3
Submerged construction of an excavation

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

Objectives

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

Geometry

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

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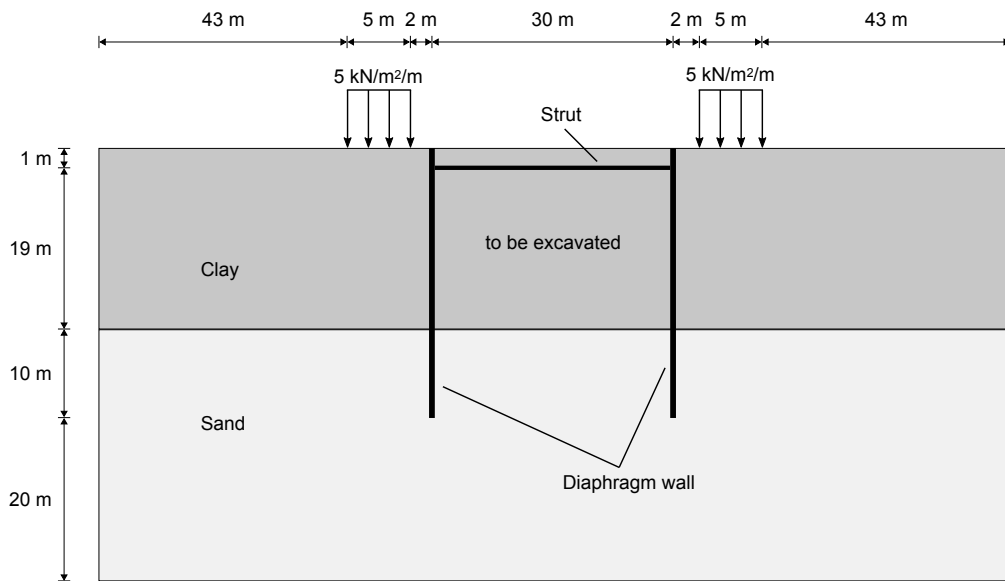
Figure 42: Geometry model of the situation of a submerged excavation

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

3.1 Create new project

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

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

3. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.

4. In the **Model** tabsheet keep the default options for **Model** (Plane strain}, and **Elements** (15-Node).

5. Set the model **Contour** to x_{min} = 0 m, x_{max} = 65 m, y_{min} = -30 m and y_{max} = 20 m.

6. Keep the default values for units and constants and click **OK**.

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

3.2 Define the soil stratigraphy

To define the soil stratigraphy:

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

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

3.3 Create and assign material data sets

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

The layers have the following properties as shown in :

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

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

Submerged construction of an excavation

Create and assign material data sets

Mechanical				
Unloading / reloading stiffness	$E_{ur}{}^{ref}$	$12 \cdot 10^3$	$120 \cdot 10^3$	kN/m^2
Poisson's ratio	ν_{ur}	0.15	0.2	-
Power for stress-level dependency of stiffness	m	1.0	0.5	-
Cohesion (constant)	c'_{ref}	1	0	kN/m^2
Friction angle	φ'	25	32	°
Dilatancy angle	ψ	0	2	°
K_0-value for normal consolidation	$K_0{}^{nc}$	0.5774	0.4701	-

Groundwater				
Data set	-	Standard	Standard	-
Soil type	-	Coarse	Coarse	-
Use defaults	-	None	None	-
Permeability in horizontal direction	k_x	$1 \cdot 10^{-3}$	1	m/day
Permeability in vertical direction	k_y	$1 \cdot 10^{-3}$	1	m/day

Interfaces				
Srength determination	-	Manual	Manual	-
Strength reduction factor	R_{inter}	0.5	0.67	-

Initial				
K_0 determination	-	Automatic	Automatic	-
Pre-overburden pressure	POP	5	0	kN/m^2
Over-consolidation ratio	OCR	1	1	-

To create the material sets, follow these steps:

1. Click the **Materials** button [icon] in the **Modify soil layers** window.
The **Material sets** window pops up, where the **Soil and interfaces** option is selected by default as the **Set type**.

2. Click the **New** button in the **Material sets** window to create a new data set.

3. For the clay layer, enter Clay for the **Identification** and select Hardening Soil as the **Soil model**. Set the **Drainage type** to Undrained (A).

4. Enter the properties of the clay layer, as listed in Table 4 (on page 54), in the **General**, **Mechanical** and **Flow parameters** tabsheets.

5. Click the **Interfaces** tab. Select the **Manual** option in the **Strength** drop-down menu. Enter a value of 0.5 for the parameter R_{inter}.

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

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

6. In the **Initial** tabsheet keep the default option for the K_0 determination and the default value for the overconsolidation ratio (OCR). Set the pre-overburden pressure (POP) value to 5.0.

7. For the sand layer, enter Sand for the **Identification** and again select Hardening soil as the **Soil model**. The drainage type should be set to Drained.

