Tutorial Lesson for Plaxis V8

by
Griffith University
Gold Coast Campus
Australia

2.2 GENERAL MODELLING ASPECTS

For each new project to be analysed it is important to create a geometry model first. A geometry model is a 2D representation of a real three-dimensional problem and consists of points, lines and clusters. A geometry model should include a representative division of the subsoil into distinct soil layers, structural objects, construction stages and loadings. The model must be sufficiently large so that the boundaries do not influence the results of the problem to be studied. The three types of components in a geometry model are described below in more detail.

Points:

Points form the start and end of lines. Points can also be used for the positioning of anchors, point forces, point fixities and for local refinements of the finite element mesh.

Lines:

Lines are used to define the physical boundaries of the geometry, the model boundaries and discontinuities in the geometry such as walls or shells, separations of distinct soil layers or construction stages. A line can have several functions or properties.

Clusters:

Clusters are areas that are fully enclosed by lines. PLAXIS automatically recognises clusters based on the input of geometry lines. Within a cluster the soil properties are homogeneous. Hence, clusters can be regarded as parts of soil layers. Actions related to clusters apply to all elements in the cluster.

After the creation of a geometry model, a finite element model can automatically be generated, based on the composition of clusters and lines in the geometry model. In a finite element mesh three types of components can be identified, as described below.

Elements:

During the generation of the mesh, clusters are divided into triangular elements. A choice can be made between 15-node elements and 6-node elements. The powerful 15-node element provides an accurate calculation of stresses and failure loads. In addition, 6-node triangles are available for a quick calculation of serviceability states. Considering the same element distribution (for example a default coarse mesh generation) the user should be aware that meshes composed of 15-node elements are actually much finer and much more flexible than meshes composed of 6-node elements, but calculations are also more time consuming. In addition to the triangular elements, which are generally used to model the soil, compatible plate elements, geogrid elements and interface elements may be generated to model structural behaviour and soil-structure interaction.

Nodes:

A 15-node element consists of 15 nodes and a 6-node triangle is defined by 6 nodes. The distribution of nodes over the elements is shown in Figure 2.1. Adjacent elements are connected through their common nodes. During a finite element calculation, displacements (u_x and u_y) are calculated at the nodes. Nodes may be pre-selected for the generation of load-displacement curves.

Stress points:

In contrast to displacements, stresses and strains are calculated at individual Gaussian integration points (or stress points) rather than at the nodes. A 15-node triangular element contains 12 stress points as indicated in Figure 2.1a and a 6-node triangular element contains 3 stress points as indicated in Figure 2.1b. Stress points may be pre-selected for the generation of stress paths or stress-strain diagrams.

