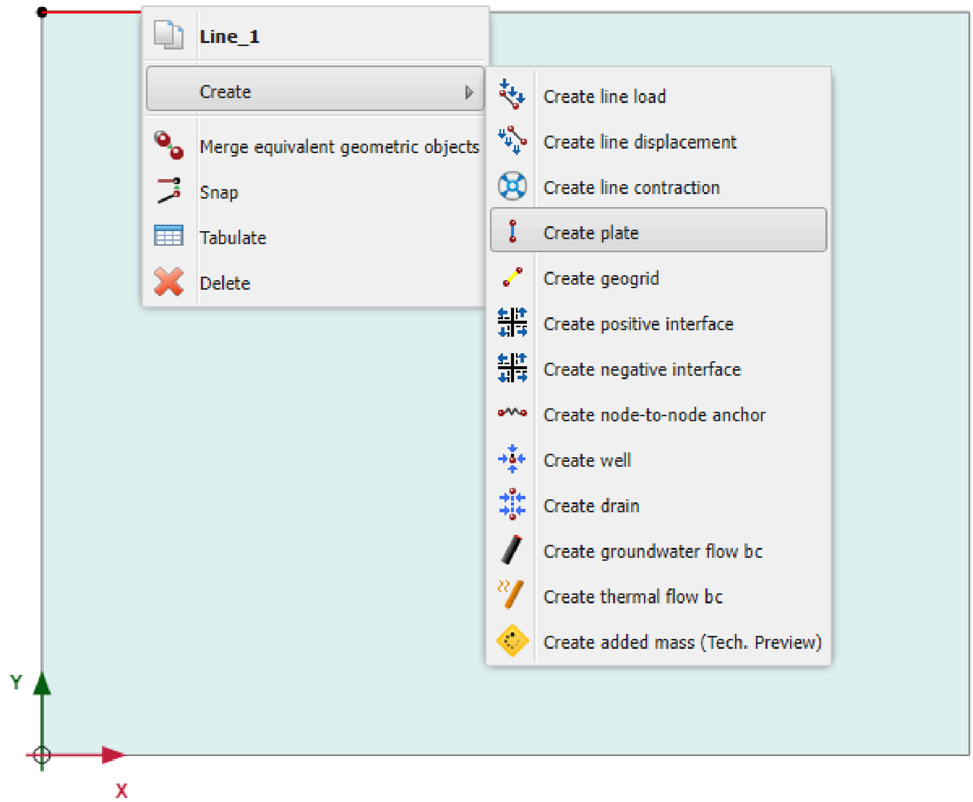

Figure 1–22: Create Plate option

A plate is created, which simulates the flexible footing.

5 In the model right-click again the line at the location of the footing and select **Create** > **Create Line load** as shown in [Figure 1–23 (p. 33)](#).

