2 SUBMERGED CONSTRUCTION OF AN EXCAVATION

This tutorial illustrates the use of PLAXIS 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.

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 (Figure 2.1).

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. 30 m of the sand layer are considered in the model.

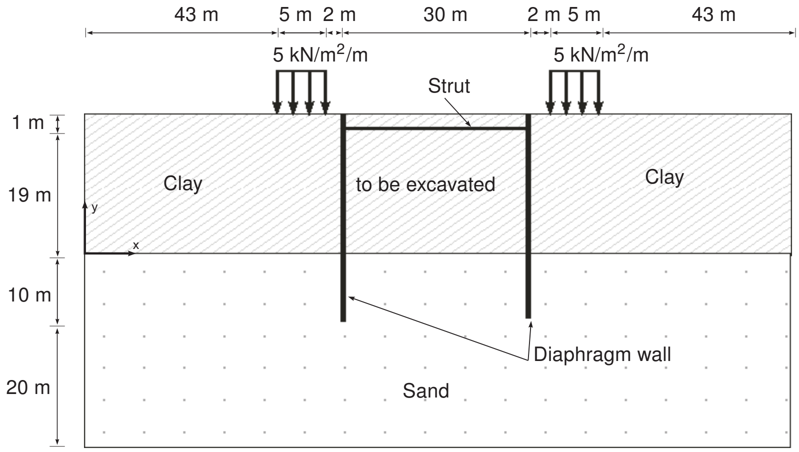

Figure 2.1 Geometry model of the situation of a submerged excavation

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.

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).

2.1 INPUT

To create the geometry model, follow these steps:

General settings

- Start the Input program and select *Start a new project* from the *Quick select* dialog box.
- In the *Project* tabsheet of the *Project properties* window, enter an appropriate title.
- In the *Model* tabsheet keep the default options for *Model* (*Plane strain*), and *Elements* (*15-Node*).
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 65.0 m, y_{min} = -30.0 m and y_{max} = 20.0.
- Keep the default values for units and constants and press *OK* to close the *Project properties* window.

Definition of soil stratigraphy

To define the soil stratigraphy:

Create a borehole at x = 0. The *Modify soil layers* window pops up.

- Add the top soil layer and specify its height by setting the top level to 20 m and the bottom level to 0 m.
- 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.
- Set the *Head* in the borehole to 18.0 m.

Two data sets need to be created; one for the clay layer and one for the sand layer. To create the material data sets, follow these steps:

Click the *Materials* button 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*.

- Click the *New* button in the *Material sets* window to create a new data set.
- For the clay layer, enter "Clay" for the *Identification* and select *Soft soil* as the *Material model*. Set the *Drainage type* to *Undrained (A)*.
- Enter the properties of the clay layer, as listed in Table 2.1, in the *General*,

Parameters and *Flow parameters* tabsheets.

Table 2.1 Material properties of the sand and clay layer and the interfaces

Parameter	Name	Clay	Sand	Unit
General				
Material model	*Model*	Soft soil	Hardening soil	-
Type of material behaviour	*Type*	Undrained (A)	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	16	17	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	18	20	kN/m^3
Initial void ratio	e_{init}	1.0	0.5	-
Parameters				
Modified compression index	λ^*	$3.0\cdot 10^{-2}$	-	-
Modified swelling index	κ^*	$8.5\cdot 10^{-3}$	-	-
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	-	$4.0\cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	-	$4.0\cdot 10^4$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	-	$1.2\cdot 10^5$	kN/m^2
Power for stress-level dependency of stiffness	m	-	0.5	-
Cohesion (constant)	c_{ref}'	1.0	0.0	kN/m^2
Friction angle	φ'	25	32	°
Dilatancy angle	ψ	0.0	2.0	°
Poisson's ratio	ν_{ur}'	0.15	0.2	-
K_0-value for normal consolidation	K_0^{nc}	0.5774	0.4701	-
Groundwater				
Permeability in horizontal direction	k_x	0.001	1.0	m/day
Permeability in vertical direction	k_y	0.001	1.0	m/day
Interfaces				
Interface strength	—	Manual	Manual	-
Strength reduction factor inter.	R_{inter}	0.5	0.67	-
Initial				
K_0 determination	—	Automatic	Automatic	-
Over-consolidation ratio	*OCR*	1.0	1.0	-
Pre-overburden pressure	*POP*	5.0	0.0	kN/m^2

- 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:

$$\tan\varphi_{interface} = R_{inter}\tan\varphi_{soil} \text{ and } c_{inter} = R_{inter}c_{soil}$$

where:

$$c_{soil} = c_{ref} \quad \text{(see Table 2.1)}$$

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.