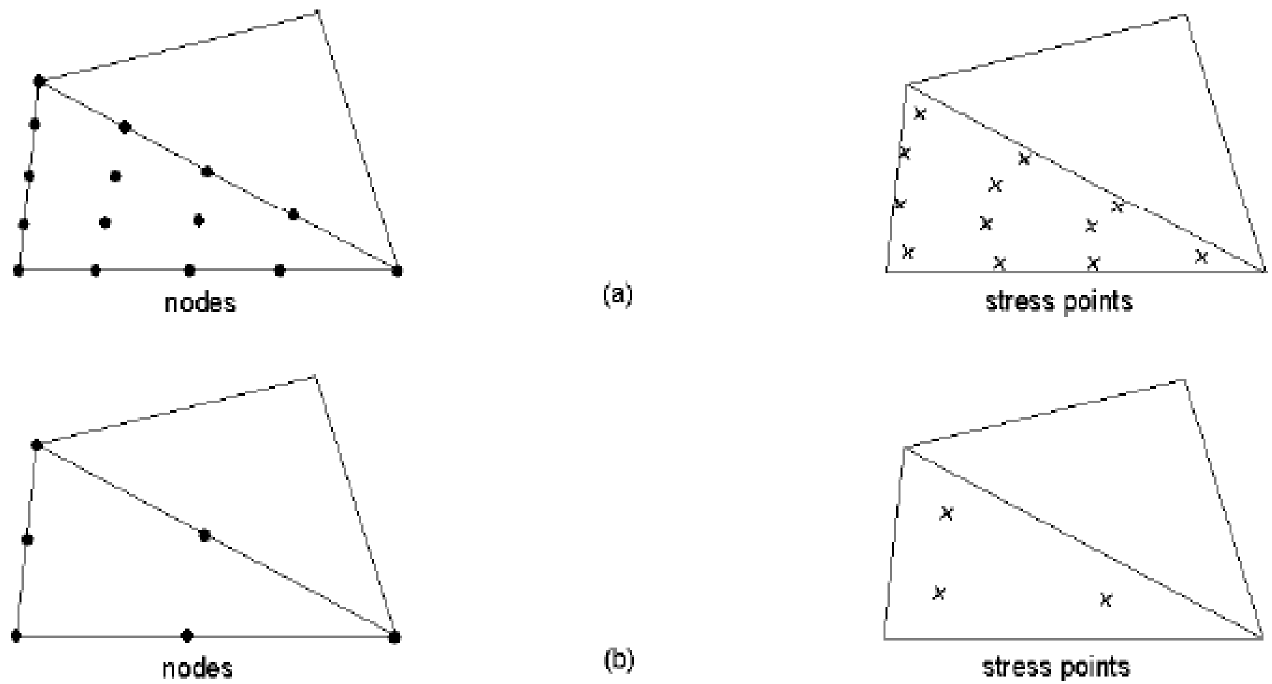


Figure 2.1 Nodes and stress points

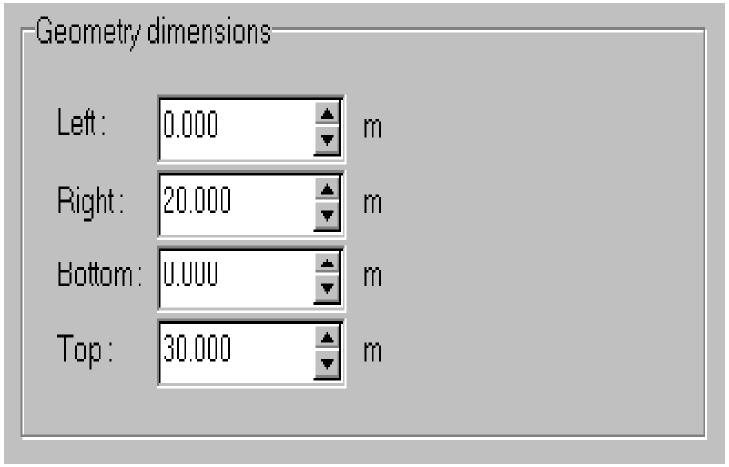


Figure 2.2 Spin edits

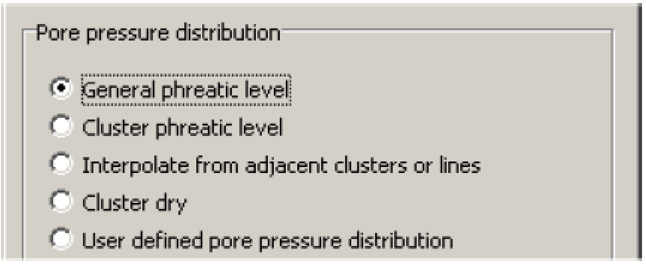


Figure 2.3 Radio buttons

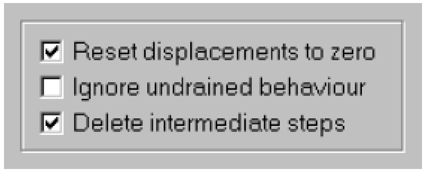


Figure 2.4 Check boxes

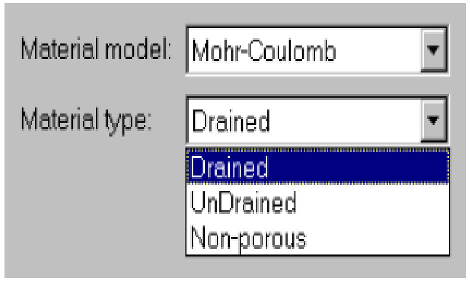


Figure 2.5 Combo boxes

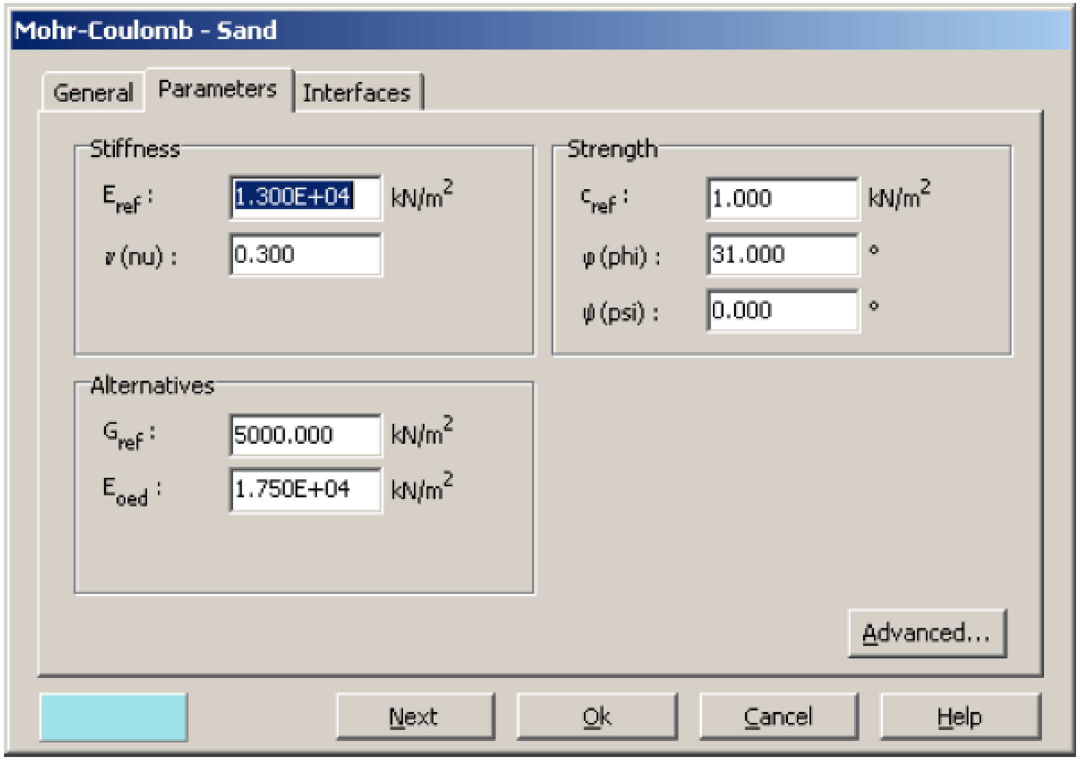


Figure 2.6 Page control and tab sheets

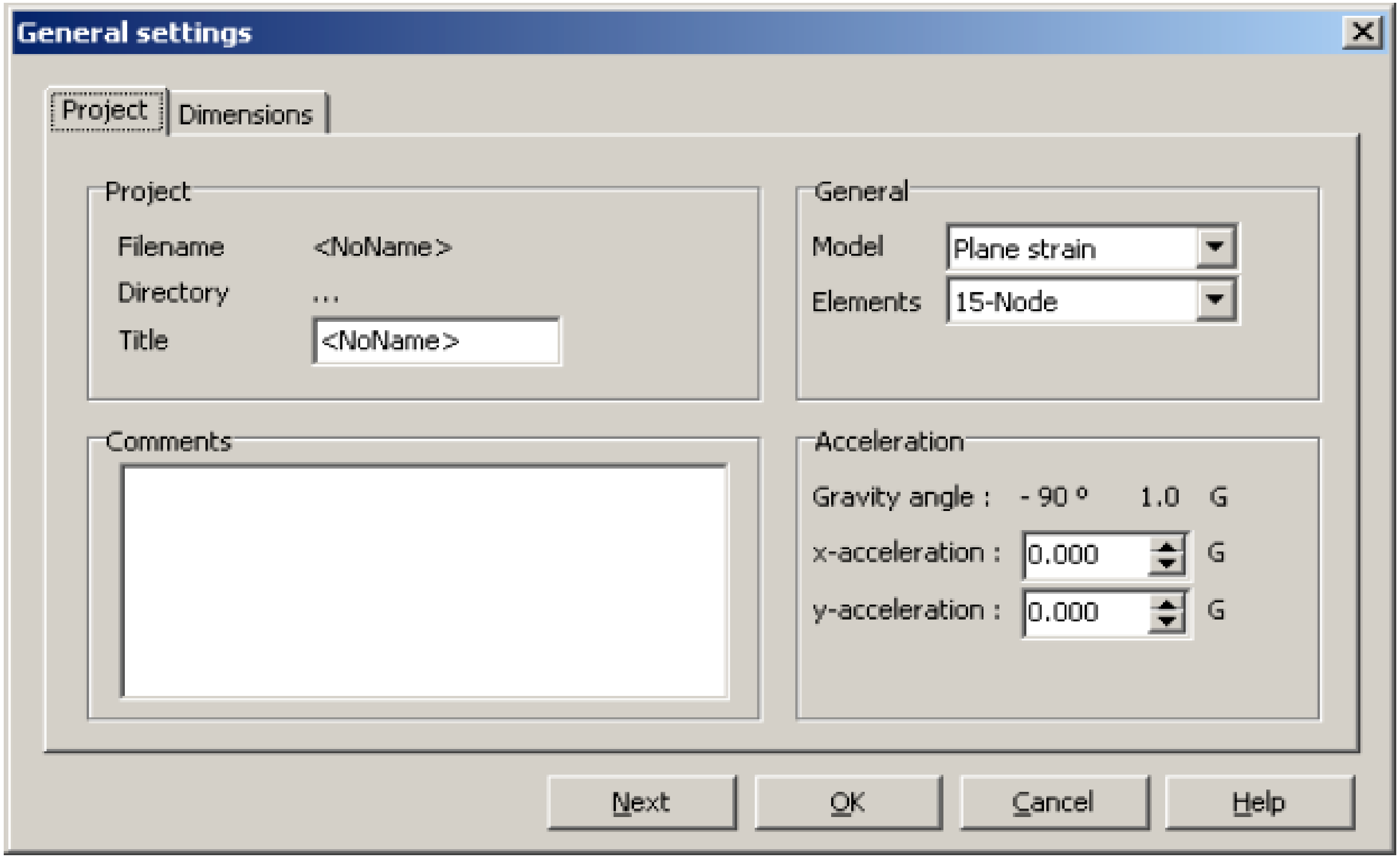


Figure 2.7 General settings - *General* tab sheet

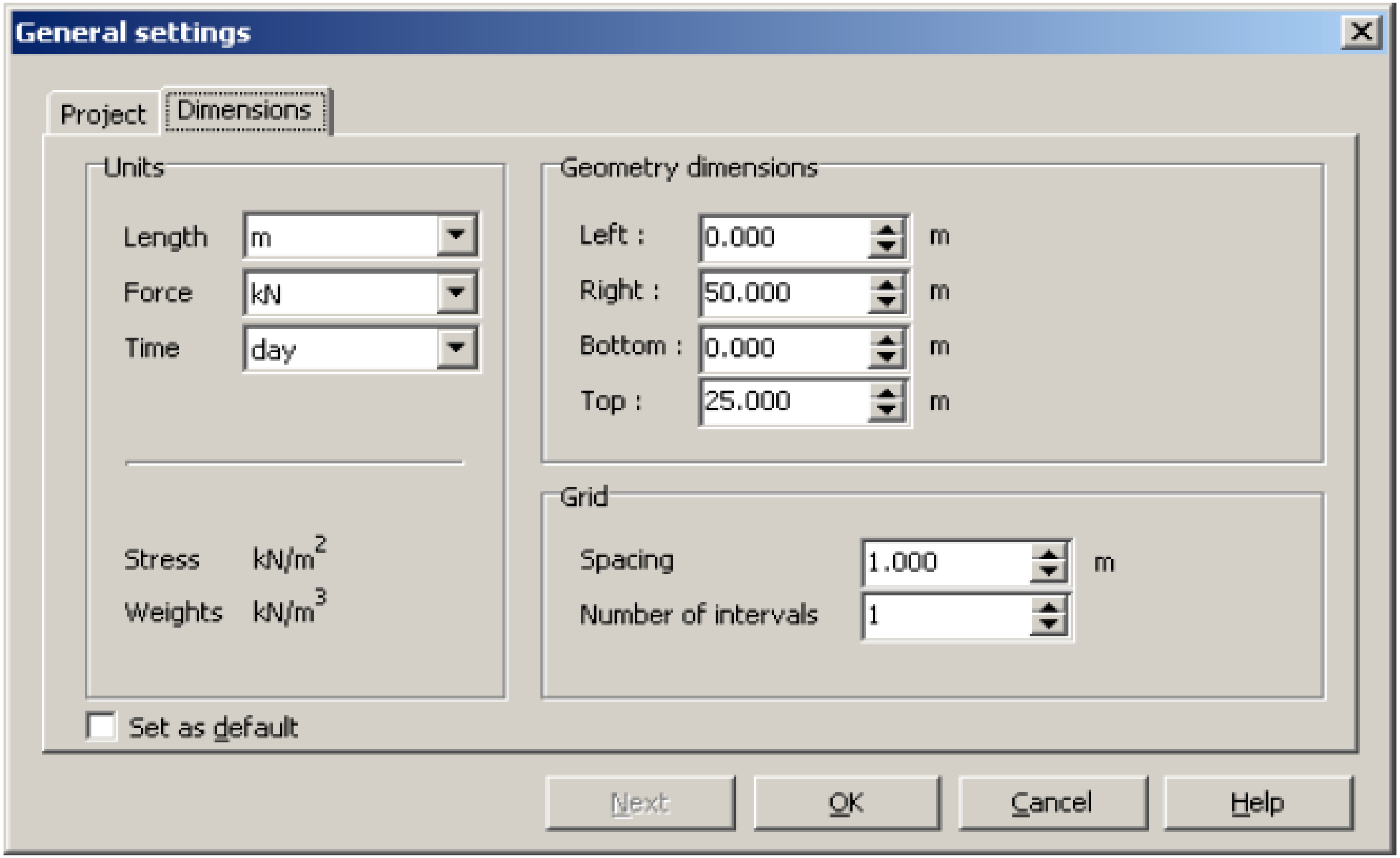


Figure 2.8 General settings - *Dimensions* tab sheet

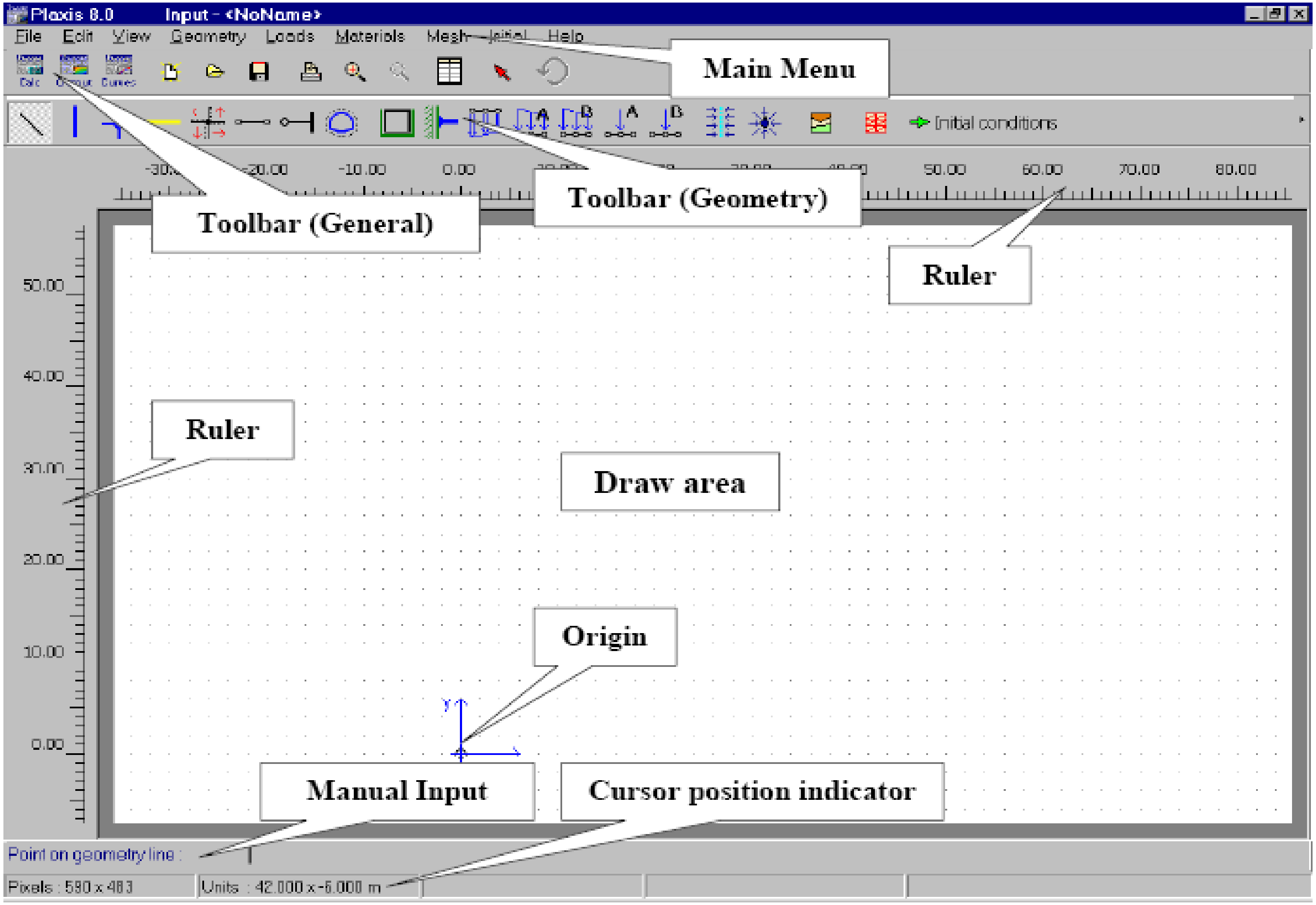


Figure 2.9 Main window of the Input program

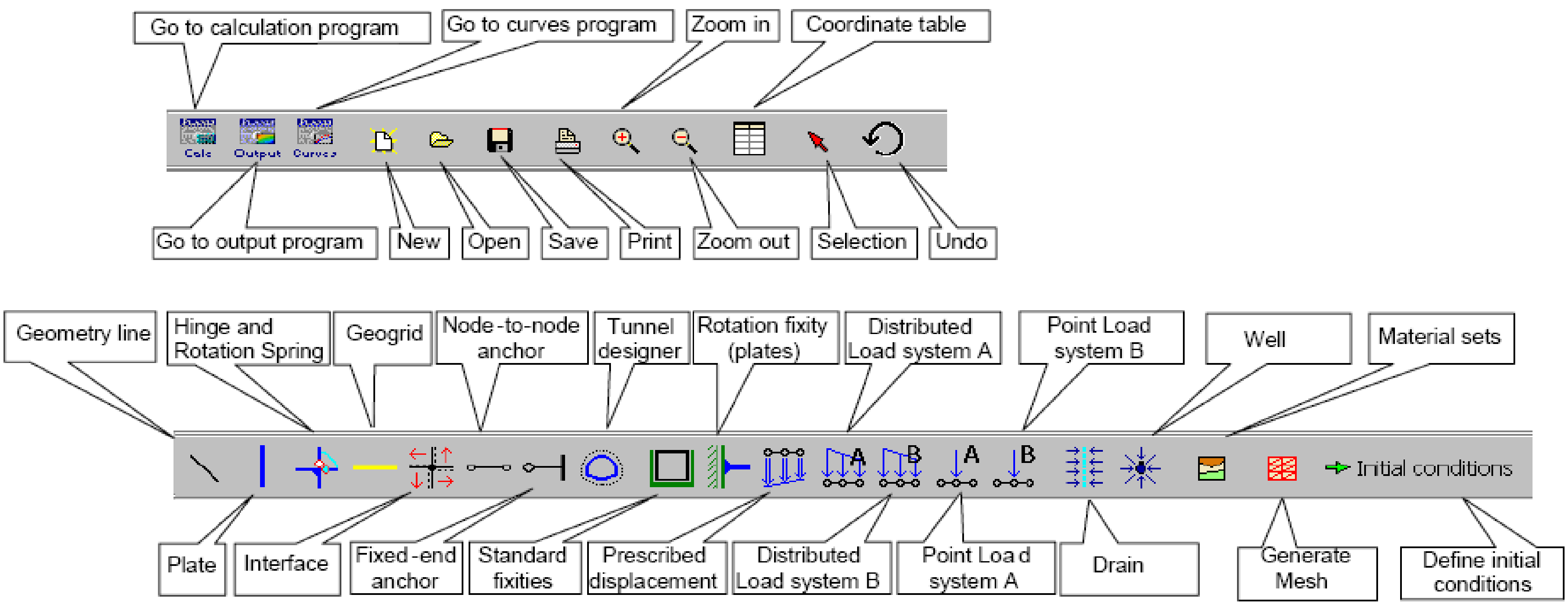


Figure 2.10 Toolbars

LESSON 1

SETTLEMENT OF A CIRCULAR FOOTING ON SAND

GEOMETRY

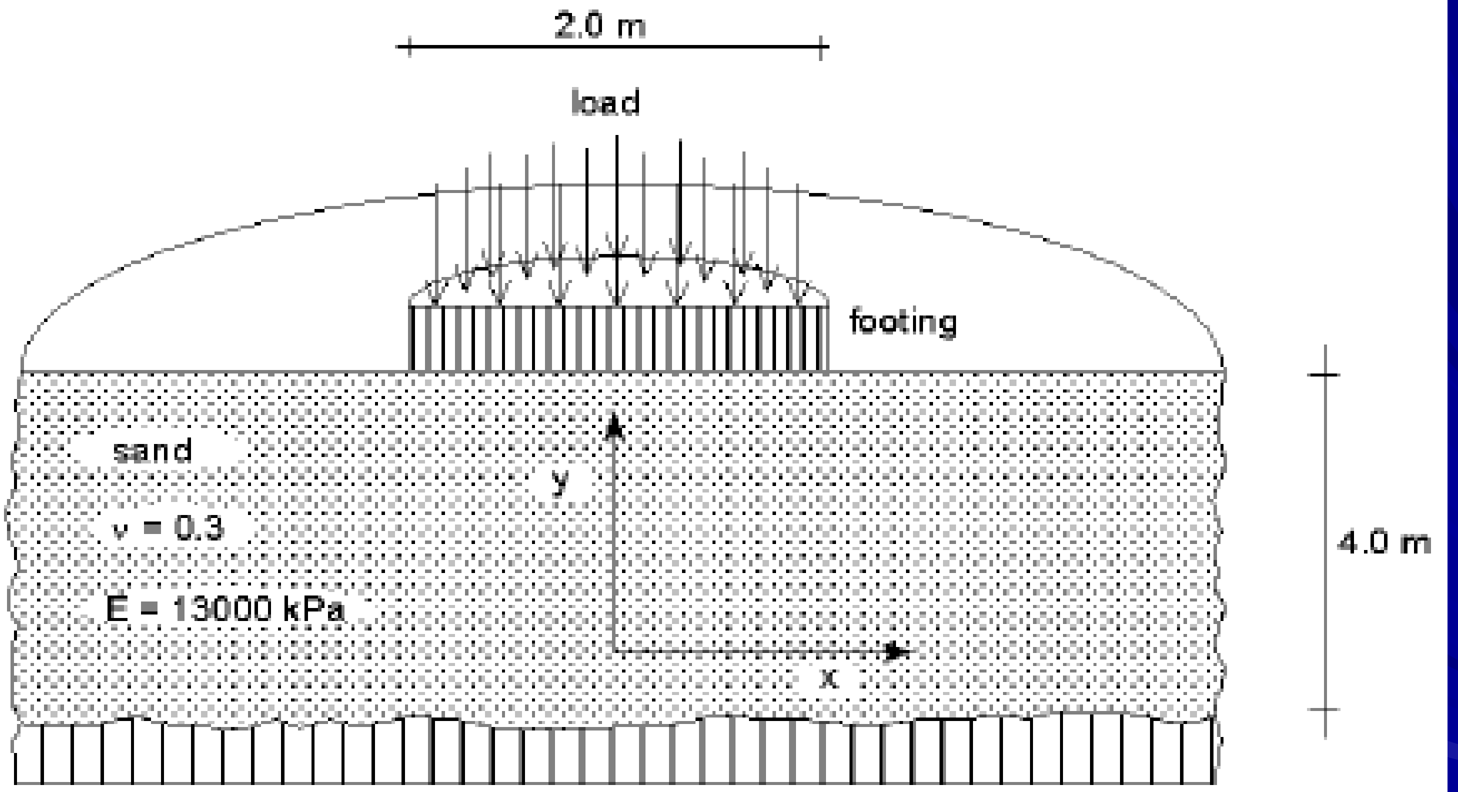
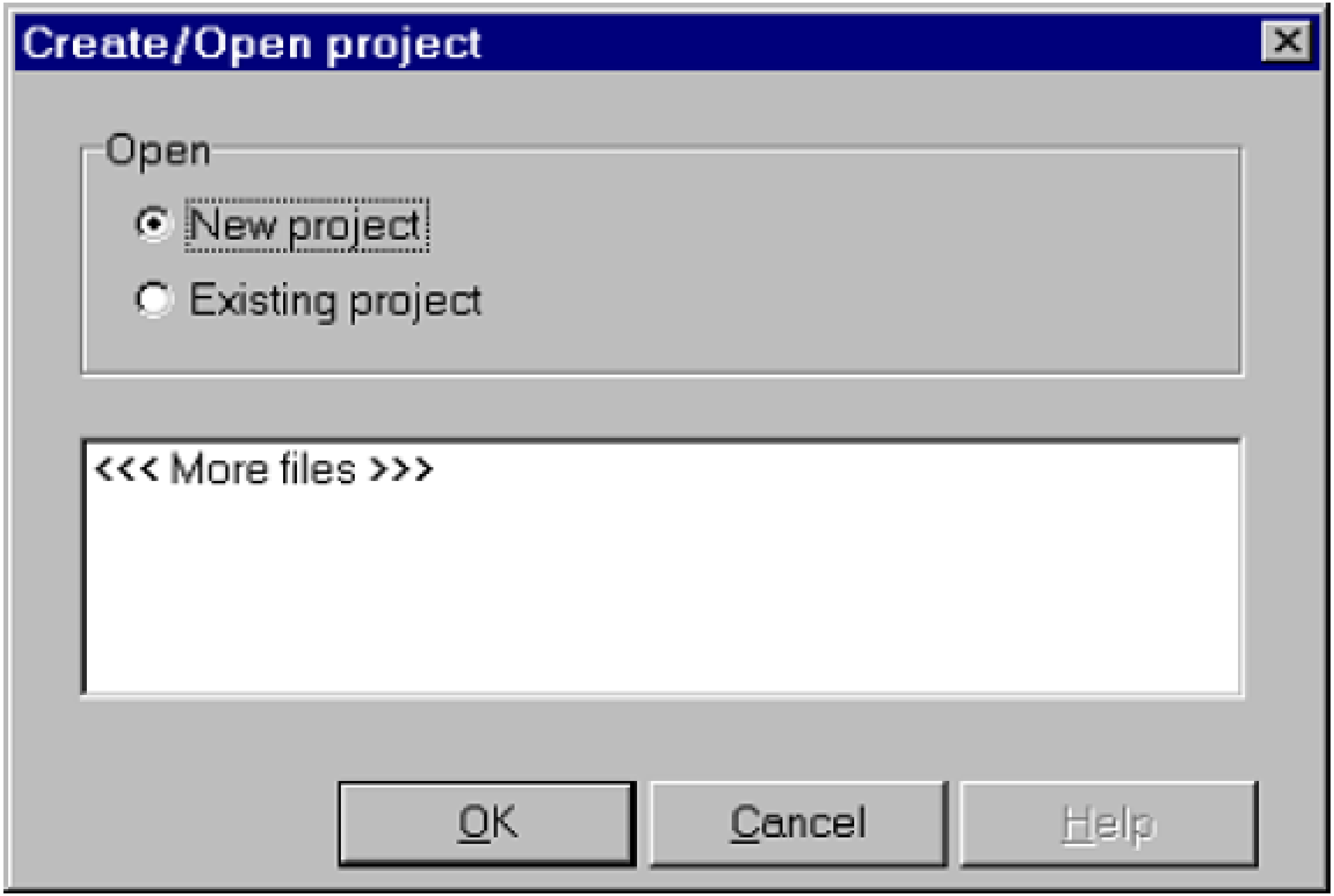