8. Enter the properties of the sand layer, as listed in Table 4 (on page 54), in the corresponding edit boxes of the **General** and **Mechanical** tabsheet.

9. Click the **Interfaces** tab. In the **Strength** box, select the **Manual** option. Enter a value of 0.67 for the parameter R_{inter}. Close the data set.

10. Assign the material datasets to the corresponding soil layers.

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

3.4 Define the structural elements

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

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

3.4.1 To define the diaphragm wall:

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

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

Property	Name	Value	Unit
General			
Material type	-	Elastic	-
Weight	w	10	kN/m/m
Prevent punching	-	No	-
Mechanical			
Isotropic	-	Yes	-
Axial stiffness	EA_1	$7.5 \cdot 10^6$	kN/m
Bending stiffness	EI	$1.0 \cdot 10^6$	kNm^2/m
Poisson's ratio	v	0.0	-

1. Click the **Create structure** button in the side toolbar.
2. In the expanded menu select the **Create plate** as shown in Figure 43 (on page 58).

Figure 43: Create plate option

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

4. Click the **Show materials** button in the side toolbar. Set the **Set type** parameter in the **Material sets** window to **Plates** and click the **New** button. Enter `Diaphragm wall` as an **Identification** of the data set and enter the properties as given in Table 5 (on page 57).

5. Click **OK** to close the data set.

6. Drag the **Diaphragm wall** data set to the wall in the geometry and drop it as soon as the cursor indicates that dropping is possible.

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

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

3.4.2 To define the interfaces:

1. Right-click on the plate representing the diaphragm wall.
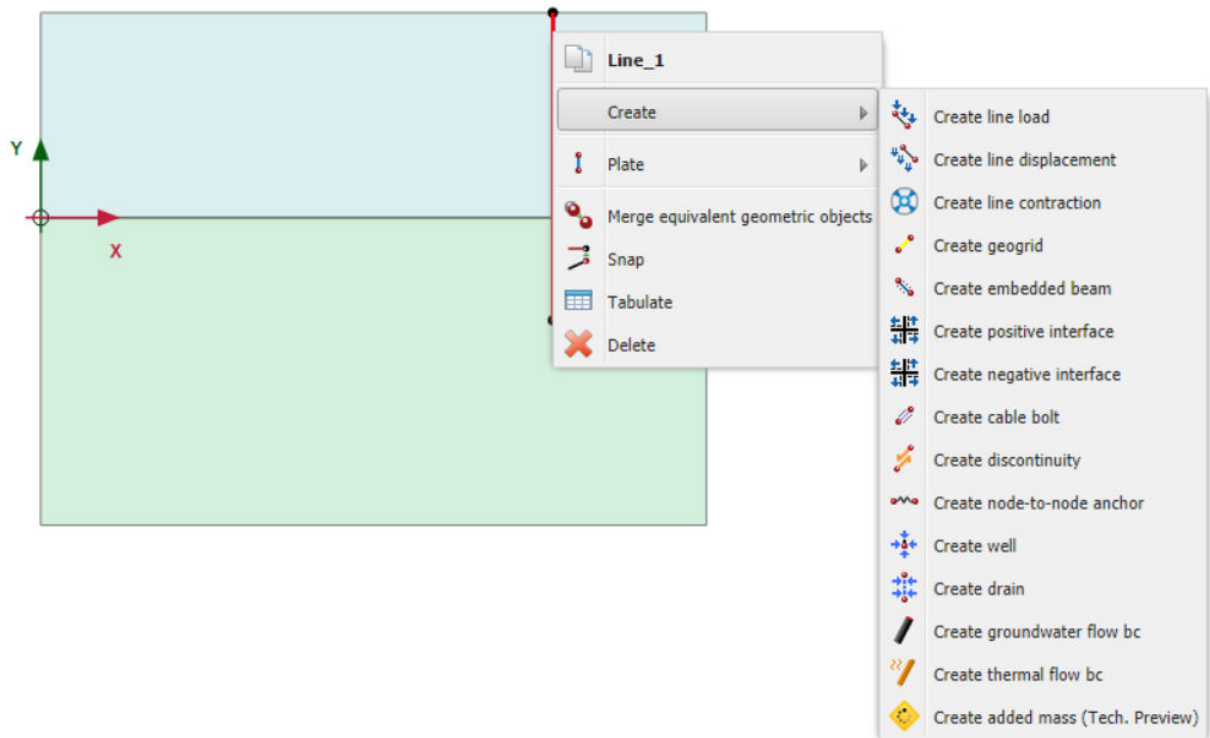
Figure 44: Create positive interface

2. Select **Create** > **Positive interface**.
3. In the same way assign a negative interface as well as shown in Figure 44 (on page 59).