- 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.
- For the sand layer, enter "Sand" for the *Identification* and select *Hardening soil* as the *Material model*. The material type should be set to *Drained*.
- Enter the properties of the sand layer, as listed in Table 2.1, in the corresponding edit boxes of the *General* and *Parameters* tabsheet.
- 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.

- Assign the material datasets to the corresponding soil layers.

Hint: 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 (Section 6.1.5 of the Reference Manual).

» Instead of accepting the default data sets of interfaces, data sets can directly be assigned to interfaces by selecting the proper data set in the *Material mode* drop-down menu in the *Object explorers*.

2.1.1 DEFINITION OF STRUCTURAL ELEMENTS

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

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

To define the diaphragm wall:

- Click the *Create structure* button in the side toolbar.
- In the expanded menu select the *Create plate* option (Figure 2.2).

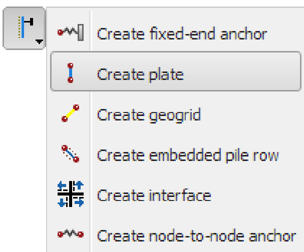

Figure 2.2 The *Create plate* option in the *Create structures* menu

- In the draw area move the cursor to position (50.0 20.0) at the upper horizontal line and click. Move 30 m down (50.0 -10.0) and click. Click the right mouse button to finish the drawing.
- 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 2.2.
- Click *OK* to close the data set.
- 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.
- Click *OK* to close the *Material sets* window.

To define interfaces:

Table 2.2 Material properties of the diaphragm wall (Plate)

Parameter	Name	Value	Unit
Type of behaviour	*Material type*	Elastic; Isotropic	
Normal stiffness	*EA*	$7.5 \cdot 10^6$	kN/m
Flexural rigidity	*EI*	$1.0 \cdot 10^6$	kNm^2/m
Unit weight	*w*	10.0	kN/m/m
Poisson's ratio	ν	0.0	-

Hint: 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 Section 5.2.4 of the Reference Manual.

- Right-click the plate representing the diaphragm wall. Point to *Create* and click on the *Positive interface* option in the appearing menu (Figure 2.3). In the same way assign a negative interface as well.

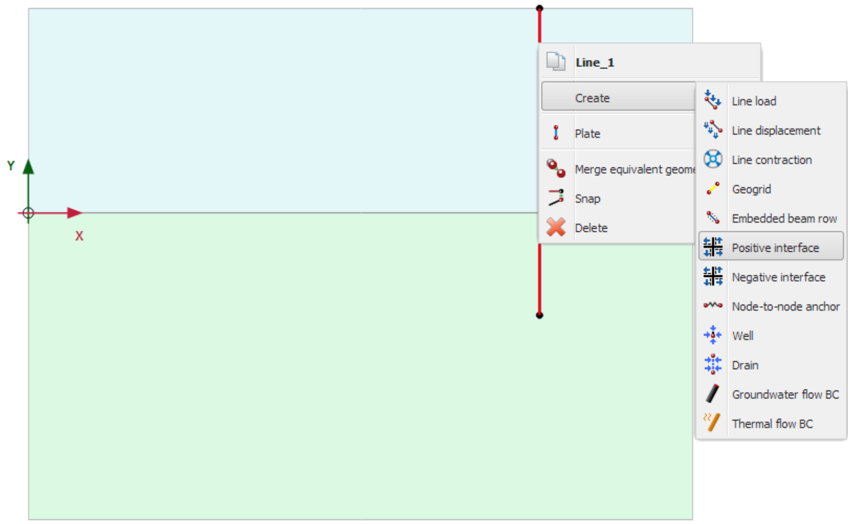

Figure 2.3 *Positive interface* assignment to existing geometry

Hint: In order to identify interfaces at either side of a geometry line, a positive sign (⊕) or negative sign (⊖) 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 draw 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.