Figure 3.1 Geometry of a circular footing on a sand layer


Create/Open project
Open
New project
Existing project
<<< More files >>>
OK
Cancel
Help

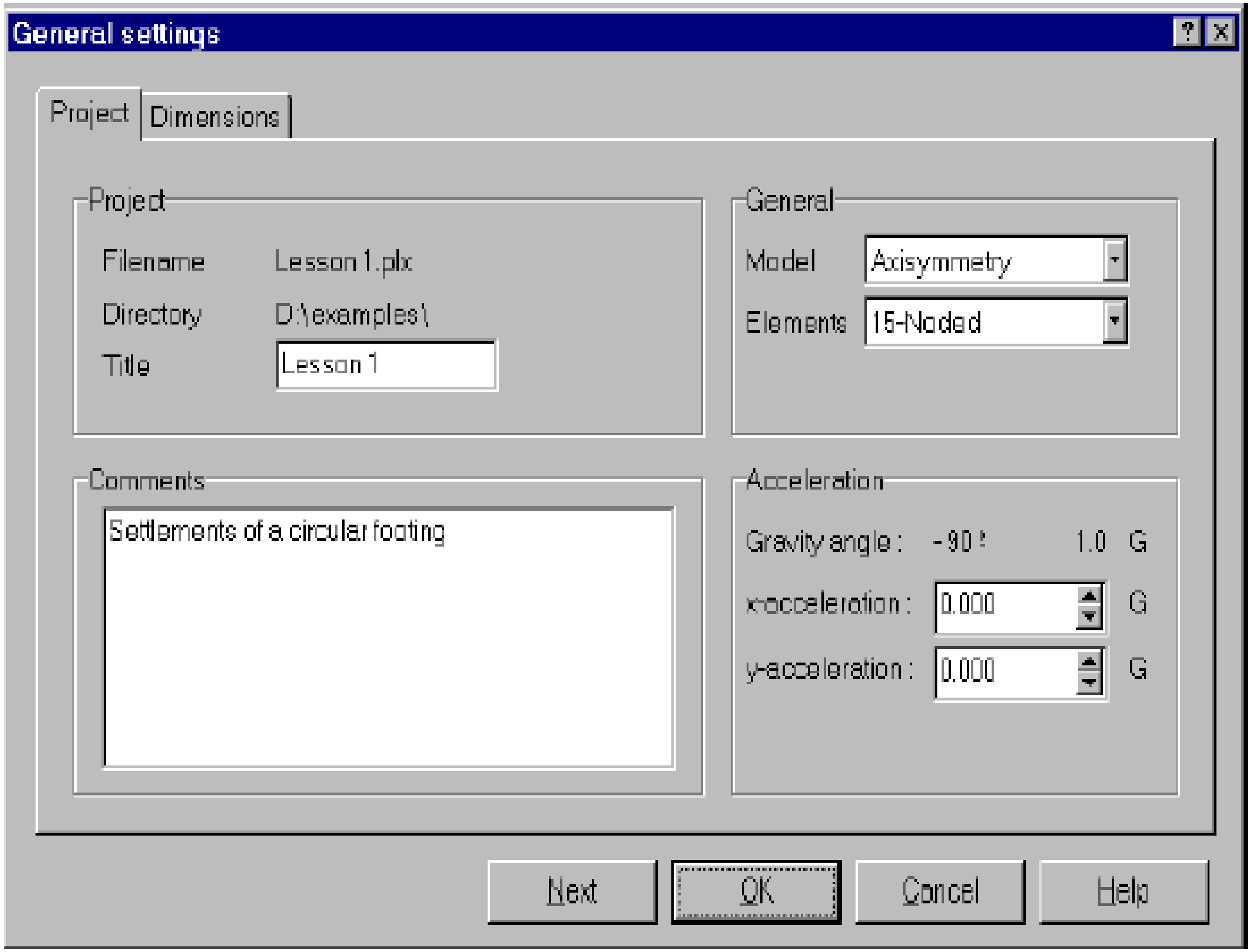


Figure 3.3 *Project* tab sheet of the *General settings* window

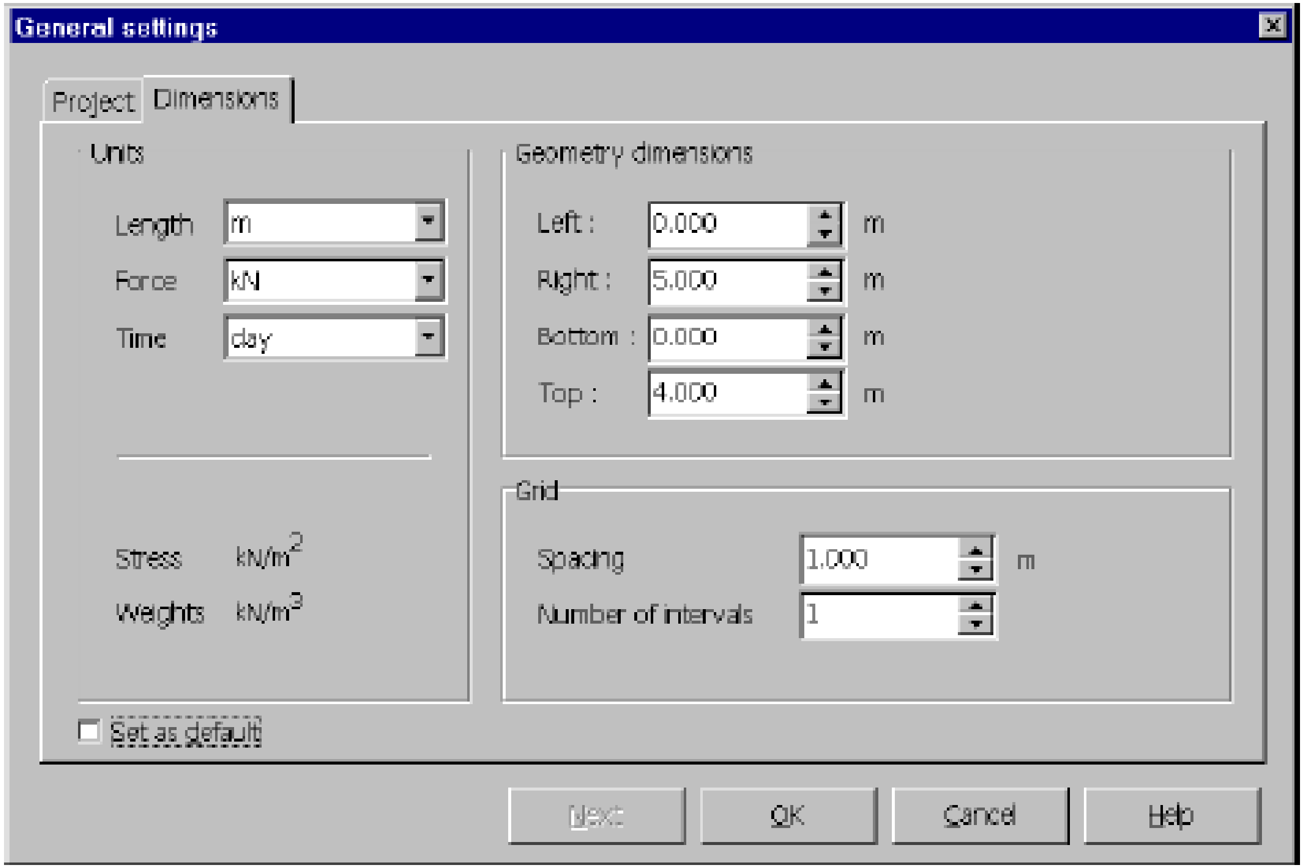
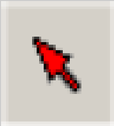
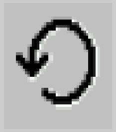


Figure 3.4 *Dimensions* tab sheet of the *General settings* window

Hint: In the case of a mistake or for any other reason that the general settings need to be changed, you can access the *General settings* window by selecting the *General settings* option from the *File* menu.

Hint: Mispositioned points and lines can be modified or deleted by first choosing the *Selection* button from the toolbar. To move a point or line, select the point or the line and drag it to the desired position. To delete a point or a line, select the point or the line and press the <Delete> button on the keyboard.





Unwanted drawing operations can be removed by pressing the *Undo* button from the toolbar or by selecting the *Undo* option from the *Edit* menu or by pressing <Ctrl><Z> on the keyboard.

> Lines can be drawn perfectly horizontal or vertical by holding down the <Shift> key on the keyboard while moving the cursor.

Hint: The full geometry model has to be completed before a finite element mesh can be generated. This means that boundary conditions and model parameters must be entered and applied to the geometry model first.

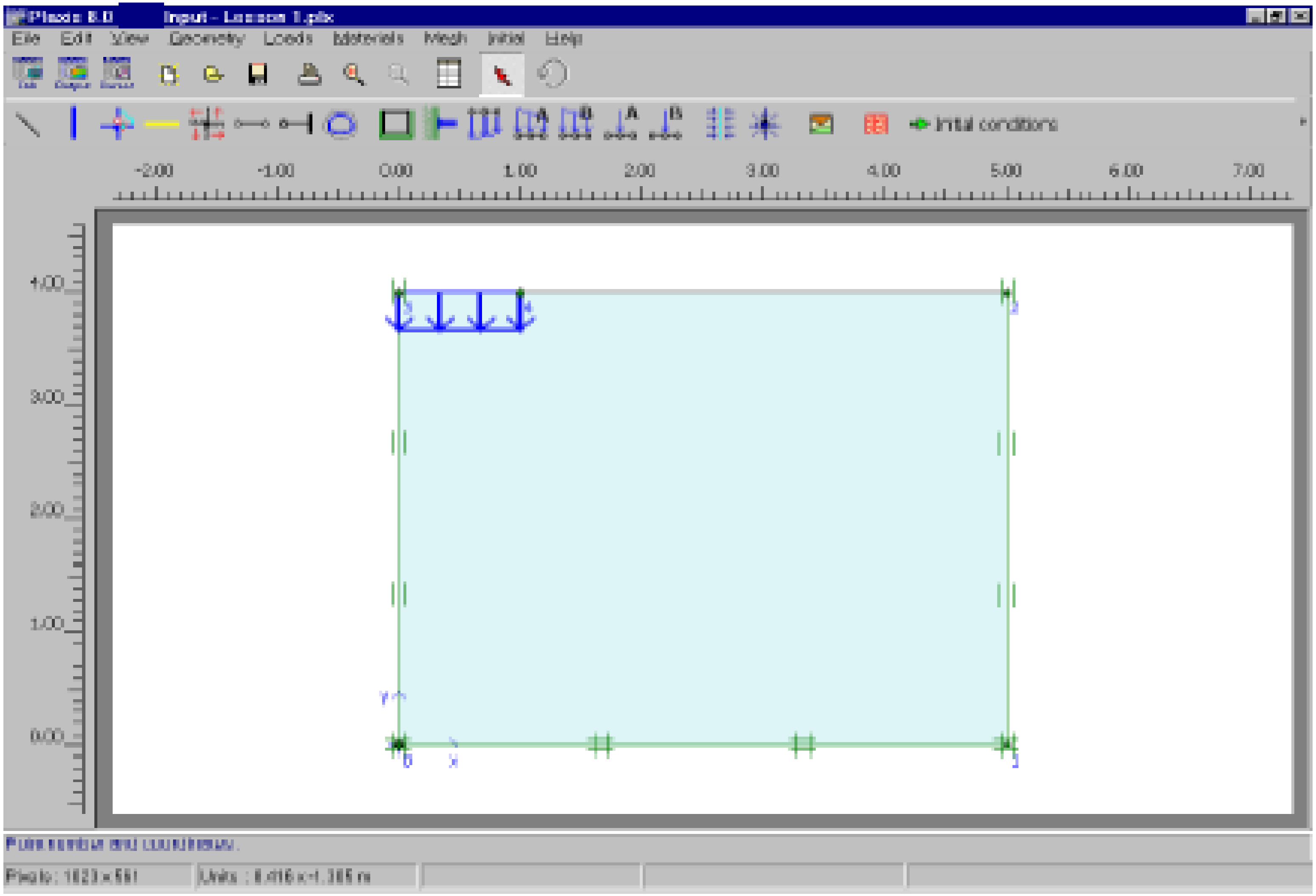


Figure 3.5 Geometry model in the Input window

Hint: The *Standard fixities* option is suitable for most geotechnical applications. It is a fast and convenient way to input standard boundary conditions.

Hint: The input value of a prescribed displacement may be changed by first clicking on the *Selection* button and then double-clicking on the line at which a prescribed displacement is applied. On selecting *Prescribed displacements* from the *Select* dialog box, a new window will appear in which the changes can be made.

> The prescribed displacement is actually activated when defining the calculation stages (Section 3.2.2). Initially it is not active.