> **Note:**
>
> - In order to identify interfaces at either side of a geometry line, a positive sign ($\oplus$) or negative sign ($\ominus$) is added. This sign has no physical relevance or influence on the results.
> - A **Virtual thickness factor** can be defined for interfaces. This is a purely numerical value, which can be used to optimise the numerical performance of the interface. To define it, select the interface in the drawing area and specify the value to the **Virtual thickness factor** parameter in the **Selection explorer**. Non-experienced users are advised not to change the default value. For more information about interface properties see the Reference Manual - *Chapter 5 - Advanced Geometric Modelling options.*

3.4.3 To define the excavation levels:

1. Click the **Create line** button in the side toolbar.
2. To define the first excavation stage move the cursor to position (50 18) at the wall and click. Move the cursor 15 m to the right (65 18) and click again. Click the right mouse button to finish drawing the first excavation stage.

3. To define the second excavation stage move the cursor to position (50 10) and click. Move to (65 10) and click again. Click the right mouse button to finish drawing the second excavation stage.
4. The third excavation stage is automatically defined as it corresponds to the boundary between the soil layers (y = 0).

3.4.4 To define the strut:

You will define a strut with the following material properties shown in Table 6 (on page 60) .

Table 6: Material properties of the strut (anchor)

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

1. Click the **Create structure** button in the side toolbar and select the **Create fixed-end anchor** in the expanded menu.
2. Move the cursor to (50 19) and click the left mouse button. A fixed-end anchor is added, being represented by a rotated T with a fixed size.
3. Click the **Show materials** button in the side toolbar. Set the **Set type** parameter in the **Material sets** window to **Anchors** and click the **New** button. Enter `Strut` as an **Identification** of the data set and enter the properties as given in Table 6 (on page 60). Click **OK** to close the data set.
4. Click **OK** to close the **Material sets**.
5. Make sure that the fixed-end anchor is selected in the drawing area.
6. In the **Selection explorer** assign the material data set to the strut by selecting the corresponding option in the **Material** drop-down menu.
   The anchor is oriented in the model according to the Direction$_x$ and Direction$_y$ parameters in the **Selection explorer** as shown in Figure 45 (on page 61). The default orientation is valid in this tutorial.
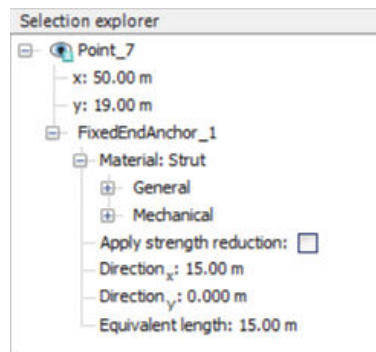
Figure 45: Parameters for fixed-end anchors in the Selection explorer

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

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

3.4.5 To define the distributed load:

1. Click the **Create load** button in the side toolbar.

2. Select the **Create line load** option in the expanded menu to define a distributed load.

3. Move the cursor to (43 20) and click, then move the cursor 5m to the right to (48 20) and click again. Right-click to finish the drawing.

4. In the **Selection explorer** assign a value of -5 kN/m/m to the y-component of the load ($q_{y,start,ref}$) as shown in Figure 46 (on page 61).
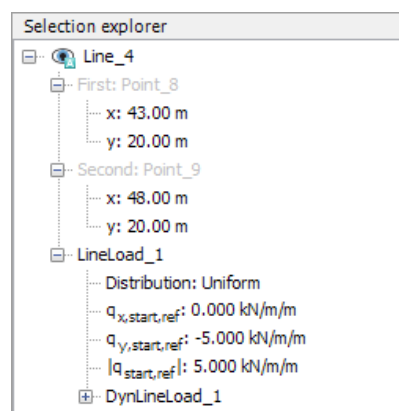
Figure 46: Components of the distributed load in the Selection explorer

3.5 Generate the mesh

In order to generate the mesh, follow these steps:

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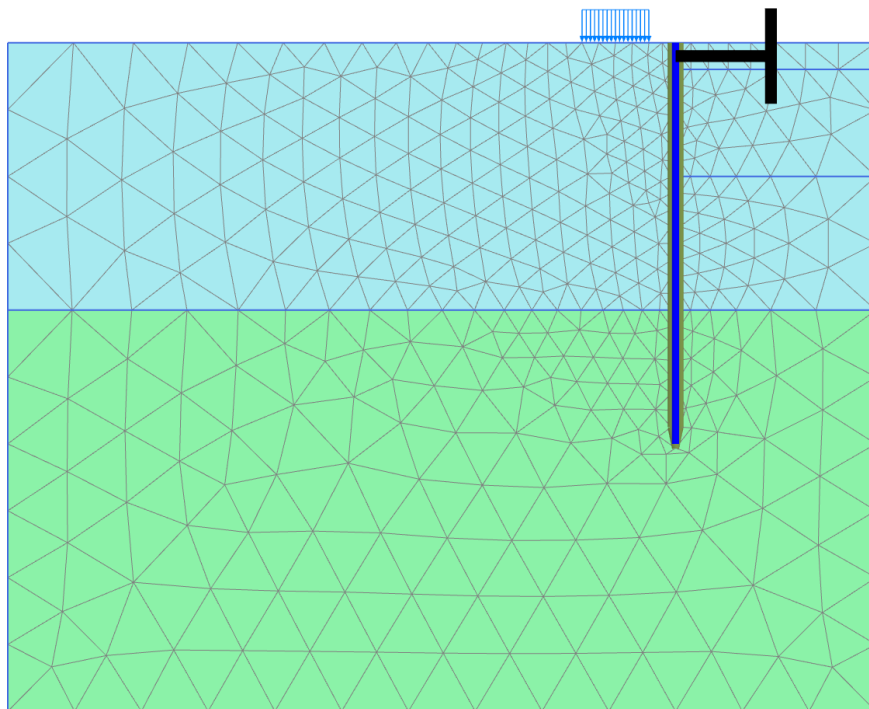3. Click the **View mesh** button 🔍 to view the mesh as shown in Figure 47 (on page 62).