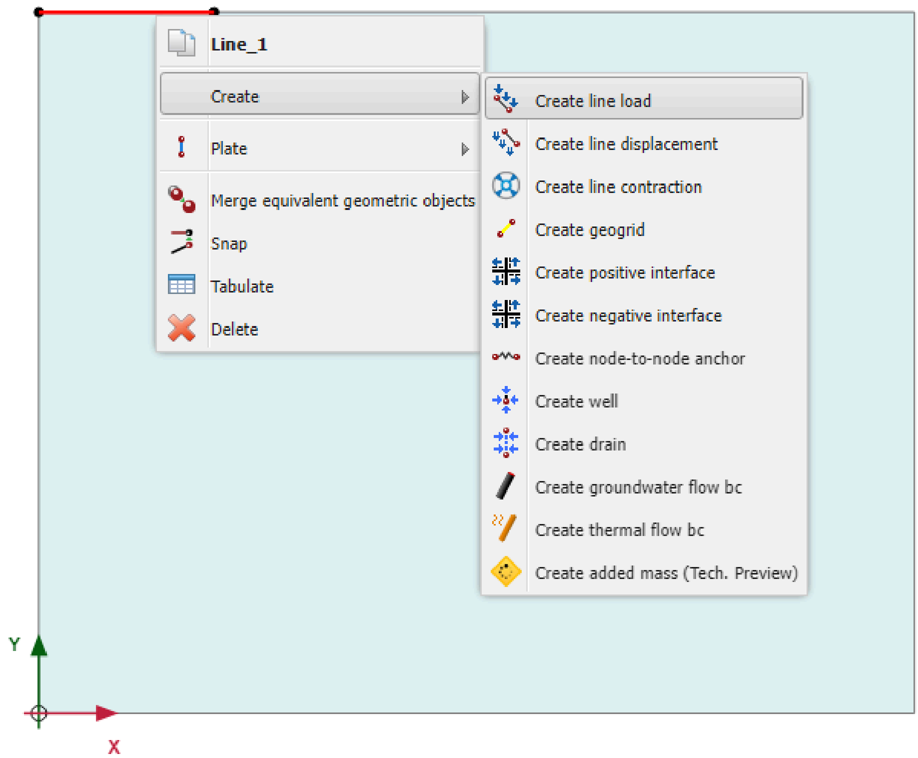

Figure 1–23: Create line load option

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

1.4.2 Add material properties for the footing

The material properties for the flexible footing are as follows:

Table 1–2: Material properties of the footing

Parameter	Name	Value	Unit
General			
Material type	-	Elastic	-
Unit weight	*w*	0.0	kN/m/m
Prevent punching	-	No	
Mechanical			
Isotropic	-	Yes	-
Axial stiffness	EA_1	$5 \cdot 10^{6}$	kN/m
Bending stiffness	EI	$8.5 \cdot 10^{3}$	kNm^2/m
Poisson's ratio	ν	0.0	-

1. Click the **Materials** button in the side toolbar.
2. In the **Material sets** window, from the **Set type** drop-down menu, select **Plates**.
3. Click the **New** button.

   A new window appears where the properties of the footing can be entered.
4. Type `Footing` in the **Identification** box. The **Elastic** option is selected by default for the material type. Keep this option for this example.
5. Enter the properties as listed in Table 1–2 (p. 33). Keep parameters that are not mentioned in the table at their default values.
6. > **Note:** The equivalent thickness is automatically calculated by PLAXIS 2D from the values of EA and EI. It cannot be defined manually.

   Click **OK**.

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

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

1.4.3 Generate the mesh

In order to generate the mesh, follow these steps:

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

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

1.4.4 Calculations

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

The model is shown- in Figure 1–24 (p. 35):

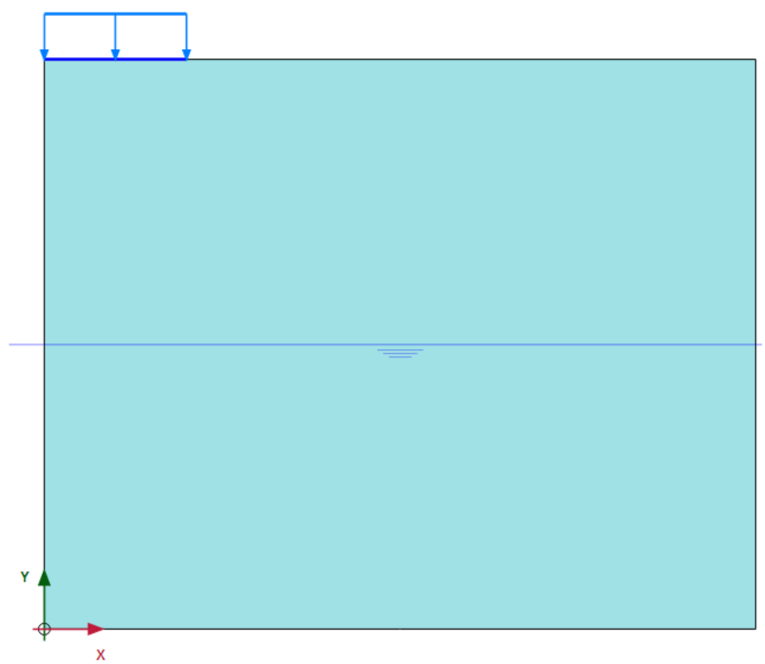

Figure 1–24: Active plate and load in the model

6. In the **Selection explorer** shown in Figure 1–25 (p. 35) assign -188 kN/m^2 to the vertical component of the line load. Note that, this gives a total load that is approximately equal to the footing force that was obtained from the first part of this tutorial. (188 kN/m^2 · π ·(1.0 m)^2 ≈ 590 kN).