Table 3.1 Material properties of the sand layer

Parameter	Name	Value	Unit
Material model	*Model*	Mohr-Coulomb	-
Type of material behaviour	*Type*	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	17.0	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20.0	kN/m^3

Parameter	Name	Value	Unit
Permeability in horizontal direction	k_x	1.0	m/day
Permeability in vertical direction	k_y	1.0	m/day
Young's modulus (constant)	E_{ref}	13000	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion (constant)	c_{ref}	1.0	kN/m^2
Friction angle	φ	31.0	°
Dilatancy angle	ψ	0.0	°

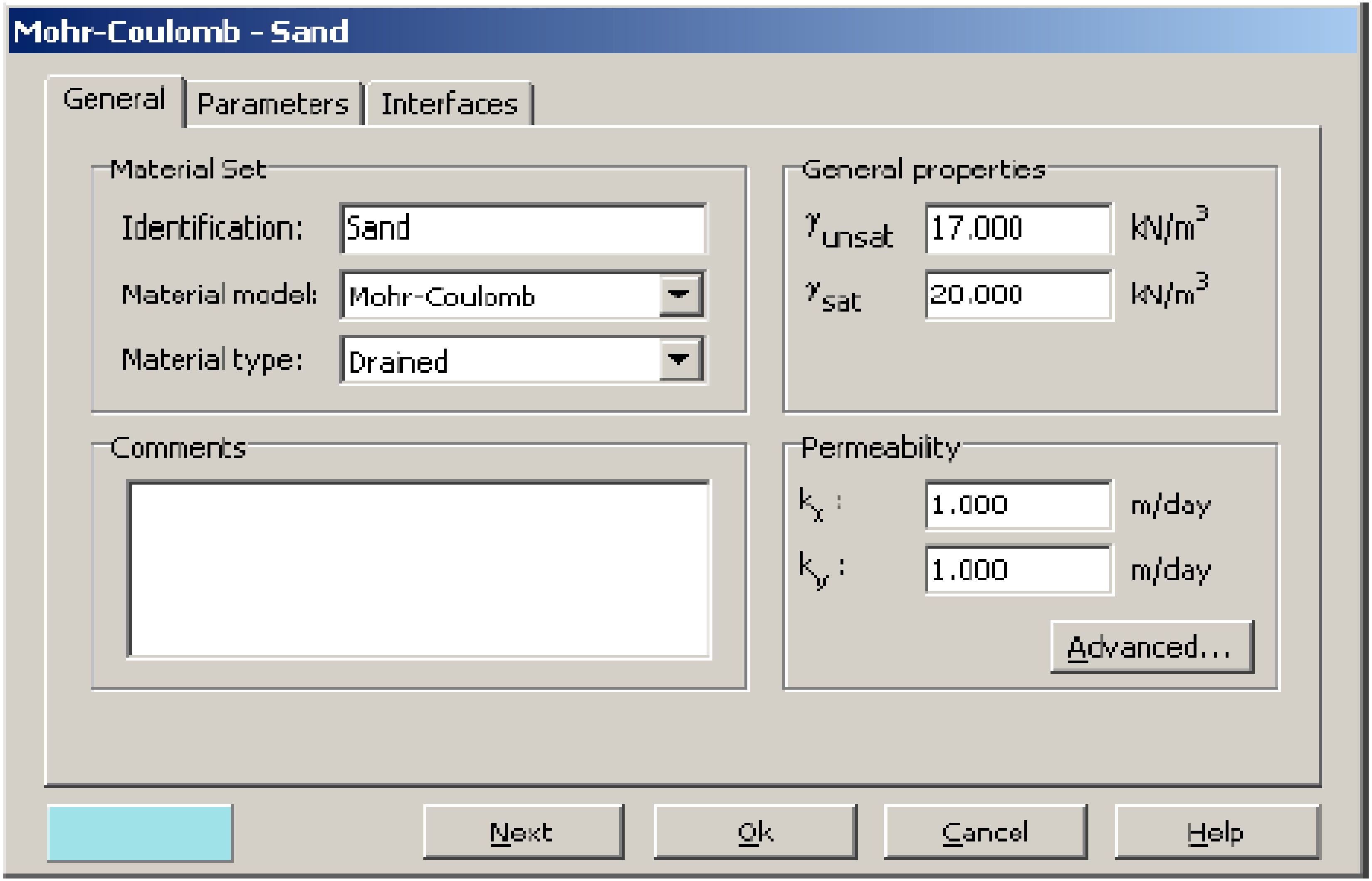


Figure 3.6 *General* tab sheet of the soil and interface data set window

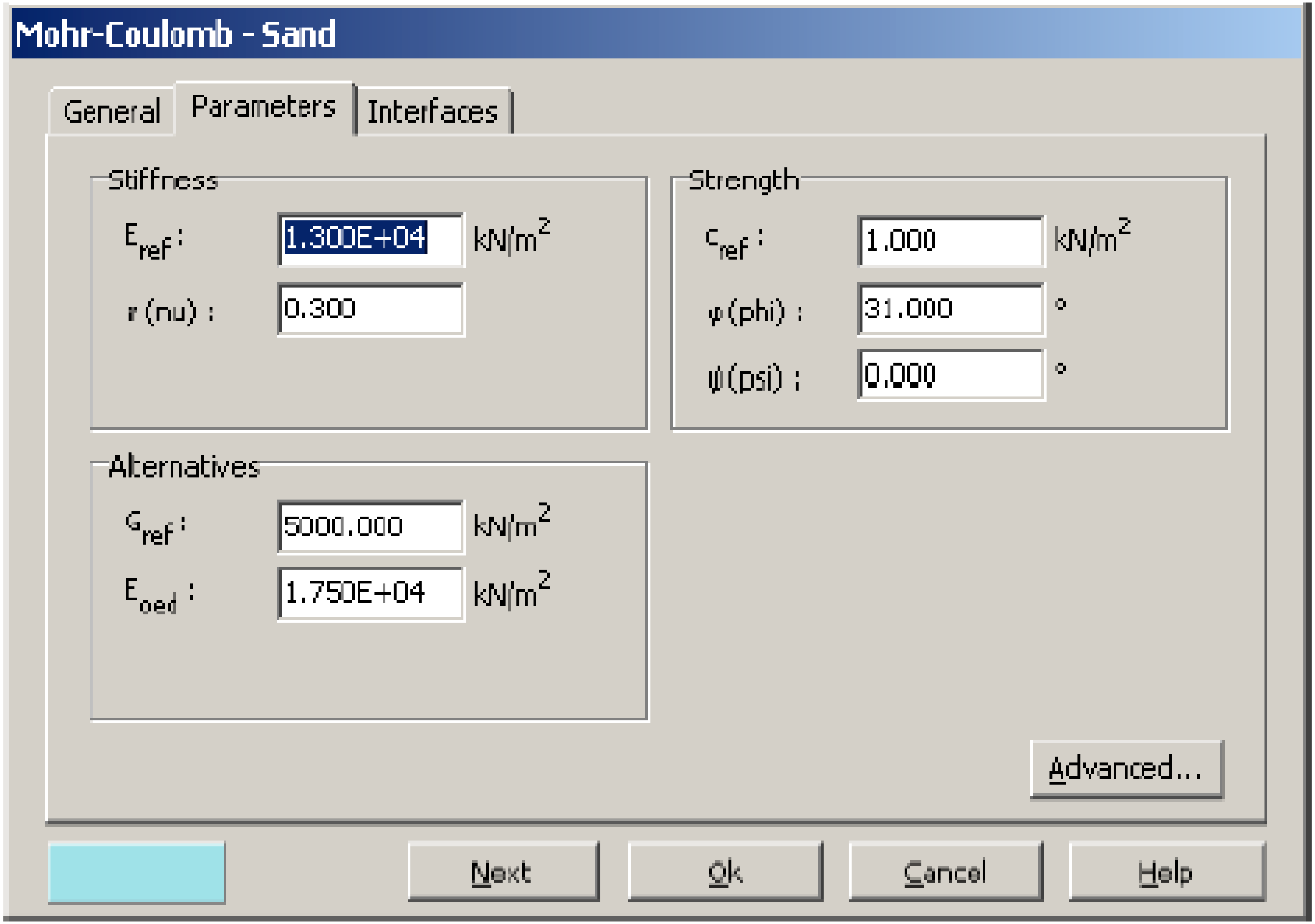


Figure 3.7 *Parameters* tab sheet of the soil and interface data set window

Hint: PLAXIS 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 data sets of all lessons in this Tutorial Manual are stored in the global database during the installation of the program. To copy an existing data set, click on the <Global> >>> button of the *Material Sets* window. Drag the appropriate data set (in this case "Lesson 1 sand") from the tree view of the global database to the project database and drop it there. Now the global data set is available for the current project. Similarly, data sets created in the project database may be dragged and dropped in the global database.

Hint: 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 on the <Edit> button. As an alternative, the material sets window can be opened by double clicking a cluster and clicking on the <Change> button behind the *Material set* box in the properties window. A data set can now be assigned to the corresponding cluster by selecting it from the project database tree view and clicking on the <Apply> button.

> 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.

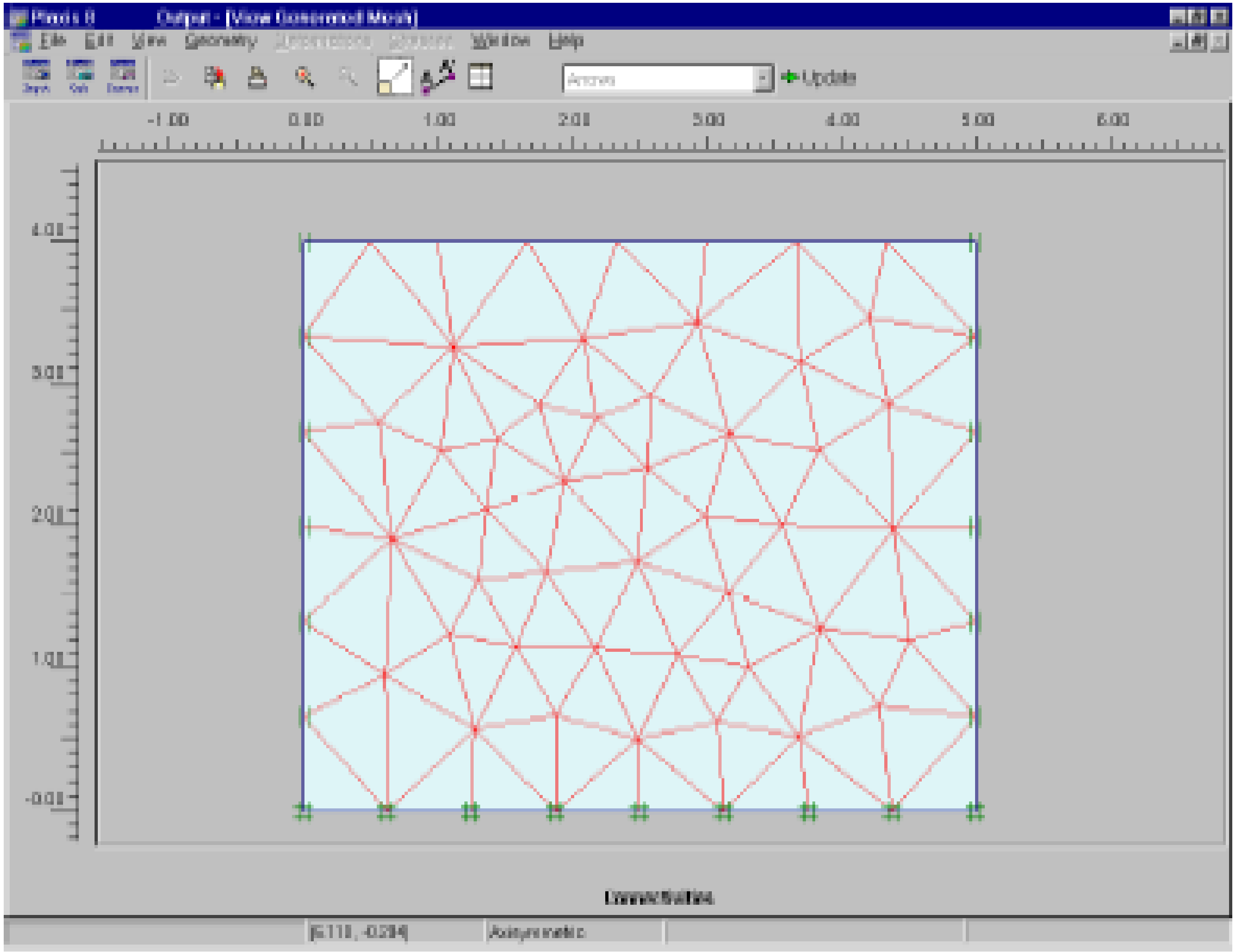


Figure 3.8 Axisymmetric finite element mesh of the geometry around the footing

Hint: The <Update> button must always be used to return to the geometry input, even if the result from the mesh generation is not satisfactory.

Hint: By default, the *Global coarseness* of the mesh is set to *Coarse*, which is adequate as a first approach in most cases. The *Global coarseness* setting can be changed in the *Mesh* menu. In additional options are available to refine the mesh globally or locally.

\> At this stage of input it is still possible to modify parts of the geometry or to add geometry objects. If modifications are made at this stage, then the finite element mesh has to be regenerated.

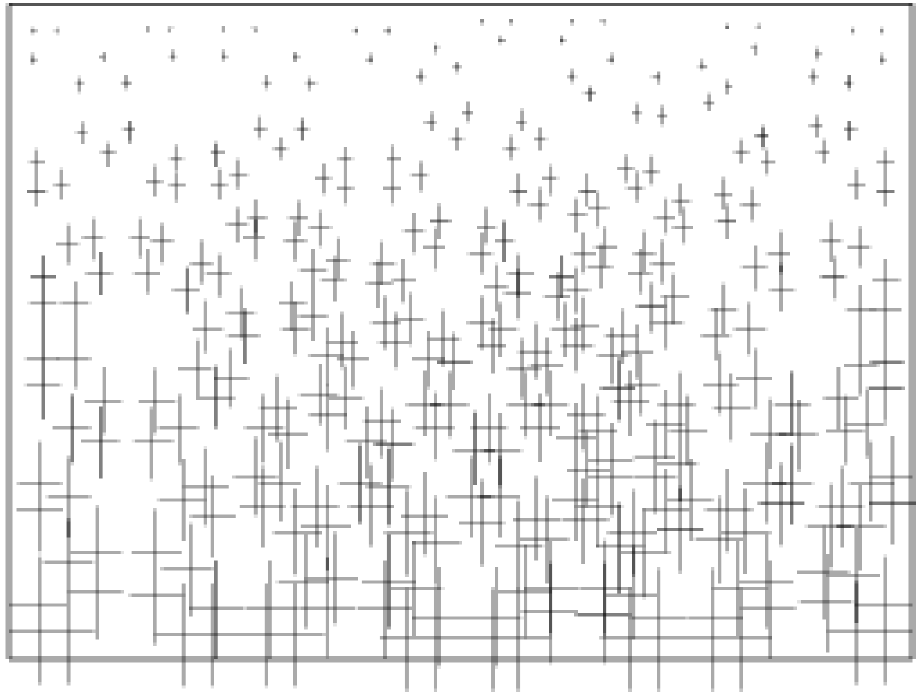


Figure 3.9 Initial stress field in the geometry around the footing

Hint: The *K_0-procedure* may only be used for horizontally layered geometries with a horizontal ground surface and, if applicable, a horizontal phreatic level. See Appendix A or the Reference Manual for more information on the *K_0-procedure*.

> The default value of K_0 is based on Jaky's formula: $K_0 = 1\text{-}\sin\varphi$. If the value was changed, the default value can be regained by entering a negative value for K_0.