To define the excavation levels:

- Click the *Create line* button in the side toolbar.
- To define the first excavation stage move the cursor to position (50.0 18.0) at the wall and click. Move the cursor 15 m to the right (65.0 18.0) and click again. Click the right mouse button to finish drawing the first excavation stage.
- To define the second excavation stage move the cursor to position (50.0 10.0) and click. Move to (65.0 10.0) and click again. Click the right mouse button to finish drawing the second excavation stage.
- The third excavation stage is automatically defined as it corresponds to the boundary between the soil layers (y = 0.0).

To define the strut:

- Click the *Create structure* button in the side toolbar and select the *Create fixed-end anchor* button in the expanded menu.
- Move the cursor to (50.0 19.0) and click the left mouse button. A fixed-end anchor is is added, being represented by a rotated T with a fixed size.
- Click the *Show materials* button in the side toolbar. Set the *Set type* parameter in the *Material sets* window to *Anchor* and click the *New* button. Enter "Strut" as an *Identification* of the data set and enter the properties as given in Table 2.3. Click *OK* to close the data set.
- Click *OK* to close the *Material sets* window.

Table 2.3 Material properties of the strut (anchor)

Parameter	Name	Value	Unit
Type of behaviour	*Material type*	Elastic	-
Normal stiffness	*EA*	$2 \cdot 10^6$	kN
Spacing out of plane	$L_{spacing}$	5.0	m

- Make sure that the fixed-end anchor is selected in the draw area.
- 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*. The default orientation is valid in this tutorial.
- Enter an *Equivalent length* of 15 m corresponding to half the width of the excavation (Figure 2.4).

Hint: The *Equivalent length* is the distance between the connection point and the position in the direction of the anchor rod where the displacement is zero.

To define the distributed load:

- Click the *Create load* button in the side toolbar
- Select the *Create line load* option in the expanded menu to define a distributed load (Figure 2.5).

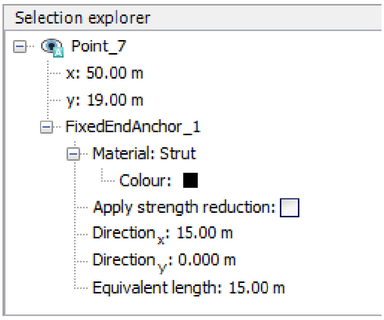

Figure 2.4 Parameters for fixed-end anchors in the *Selection explorer*

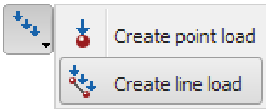

Figure 2.5 The *Create line load* option in the *Create load* menu

- Move the cursor to (43.0 20.0) and click. Move the cursor 5 m to the right to (48.0 20.0) and click again. Right-click to finish drawing.
- In the *Selection explorer* assign a value of -5 kN/m/m to the y-component of the load ($q_{y,start,ref}$) (Figure 2.6).

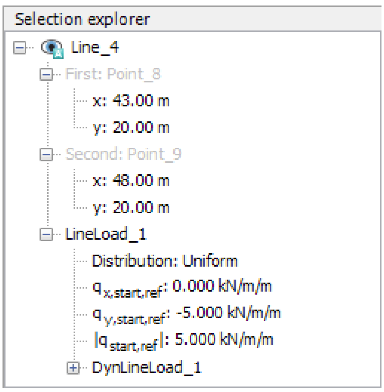

Figure 2.6 Components of the distributed load in the *Selection explorer*

2.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Create the mesh. Use the default option for the *Element distribution* parameter (*Medium*).
- View the mesh. The resulting mesh is displayed in Figure 2.7.
- Click on the *Close* tab to close the Output program.

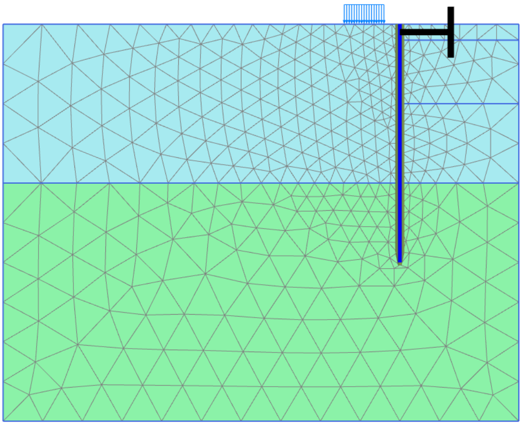

Figure 2.7 The generated mesh

2.3 CALCULATIONS

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, 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.