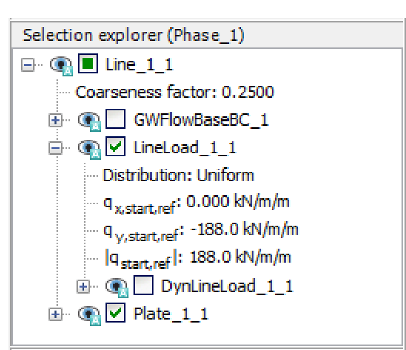

Figure 1–25: Definition of the load components in the Selection explorer

7 No changes are required in the **Flow conditions** tabsheet.

The calculation definition is now complete. Before starting the calculation it is advisable to select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow these steps:

8 Click the **Select points for curves** button in the side toolbar.

As a result, all the nodes and stress points are displayed in the model in the Output program. The points can be selected either by directly clicking on them or by using the options available in the **Select points** window.

9 In the **Select points** window enter (0.0 4.0) for the coordinates of the point of interest and click **Search closest**.

The nodes and stress points located near that specific location are listed.

10 Select the node at exactly (0.0 4.0) by checking the box in front of it. The selected node is indicated by **Node 4*** in the model when the **Selection labels** option is selected in the **Mesh** menu.

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

11 Click the **Update** button on the top left to return to the Input program.

12 Check if both calculation phases are marked for calculation by a blue arrow. If this is not the case click the symbol of the calculation phase or right-click and select **Mark for calculation** from the pop-up menu.

13 Click the **Calculate** button to start the calculation.

14 Click the **Save** button to save the project after the calculation has finished.

1.4.5 View the calculation results

1. After the calculation the results of the final calculation step can be viewed by clicking the **View calculation results** button. Select the plots that are of interest. The displacements and stresses should be similar to those obtained from the first part of the exercise.
2. Click the **Select structures** button in the side toolbar and double-click the footing.

   A new window opens in which either the displacements or the bending moments of the footing may be plotted (depending on the type of plot in the first window).
3. Note that the menu has changed. Select the various options from the **Forces** menu to view the forces in the footing.

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

1.4.6 Generate a load-displacement curve

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

1. Click the **Curves manager** button in the toolbar.

   The **Curves manager** window pops up.
2. In the **Charts** tabsheet, click **New**.

   The **Curve generation** window pops up as shown in Figure 1–26 (p. 38).