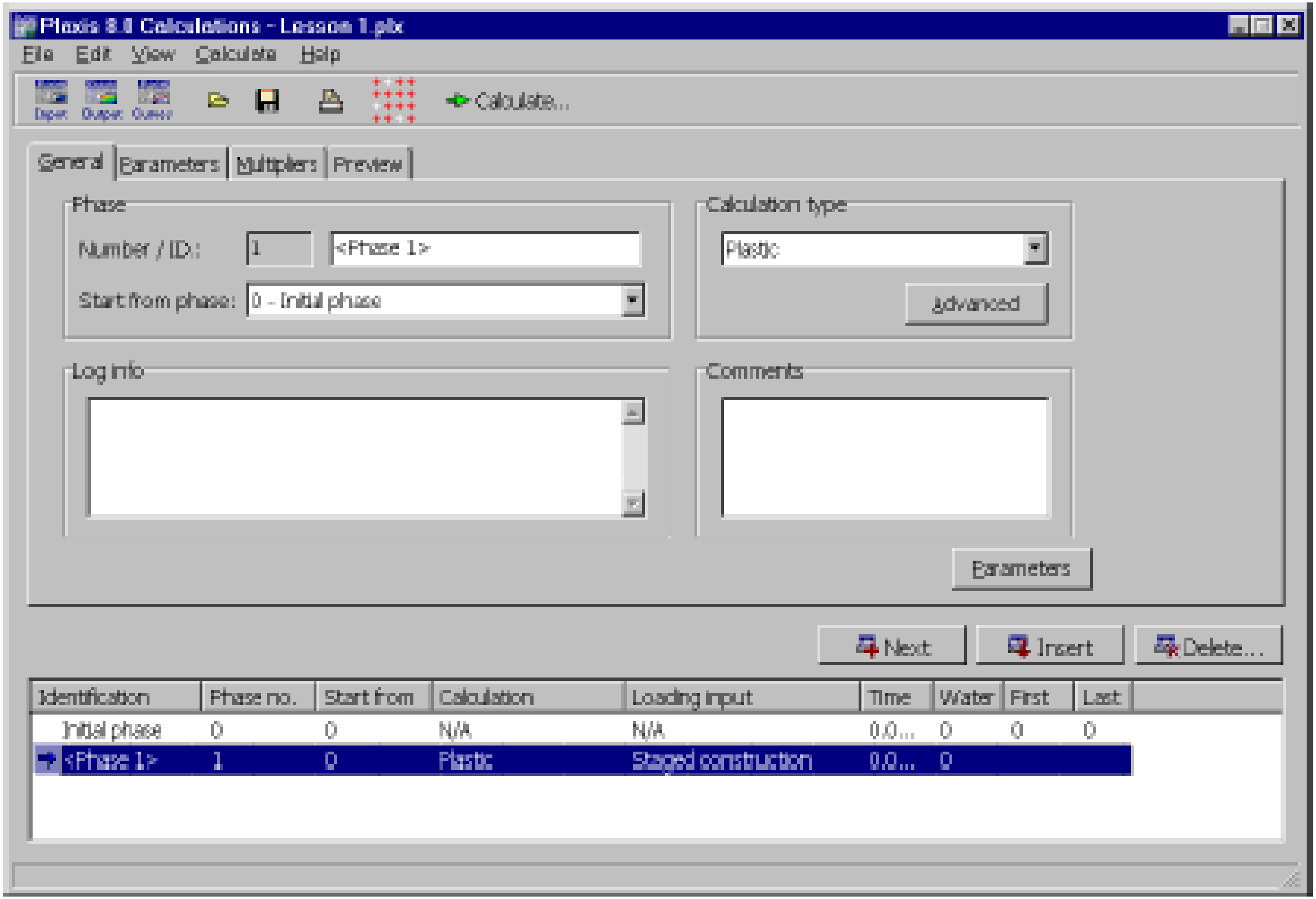


Figure 3.10 The *Calculations* window with the *General* tab sheet

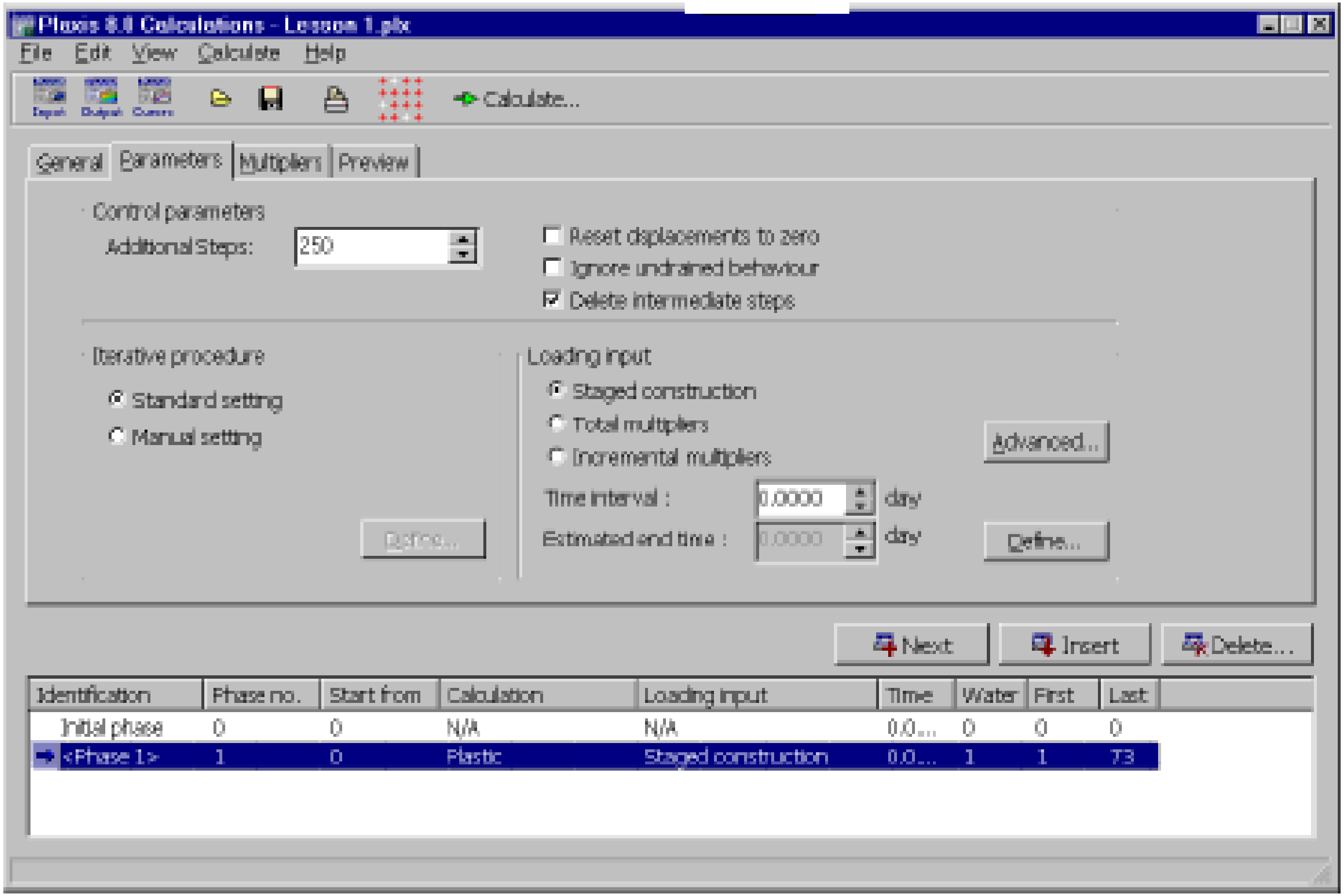


Figure 3.11 The *Calculations* window with the *Parameters* tab sheet

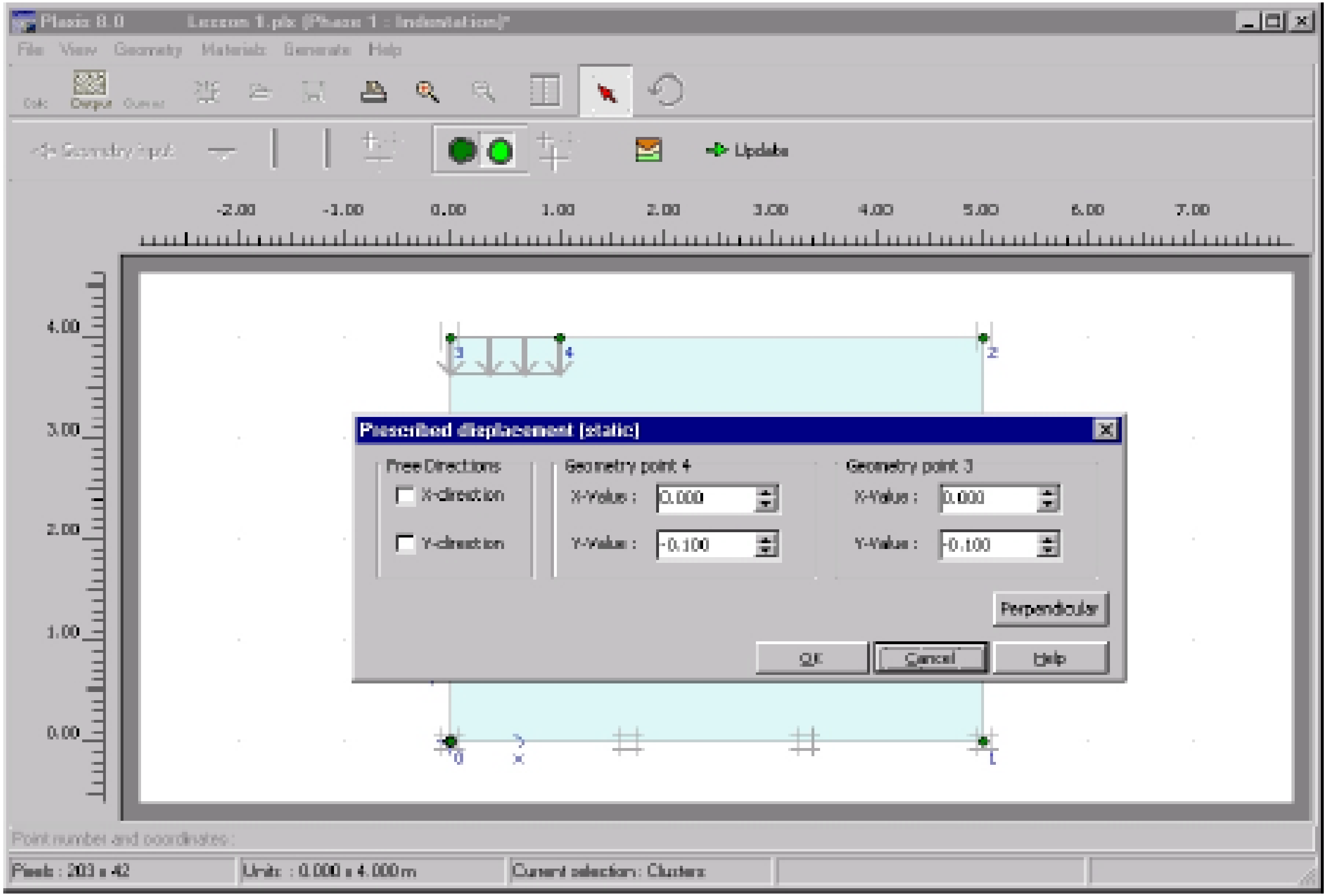


Figure 3.12 The *Prescribed Displacements* dialog box in the *Staged Construction* window

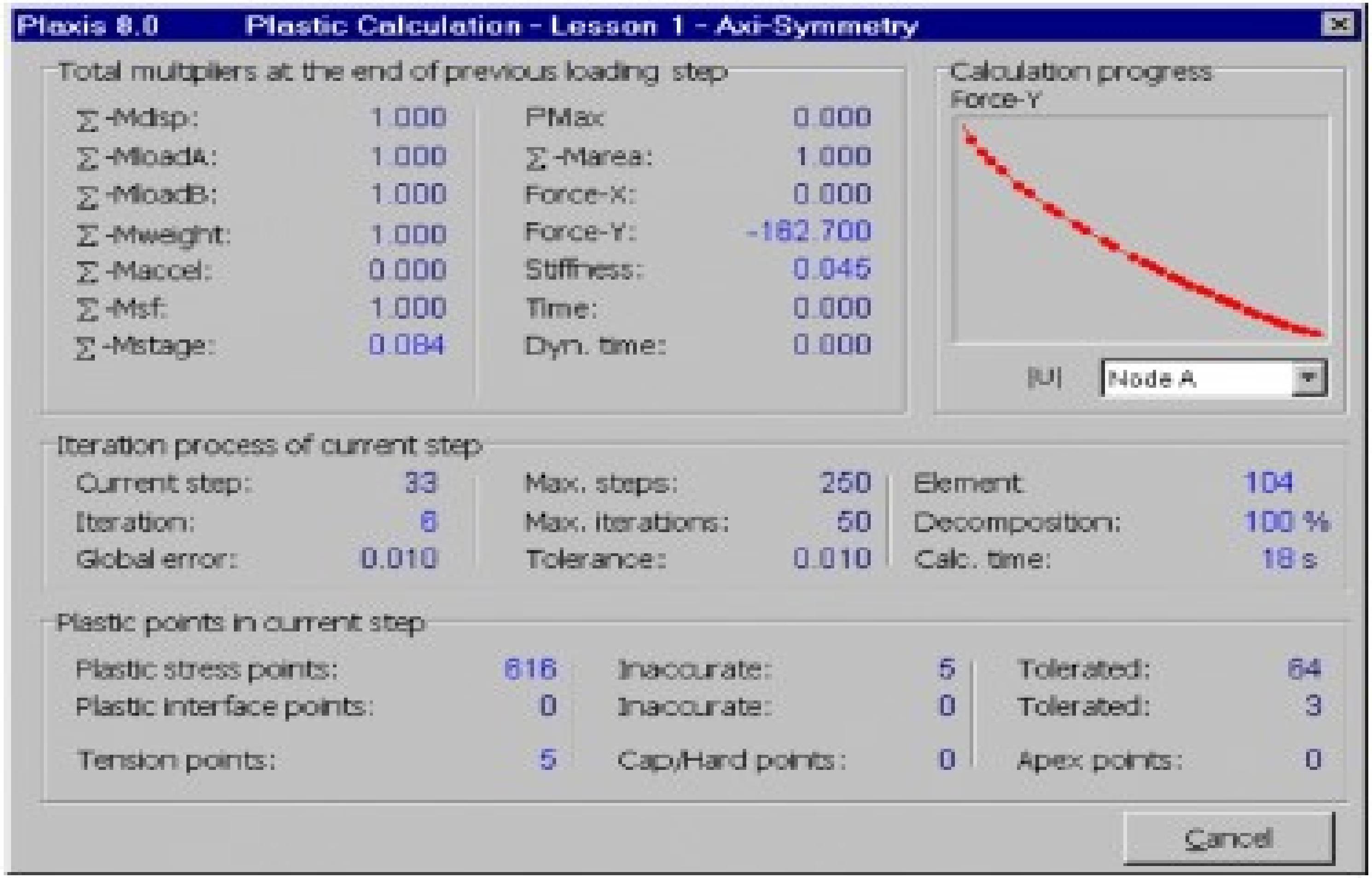


Figure 3.13 The calculations info window

Hint: The <Calculate> button is only visible if a calculation phase that is selected for execution is focused in the list.

Hint: Calculation phases may be added, inserted or deleted using the <Next>, <Insert> and <Delete> buttons half way the Calculations window.

> Check the list of calculation phases carefully after each execution of a (series of) calculation(s). A successful calculation is indicated in the list with a green check mark (√) whereas an unsuccessful calculation is indicated with a red cross (×). Calculation phases that are selected for execution are indicated by a blue arrow (→).

> When a calculation phase is focused that is indicated by a green check mark or a red cross, the toolbar shows the <Output> button, which gives direct access to the Output program. When a calculation phase is focused that is indicated by a blue arrow, the toolbar shows the <Calculate> button.