Figure 47: The generated mesh

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

3.6 Define and perform the calculation

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

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

3.6.1 Initial phase

1. Click the **Staged construction** tab to proceed with the definition of calculation phases.
   The initial phase has already been created.
2. Keep its calculation type as **K0 procedure**. Make sure all the soil volumes are active and all the structural elements and load are inactive.

3.6.2 Phase 1: External load

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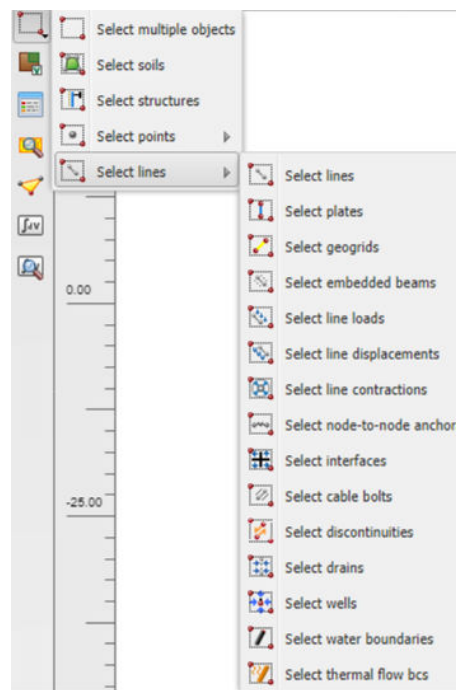2. Click the **Select multiple objects** button in the side toolbar. In the appearing menu select **Select line** > **Select plates** as shown in .

Figure 48: Select multiple objects

3. In the drawing area define a rectangle that includes all the plate elements as shown in .

4. Right-click the wall in the drawing area and select the **Activate option** from the context menu.
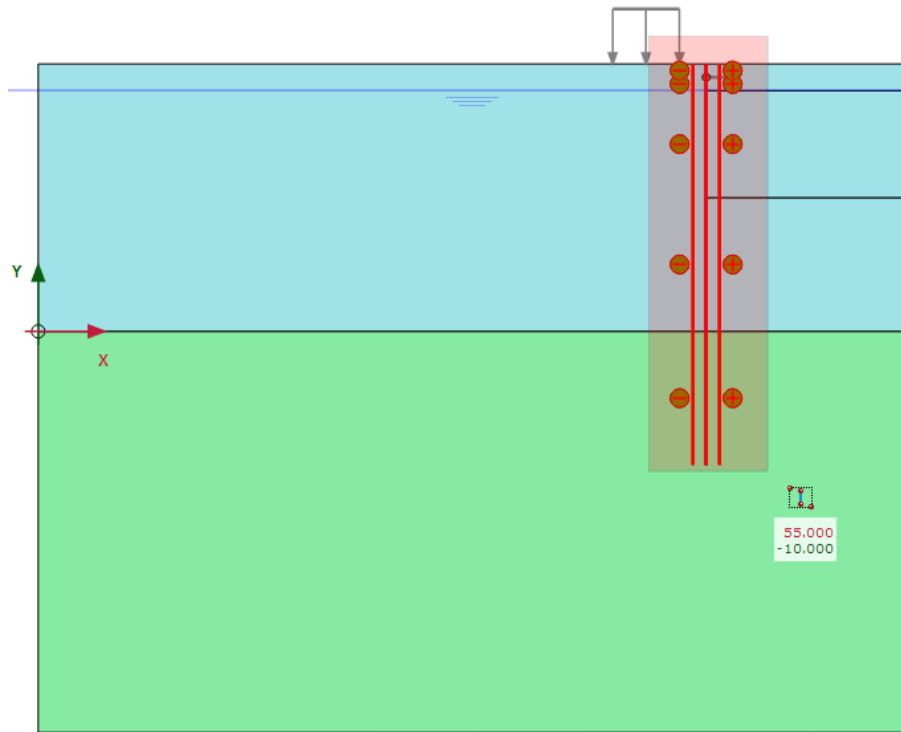
Figure 49: Selected multiple plate elements in the model

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

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

6. Make sure all the interfaces in the model are active.

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

3.6.3 Phase 2: First excavation stage

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

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

2. The default settings are valid for this phase. In the **Staged construction mode** all the structure elements except the fixed-end anchor are active.