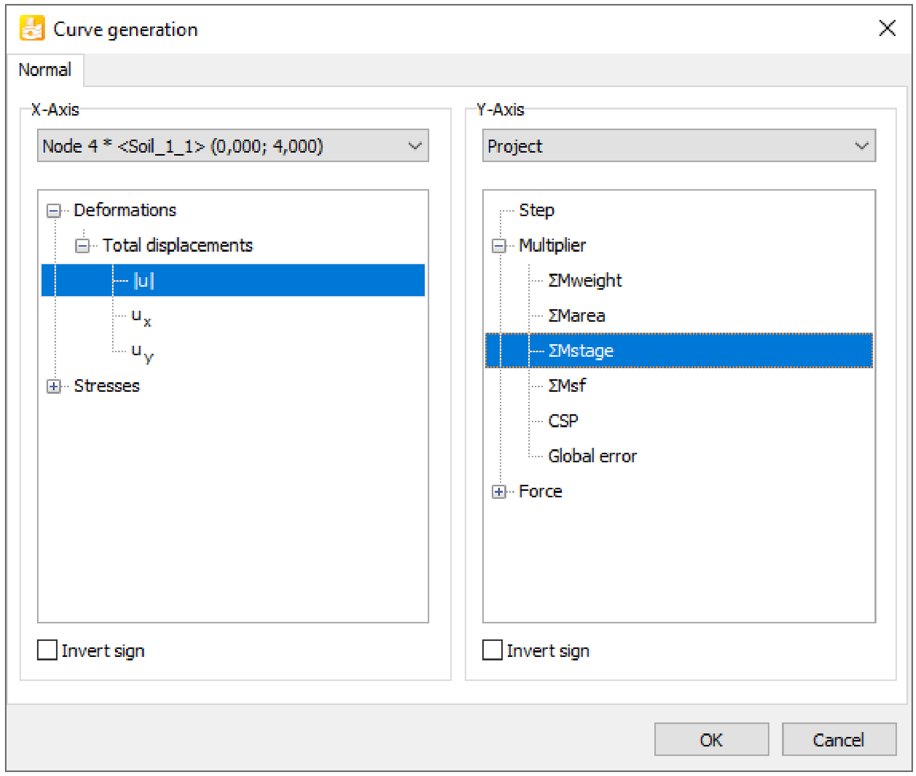

Figure 1–26: Curve generation window

3. For the x-axis, select **Node 4* (0.00 , 4.00)** from the drop-down menu. Select the **Deformations** > **Total displacements** > **|u|**.
4. For the y-axis, select the Project option from the drop-down menu. Select the **Multipliers** > **ΣMstage** option. ΣMstage is the proportion of the specified changes that has been applied. Hence the value will range from 0 to 1, which means that 100% of the prescribed load has been applied and the prescribed ultimate state has been fully reached.
5. Click **OK** to accept the input and generate the load-displacement curve.

As a result the curve of is plotted as shown in Figure 1–27 (p. 39):

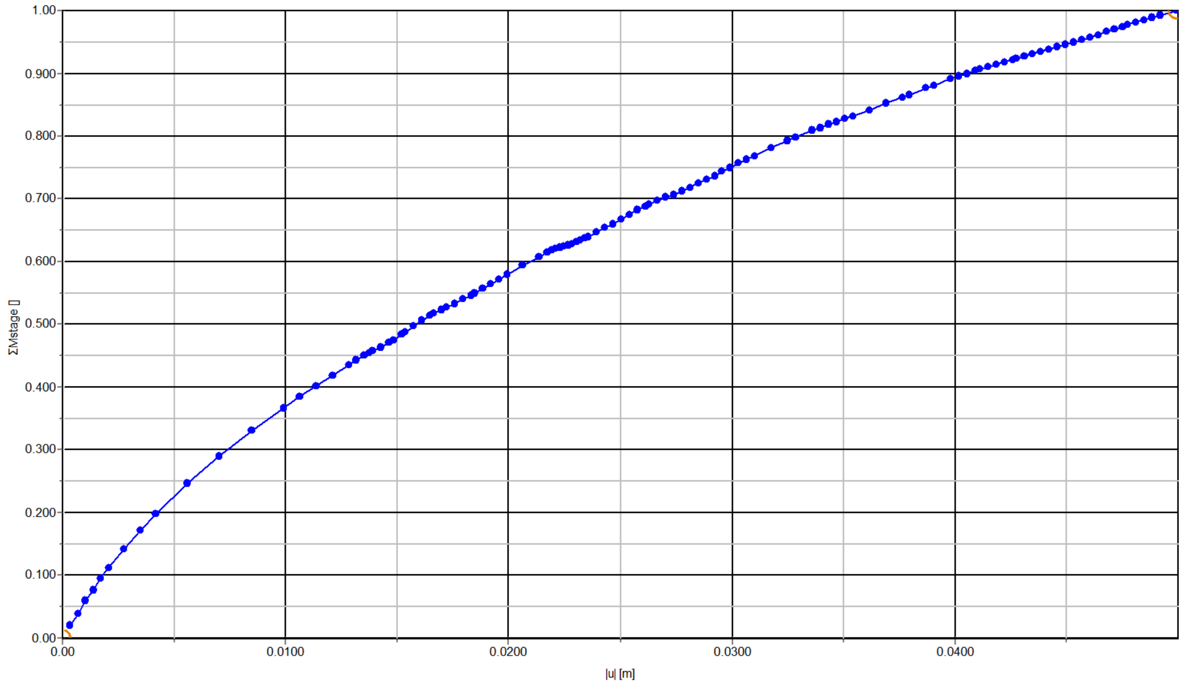

Figure 1–27: Load-displacement curve for the footing

Tip:

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

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

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