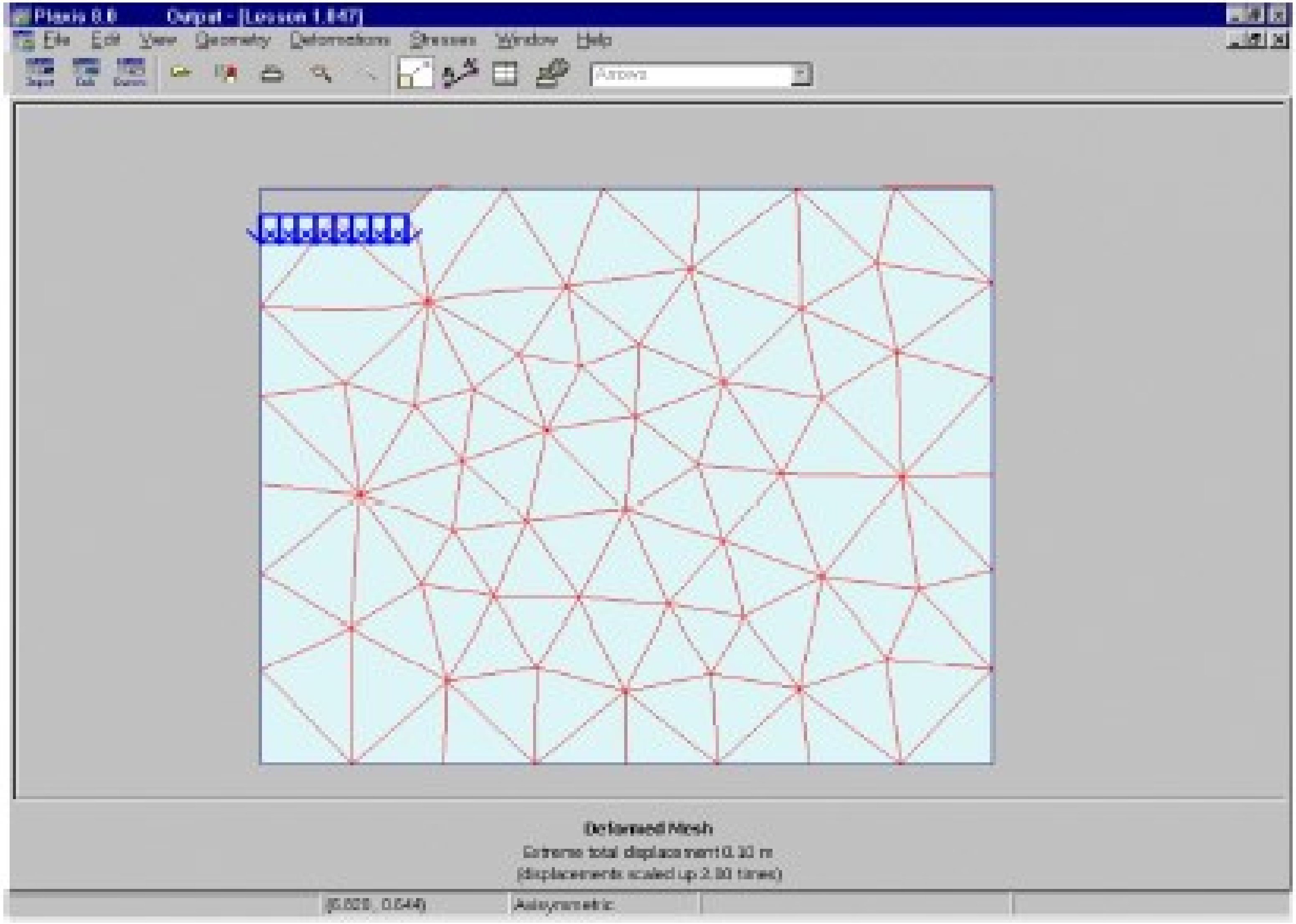


Figure 3.14 Deformed mesh

Hint: 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.

Hint: The plots of stresses and displacements may be combined with geometrical features, as available in the *Geometry* menu.

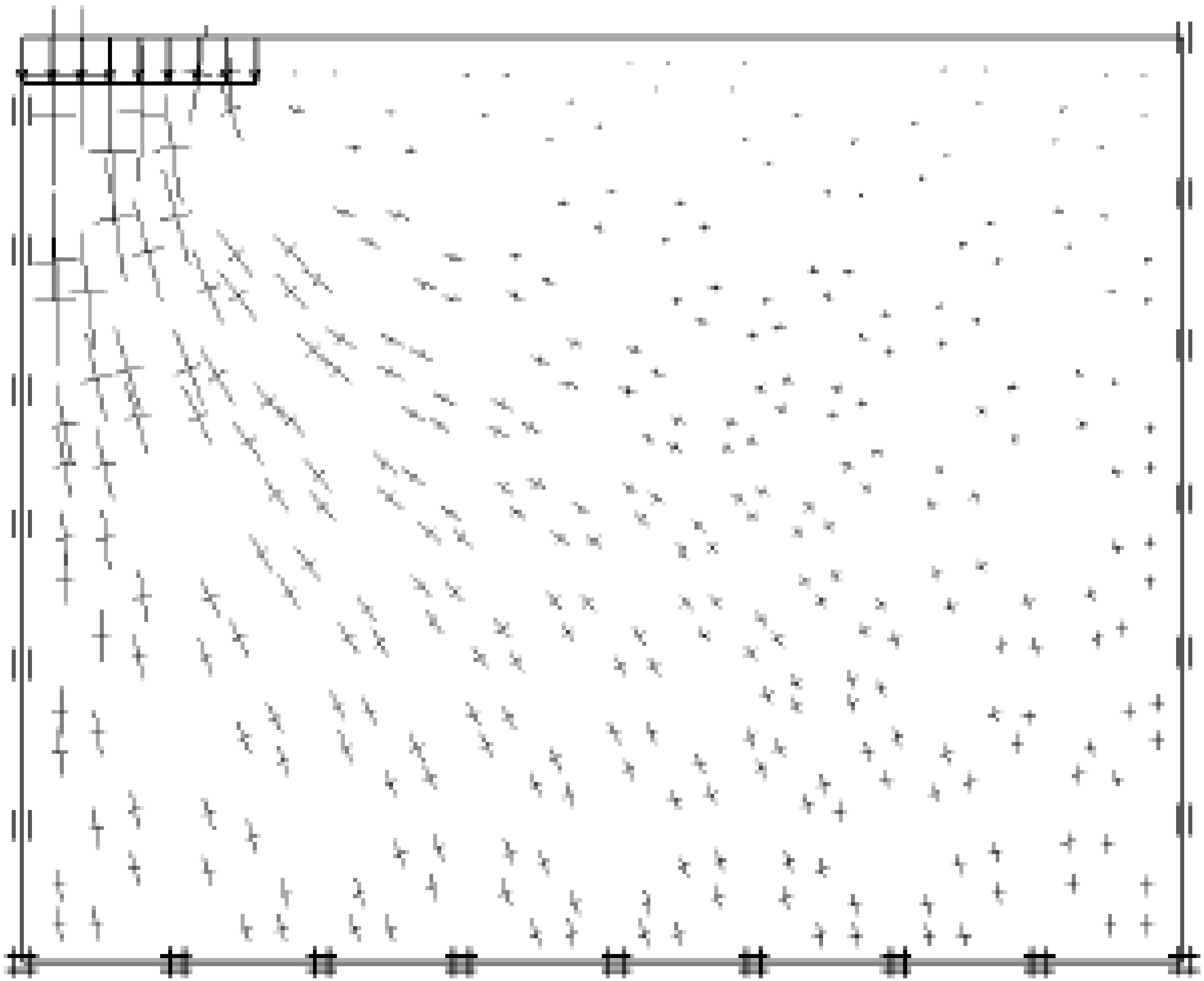


Figure 3.15 Principal stresses

Table 3.2. Material properties of the footing

Parameter	Name	Value	Unit
Normal stiffness	EA	$5 \cdot 10^6$	kN/m
Flexural rigidity	EI	8500	kNm^2/m
Equivalent thickness	d	0.143	m
Weight	w	0.0	kN/m/m
Poisson's ratio	ν	0.0	-

Hint: If the *Material Sets* window is displayed over the footing and hides it, move the window to another position so that the footing is clearly visible.

Hint: The equivalent thickness is automatically calculated by PLAXIS from the values of EA and EI. It cannot be entered by hand.

Hint: Regeneration of the mesh results in a redistribution of nodes and stress points. In general, existing stresses will not correspond with the new position of the stress points. Therefore it is important to regenerate the initial water pressures and initial stresses after regeneration of the mesh.

Hint: 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 follows the Windows standard for the presentation of sub-windows (*Cascade*, *Tile*, *Minimize*, *Maximize*, etc). See your Windows manual for a description of these standard possibilities.

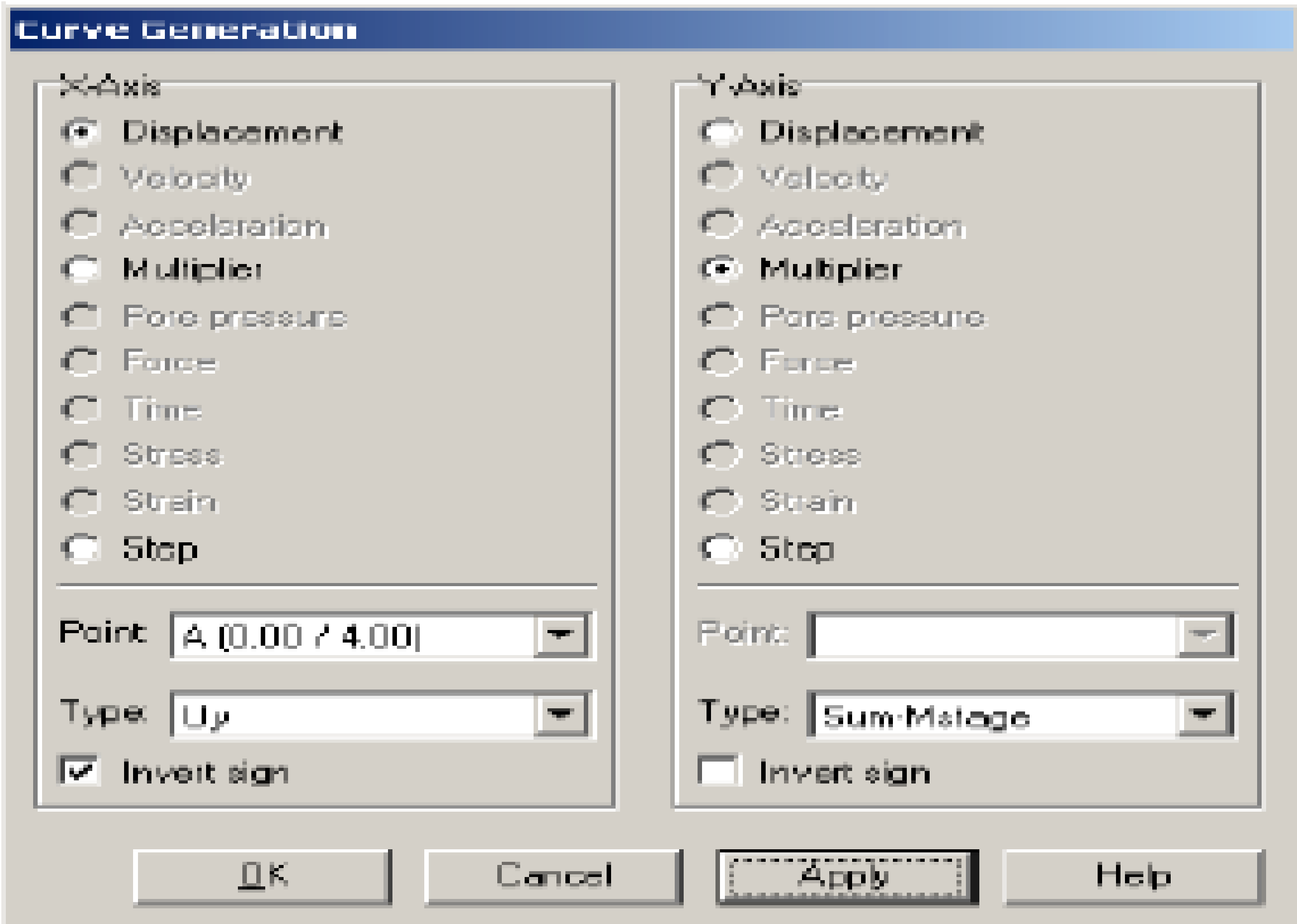


Figure 3.16 Curve generation window

Hint: The *Curve settings* window may also be used to modify the attributes or presentation of a curve.

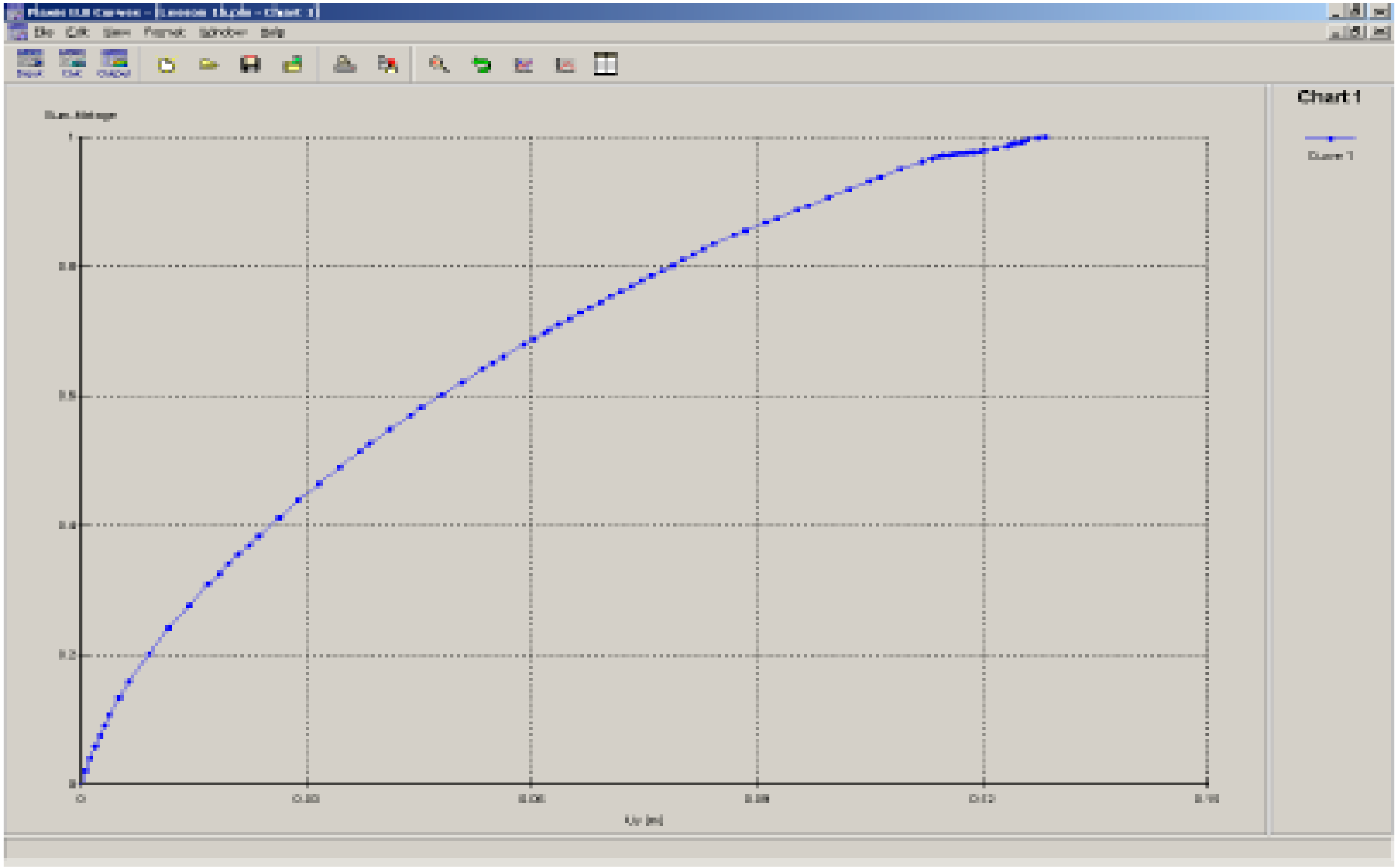


Figure 3.17 Load-displacement curve for the footing

Hint: To re-enter the *Curve generation* window (in the case of a mistake, a desired regeneration or modification) you can click on the *Change curve settings* button from the toolbar. As a result the *Curve settings* window appears, on which you should click on the <Regenerate> button. Alternatively, you may open the *Curve settings* window by selecting the *Curve* option from the *Format* menu.