3. In the drawing area right-click the top right cluster and select the **Deactivate** option in the appearing menu.

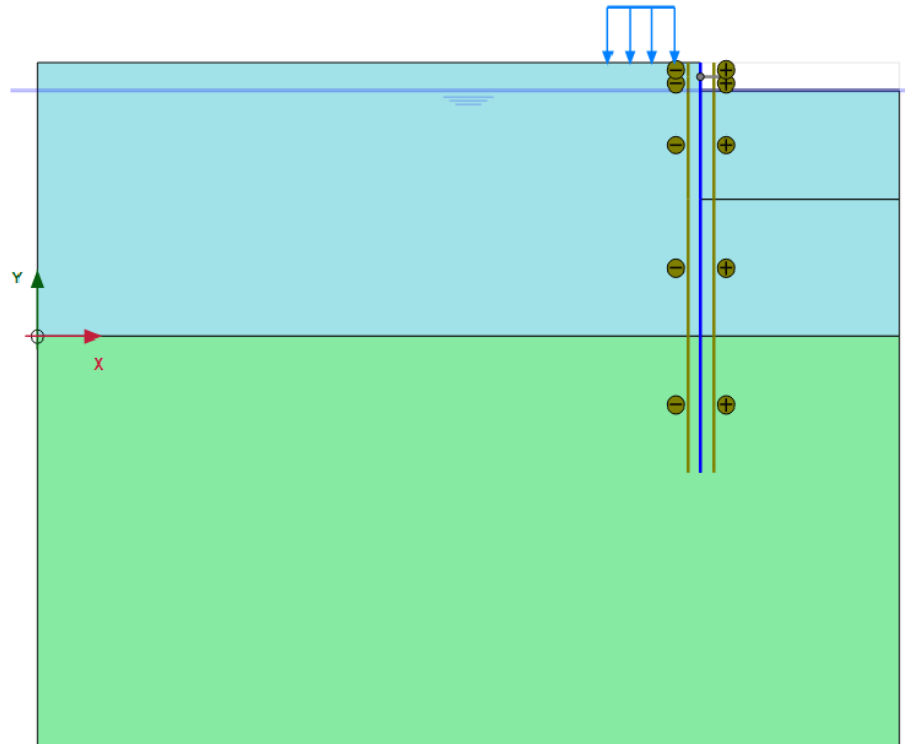The model for the first excavation phase is shown in :

Figure 50: Model for the first excavation phase

3.6.4 Phase 3: Installation of a strut

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

2. Activate the strut.
The strut turns black to indicate it is active.

3.6.5 Phase 4: Second (submerged) excavation stage

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

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

> **Note:** Note that in PLAXIS 2D the pore pressures are not automatically deactivated when deactivating a soil cluster. Hence, in this case, the water remains in the excavated area and a submerged excavation is simulated.
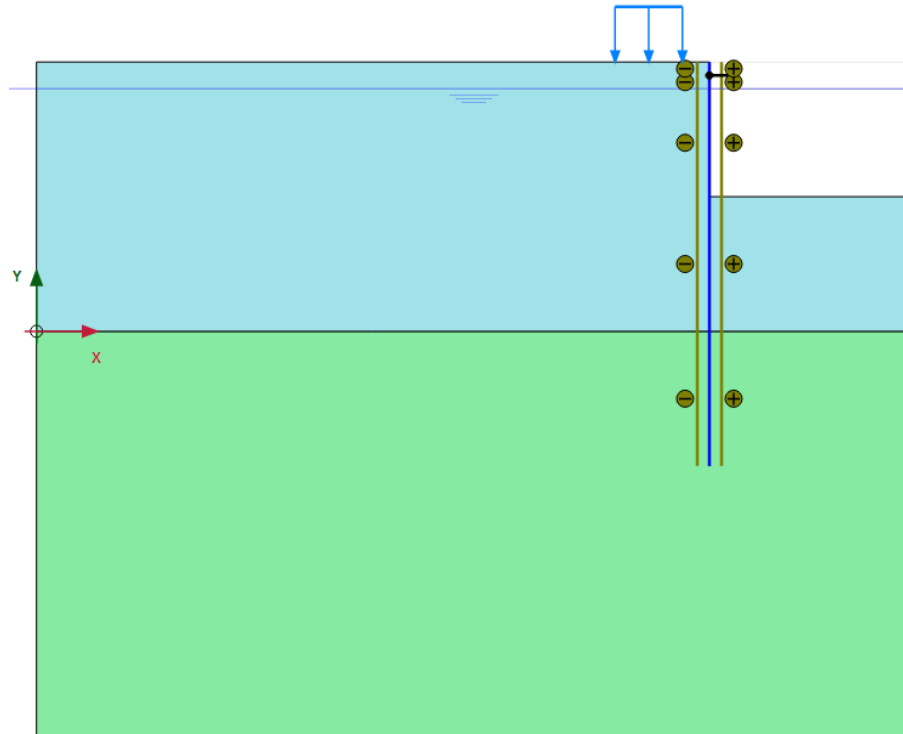
The resulting model is shown in Figure 51 (on page 67) :