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

Phase 1: External load

In the *Phases explorer* click the *Add phase* button to introduce 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.

Click the *Select multiple objects* button in the side toolbar. In the appearing menu point to *Select line* and click on the *Select plates* option (Figure 2.8).

- In the draw area define a rectangle including all the plate elements (Figure 2.9).
- Right-click the wall in the draw area and select the *Activate option* from the appearing menu. The wall is now visible in the color that is specified in the material dataset.

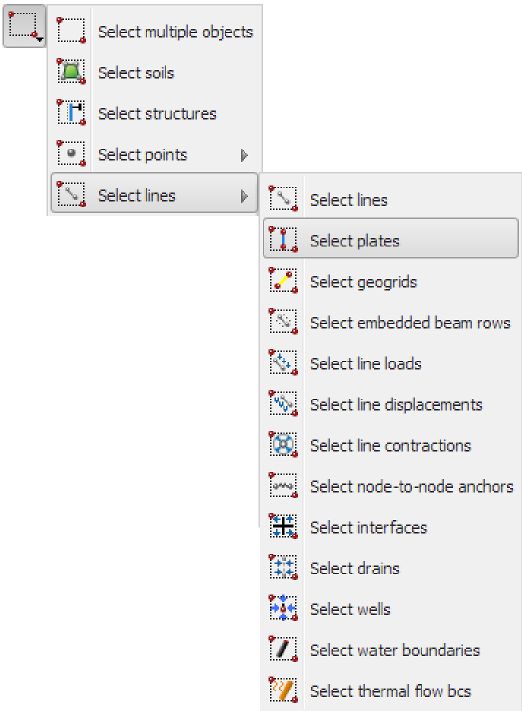

Figure 2.8 The *Select plates* option

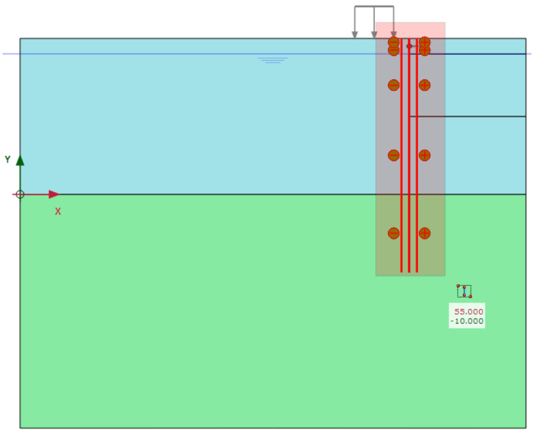

Figure 2.9 Multi-selection of plates in the draw area

- 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*.
- Make sure all the interfaces in the model are active.

Phase 2: First excavation stage

In the *Phases explorer* click the *Add phase* button to introduce a new phase.

Hint: 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 appearing menu.

- 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.

Hint: 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.

- The default settings are valid for this phase. In the *Staged construction* mode all the structure elements except the fixed-end anchor are active.
- In the draw area right-click the top right cluster and select the *Deactivate* option in the appearing menu. Figure 2.10 displays the model for the first excavation phase.

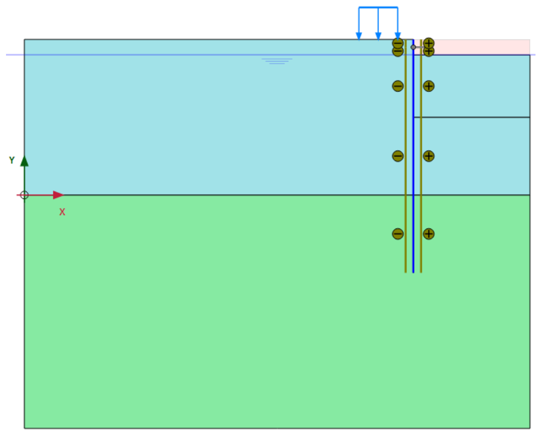

Figure 2.10 Model view for the first excavation phase

Phase 3: Installation of strut

Add a new phase.