> The *Frame settings* window may be used to modify the settings of the frame. This window can be opened by clicking on the *Change frame settings* button from the toolbar or selecting the *Frame* option from the *Format* menu.

LESSON 2

SUBMERGED CONSTRUCTION OF AN EXCAVATION

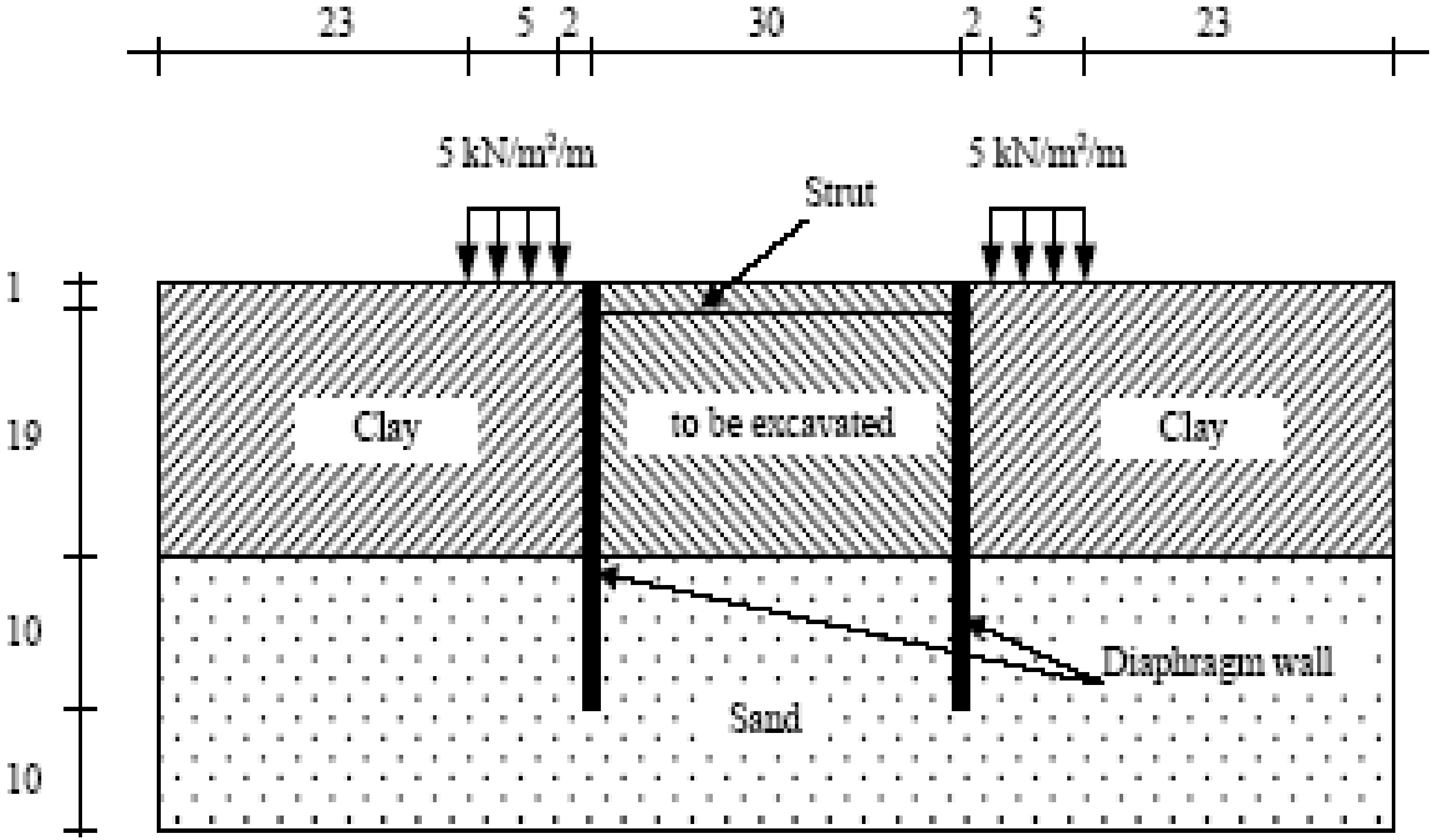


Figure 4.1 Geometry model of the situation of a submerged excavation

Hint: Within the geometry input mode it is not strictly necessary to select the buttons in the toolbar in the order that they appear from left to right. In this case, it is more convenient to create the wall first and then enter the separation of the excavation stages by means of a *Geometry line*.

> When creating a point very close to a line, the point is usually snapped onto the line, because the mesh generator cannot handle non-coincident points and lines at a very small distance. This procedure also simplifies the input of points that are intended to lie exactly on an existing line.

> If the pointer is substantially mispositioned and instead of snapping onto an existing point or line a new isolated point is created, this point may be dragged (and snapped) onto the existing point or line by using the *Selection* button.

> 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. The procedure to drag points onto existing points may be used to eliminate redundant points (and lines).

Hint:	The selection of an interface is done by selecting the corresponding geometry line and subsequently selecting the corresponding interface (positive or negative) from the *Select* dialog box.

Hint:	Interfaces are indicated as dotted lines along a geometry line. 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.

Hint:	A fixed-end anchor is represented by a rotated T with a fixed size. This object is actually a spring of which one end is connected to the mesh and the other end is fixed. The orientation angle and the equivalent length of the anchor must be directly entered in the properties window. 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. By default, the equivalent length is 1.0 unit and the angle is zero degrees (i.e. the anchor points in the positive x-direction).
>	Clicking on the 'middle bar' of the corresponding T selects an existing fixed-end anchor.

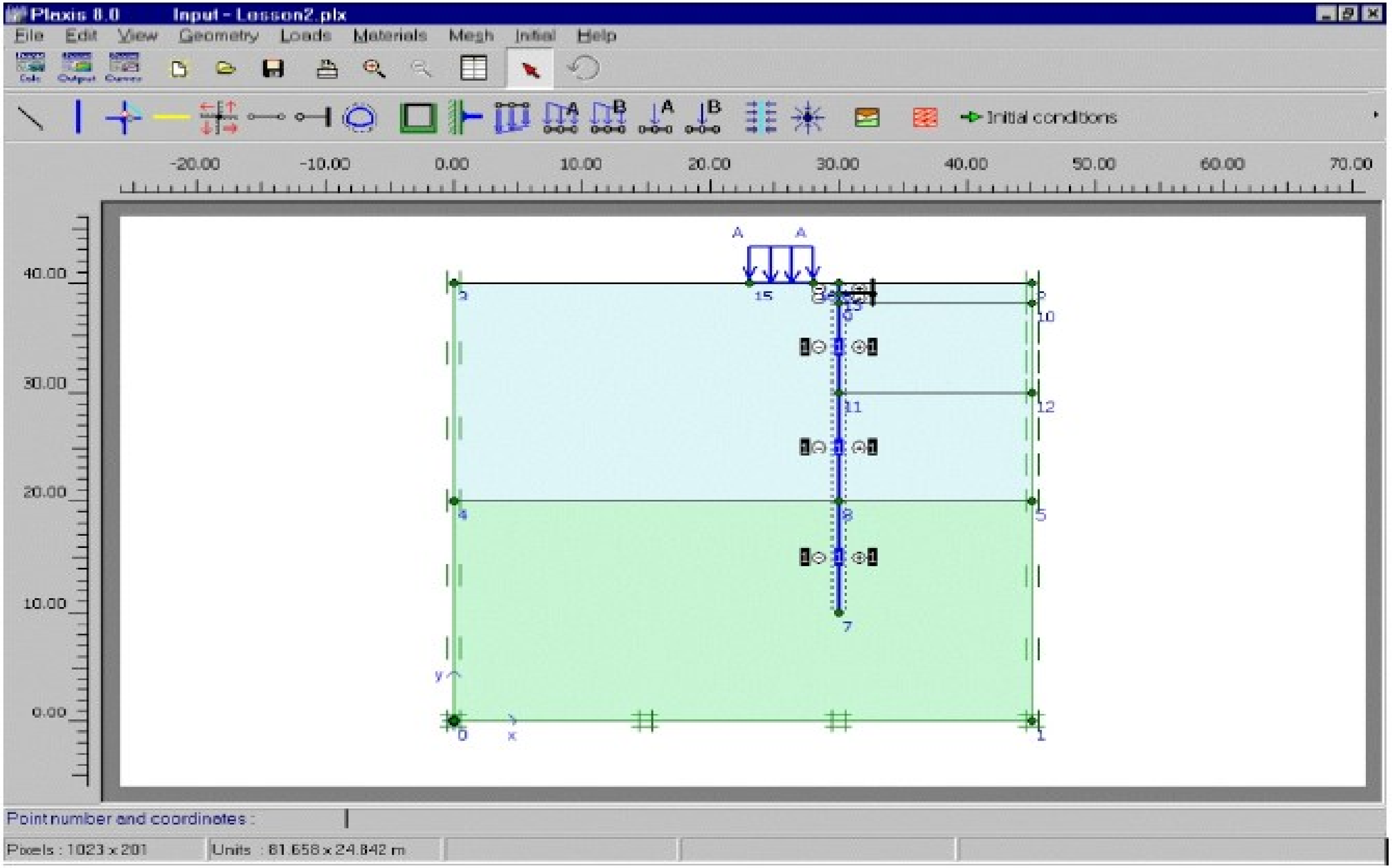


Figure 4.2 Geometry model in the Input window

Table 4.1. Material properties of the sand and clay layer and the interfaces

Parameter	Name	Clay layer	Sand layer	Unit
Material model	*Model*	Mohr-Coulomb	Mohr-Coulomb	-
Type of material behaviour	*Type*	Drained	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
Permeability in hor. direction	k_x	0.001	1.0	m/day
Permeability in ver. direction	k_y	0.001	1.0	m/day
Young's modulus (constant)	E_{ref}	10000	40000	kN/m^2
Poisson's ratio	ν	0.35	0.3	-
Cohesion (constant)	c_{ref}	5.0	1.0	kN/m^2
Friction angle	φ	25	32	°
Dilatancy angle	ψ	0.0	2.0	°
Strength reduction factor inter.	R_{inter}	0.5	0.67	-

Hint: Instead of accepting the default data sets of interfaces, data sets can directly be assigned to interfaces in their properties window. This window appears after double clicking the corresponding geometry line and selecting the appropriate interface from the *Select* dialog box. On clicking the <Change> button behind the *Material set* parameter, the proper data set can be selected from the *Material sets* tree view.

> In addition to the *Material set* parameter in the properties window, the *Virtual thickness factor* can be entered. This is a purely numerical value, which can be used to optimise the numerical performance of the interface. Non-experienced users are advised not to change the default value. For more information about interface properties see the Reference Manual.

Table 4.2. Material properties of the diaphragm wall (Plate)

Parameter	Name	Value	Unit
Type of behaviour	*Material type*	Elastic	
Normal stiffness	EA	$7.5 \cdot 10^6$	kN/m
Flexural rigidity	EI	$1.0 \cdot 10^6$	kNm^2/m
Equivalent thickness	d	1.265	m
Weight	w	10.0	kN/m/m
Poisson's ratio	ν	0.0	-

Hint: The radio button *Rigid* in the *Strength* box is a direct option for an interface with the same strength properties as the soil (R_{inter} = 1.0).

Table 4.3. 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_s	5.0	m
Maximum force	$F_{max,comp}$	$1 \cdot 10^{15}$	kN
	$F_{max,tens}$	$1 \cdot 10^{15}$	kN

Hint: The mesh settings are stored together with the rest of the input. On re-entering an existing project and not changing the geometry configuration and mesh settings, the same mesh can be regenerated by just clicking on the *Generate mesh* button on the toolbar. However, any slight change of the geometry will result in a different mesh.

> The *Reset all* option from the *Mesh* menu may be used to restore the default setting for the mesh generation (*Global coarseness* = *Coarse* and no local refinement).

Hint: When a project is newly created, the water weight is presented directly on entering the *Groundwater mode*. On re-entering an existing project the input of the water weight can be accessed by selecting the *Water weight* option from the *Geometry* menu in the *Groundwater mode*.

> To create an accurate pore pressure distribution in the geometry, an additional geometry line can be included corresponding with the level of the groundwater head or the position of the phreatic level in a problem.

Hint: An existing phreatic level may be modified using the *Selection* button from the toolbar. On deleting the *General* phreatic level (selecting it and pressing the <Del> key on the keyboard), the default general phreatic level will be created again at the bottom of the geometry. The graphical input or modification of phreatic levels does not affect the existing geometry.

Hint: Inactive clusters are white, just like the background, whereas active clusters have the colour of the corresponding material set. Inactive structural objects are grey, whereas active structures have the basic colour as used during the creation of the geometry model.

Hint: The *Staged construction* option is not only intended to simulate excavations or constructions, but it can also be used to change the water pressure distribution, to change material properties (to simulate soil improvement, for example) or to improve the accuracy of previous computational results.

Hint: You can also enter or change the values of the load at this time by double clicking on the load and entering a value. If a load is applied on a structural object such as a plate, load values can be changed by clicking on the load or the object. As a result a window appears in which you can select the load. Then click on the <Change> button to modify the load values.

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.