Figure 51: Model for the second excavation phase

3.6.6 Phase 5: Third excavation stage

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

1. Click the **Add phase** button 🔲 to add a new phase.
2. Deactivate the third cluster from the top on the right hand side of the mesh.
   The model for the phase is shown in Figure 52 (on page 68):
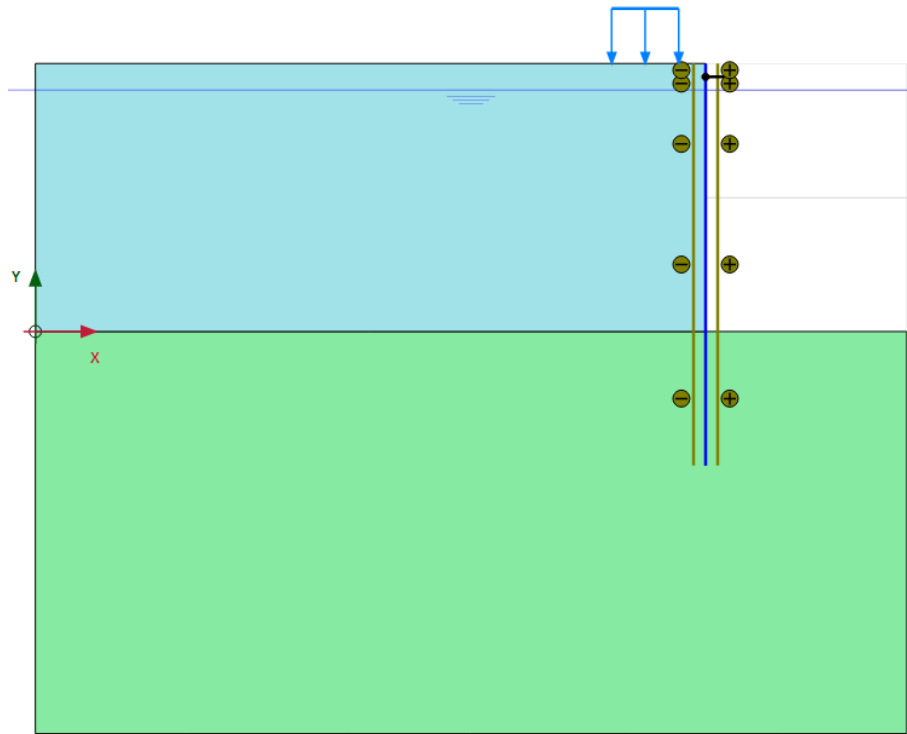
Figure 52: Model for the third excavation phase

The calculation definition is now complete.

3.6.7 Execute the calculation

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

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

During a **Staged construction** calculation phase, a multiplier called ΣMstage is increased from 0.0 to 1.0. This parameter is displayed in the calculation info window. As soon as ΣMstage has reached the value 1.0, the construction stage is completed and the calculation phase is finished. If a **Staged construction** calculation finishes while ΣMstage is smaller than 1.0, the program will give a warning message. The most likely reason for

not finishing a construction stage is that a failure mechanism has occurred, but there can be other causes as well. See the Reference Manual for more information about **Staged construction**.

3.7 View the calculation results

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

3.7.1 Displacements and stresses

To view displacements and stresses, follow these steps:

1. Click the final calculation phase in the **Calculations** window.

2. Click the **View calculation results** button on the toolbar.
   As a result, the Output program is started, showing the deformed mesh (scaled up) as shown in Figure 53 (on page 69) at the end of the selected calculation phase, with an indication of the maximum displacement:
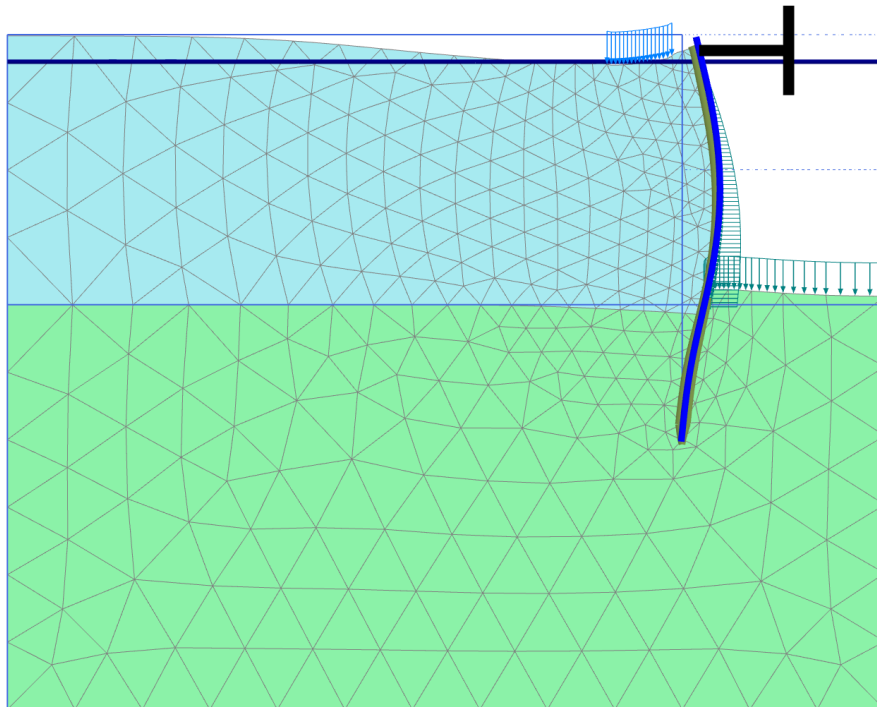