- Activate the strut. The strut should turn black to indicate it is active.

Phase 4: Second (submerged) excavation stage

Add a new phase.

- Deactivate the second cluster from the top on the right side of the mesh. It should be the topmost active cluster (Figure 2.11).

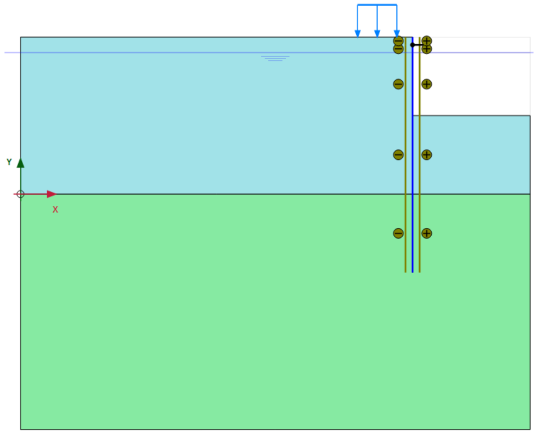

Figure 2.11 Model view for the second excavation phase

Hint: Note that in PLAXIS 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.

Phase 5: Third excavation stage

Add a new phase.

- In the final calculation stage the excavation of the last clay layer inside the pit is simulated. Deactivate the third cluster from the top on the right hand side of the mesh (Figure 2.12).

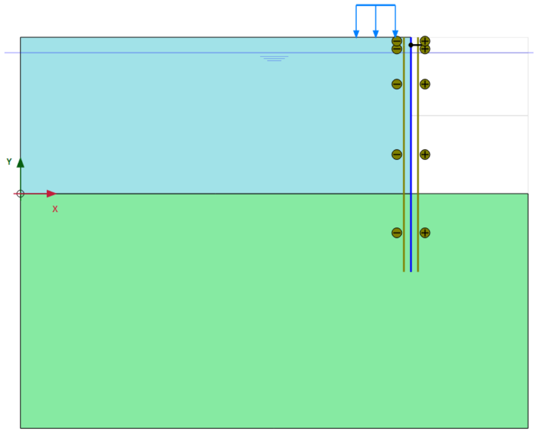

Figure 2.12 Model view for the third excavation phase

The calculation definition is now complete. 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.

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.

- Select some nodes on the wall at points where large deflections can be expected (e.g. 50.0 10.0). The nodes located near that specific location are listed. Select the convenient one by checking the box in front of it in the list. Close the *Select points* window.
- Click on the *Update* tab to close the Output program and go back to the *Input* program.

 Calculate the project.

During a *Staged construction* calculation phase, a multiplier called $\Sigma Mstage$ is increased from 0.0 to 1.0. This parameter is displayed on the calculation info window. As soon as $\Sigma Mstage$ has reached the value 1.0, the construction stage is completed and the calculation phase is finished. If a *Staged construction* calculation finishes while $\Sigma Mstage$ 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*.

2.4 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:

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

 Click the *View calculation results* button on the toolbar. As a result, the Output program is started, showing the deformed mesh (scaled up) at the end of the selected calculation phase, with an indication of the maximum displacement (Figure 2.13).

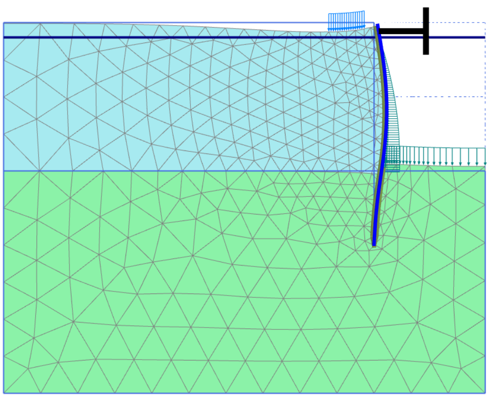

Figure 2.13 Deformed mesh after the third excavation stage

- Select $|\Delta u|$ from the side menu displayed as the mouse pointer is located on the *Incremental displacements* option of the *Deformations* menu. The plot shows colour shadings of the displacement increments, which indicates the forming of a

Hint: 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.