Figure 53: Deformed mesh after the third excavation stage

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

3. Select the menu **Deformations** > **Incremental displacements** > **|Δu|**.
   The plot shows colour shadings of the displacement increments, which indicates the forming of a mechanism of soil movement behind the wall.

4. Click the **Arrows** button ➔ in the toolbar.
   The plot shows the displacement increments of all nodes as arrows. The length of the arrows indicates the relative magnitude.

5. Select the menu **Stresses** > **Principal effective stresses** > **Effective principal stresses**.

   The plot in Figure 54 (on page 70) shows the effective principal stresses at the three middle stress points of each soil element with an indication of their direction and their relative magnitude. Note that the **Center principal stresses** button is selected in the toolbar. The orientation of the principal stresses indicates a large passive zone under the bottom of the excavation and a small passive zone behind the strut.
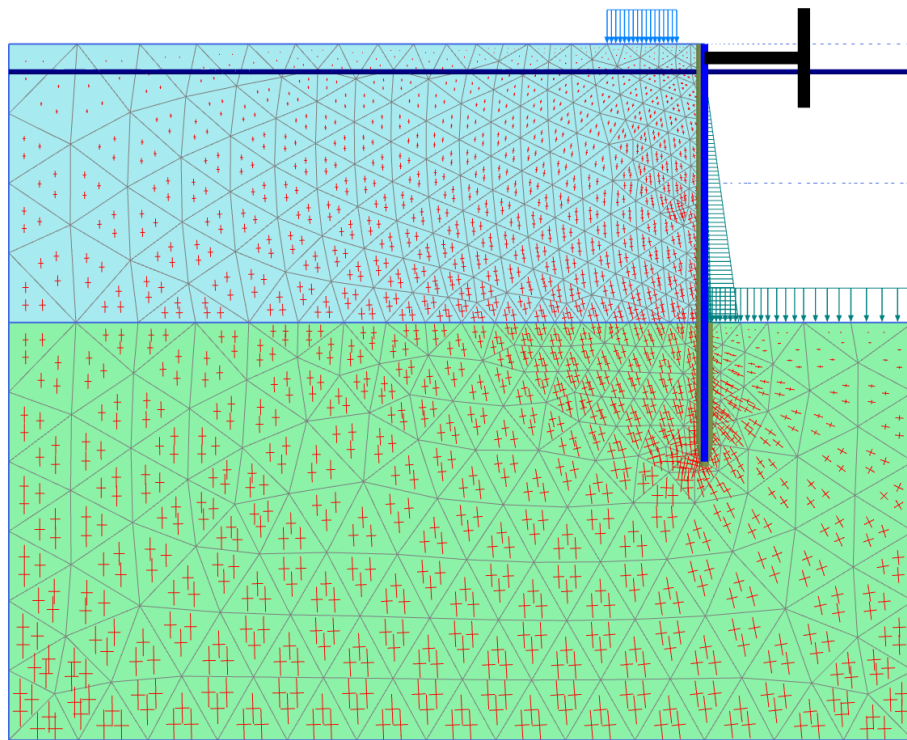

Figure 54: Principal stresses after excavation

3.7.2 Shear forces and bending moments

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

1. Double-click the wall.

A new window is opened showing the axial force.