'mechanism' of soil movement behind the wall.

- 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.
- In the *Stresses* menu point to the *Principal effective stresses* and select the *Effective principal stresses* option from the appearing menu. The plot 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 2.14).

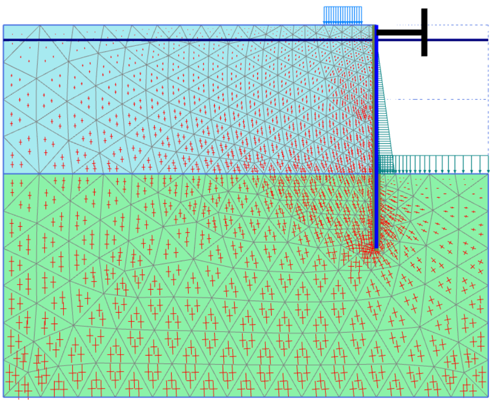

Figure 2.14 Principal stresses after excavation

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

- Double-click the wall. A new window is opened showing the axial force.
- Select the *bending moment M* from the *Forces* menu. The bending moment in the wall is displayed with an indication of the maximum moment (Figure 2.15).
- Select *Shear forces Q* from the *Forces* menu. The plot now shows the shear forces in the wall.

Hint: 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.

- 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

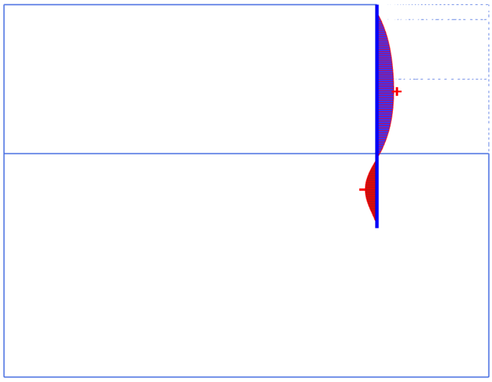

Figure 2.15 Bending moments in the wall

displayed table.

- Click the *Curves manager* button on the toolbar. As a result, the *Curves manager* window will pop up.
- Click *New* to create a new chart. The *Curve generation* window pops up.
- For the x-axis select the point A from the drop-down menu. In the tree select *Deformations* - *Total displacements* - *|u|*.
- For the y-axis keep the *Project* option in the drop-down menu. In the tree select *Multiplier* - Σ*Mstage*.
- Click *OK* to accept the input and generate the load-displacement curve. As a result the curve of Figure 2.16 is plotted.

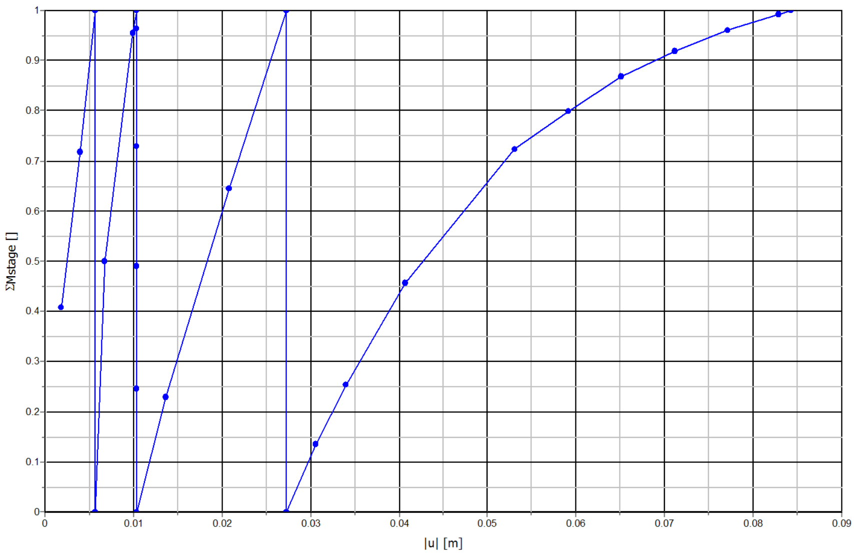

Figure 2.16 Load-displacement curve of deflection of wall

The curve shows the construction stages. For each stage, the parameter Σ*Mstage* 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.