2. Select the menu **Forces** > **bending moment M**.
   The bending moment in the wall is displayed in Figure 55 (on page 71) with an indication of the maximum moment:
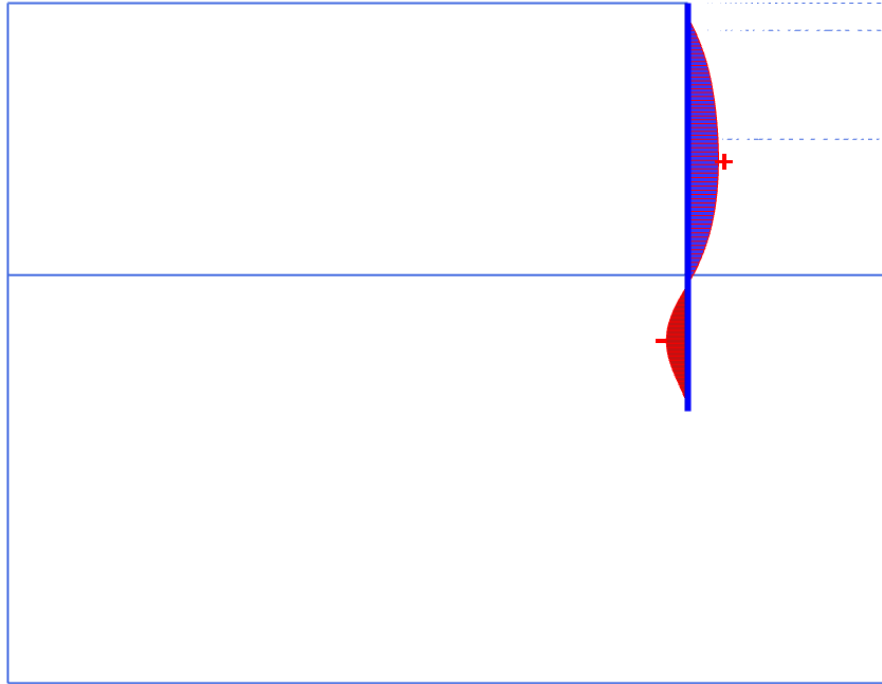

Figure 55: Bending moments in the wall

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

   The plot now shows the shear forces in the wall.

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

4. Select the first window (showing the effective stresses in the full geometry) from the **Window** menu. Double-click the strut.
   The strut force (in kN) is shown in the displayed table.

5. Click the **Curves manager** button on the toolbar.
   As a result, the **Curves manager** window pops up.

6. Click **New** to create a new chart.
   The **Curve generation** window pops up.

7. For the x-axis select the point A from the drop-down menu. In the tree select **Deformations** - **Total displacements** - **|u|**.

8. For the y-axis keep the **Project** option in the drop-down menu. In the tree select **Multiplier** - **ΣMstage**.

9. Click **OK** to accept the input and generate the load-displacement curve.
   As a result the curve is plotted as shown in Figure 56 (on page 72) :

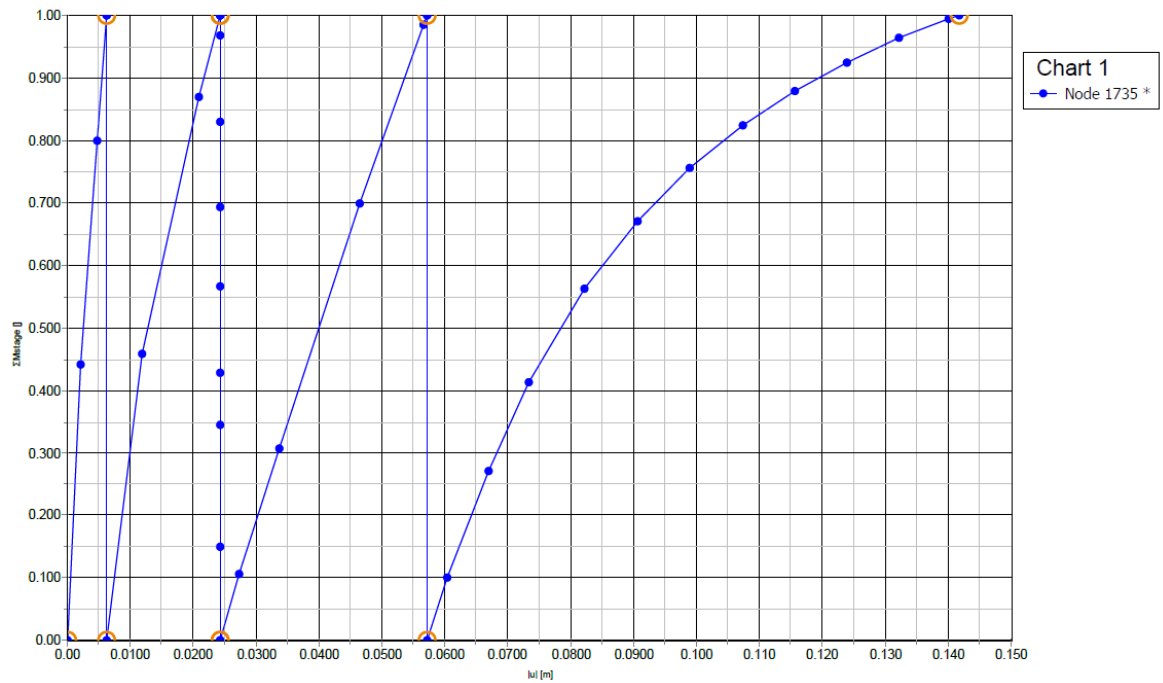*Figure 56: Load-displacement curve of deflection of wall*

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