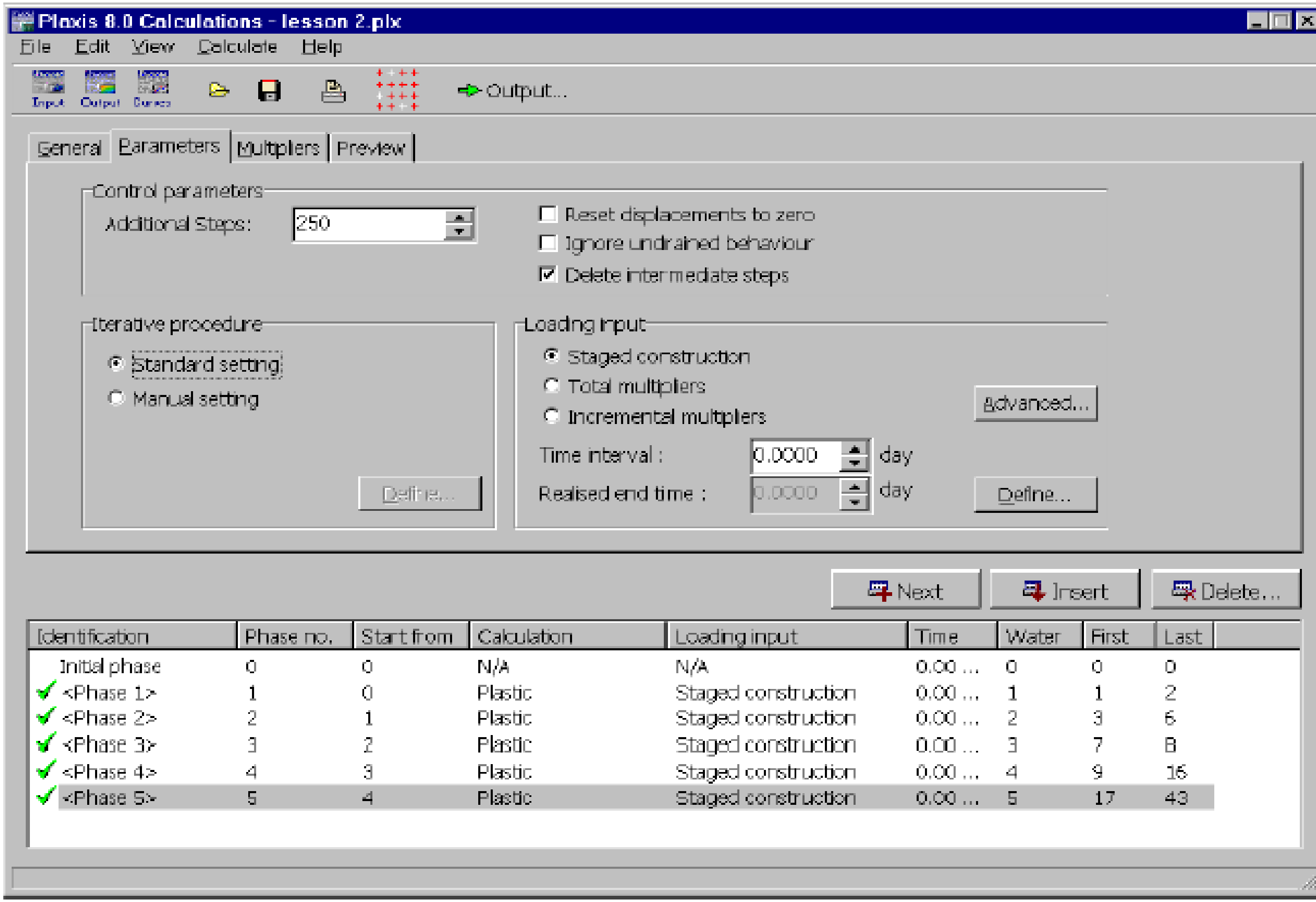


Figure 4.3 The *Calculations* window with the *Parameters* tab sheet

Hint: To select the desired nodes, it may be convenient to use the *Zoom in* option on the toolbar to zoom into the area of interest.

Hint: The *Staged construction* window is similar to the *Initial conditions* window of the Input program. The main difference is that *Initial conditions* is used to create an initial situation, whereas *Staged construction* is used as a type of loading.

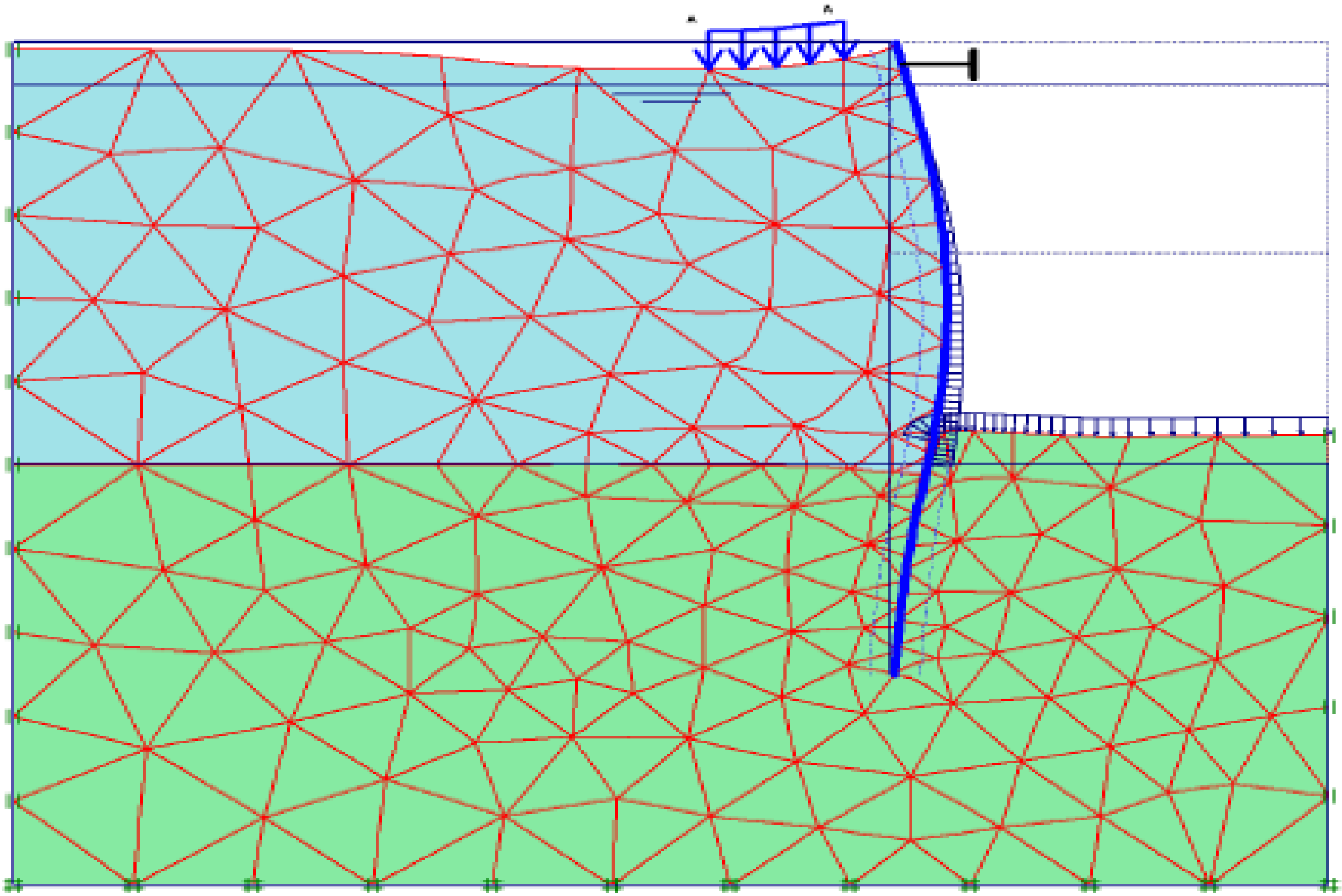


Figure 4.4 Deformed mesh after submerged excavation

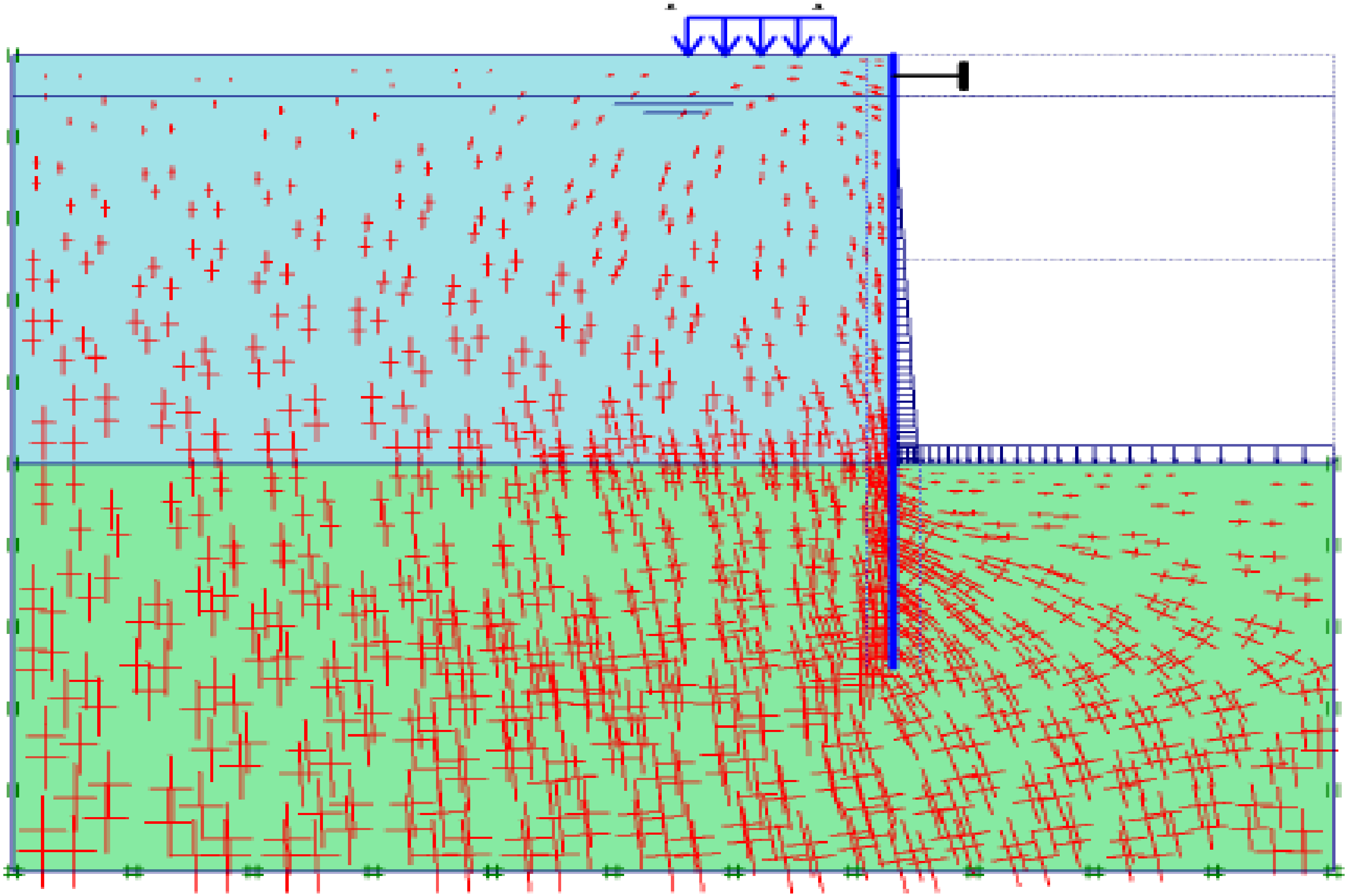


Figure 4.5 Principal stresses after excavation

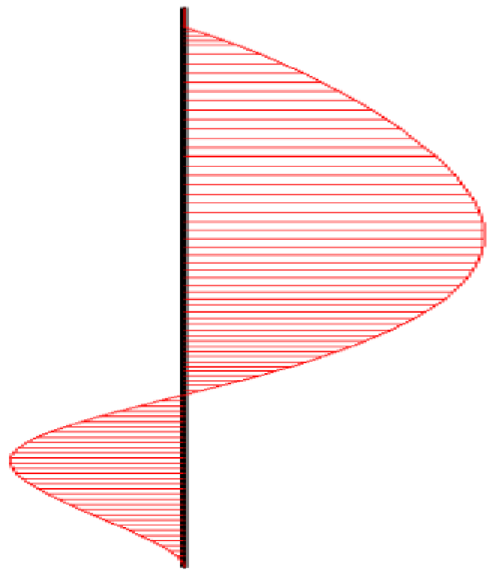


Figure 4.6 Bending moments in the wall

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.

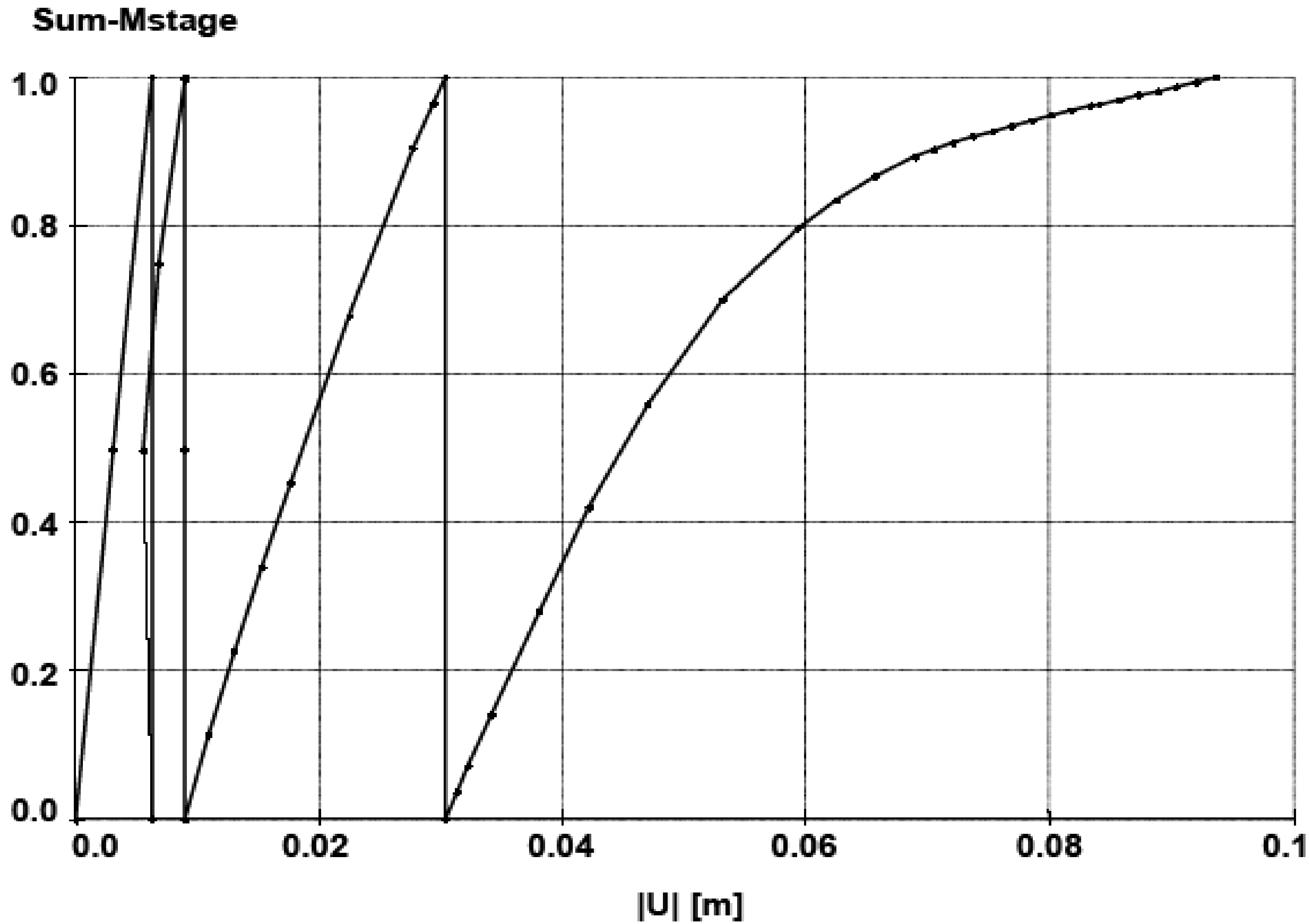


Figure 4.7 Load-displacement curve of deflection of wall

LESSON 3

UNDRAINED RIVER EMBEDMENT

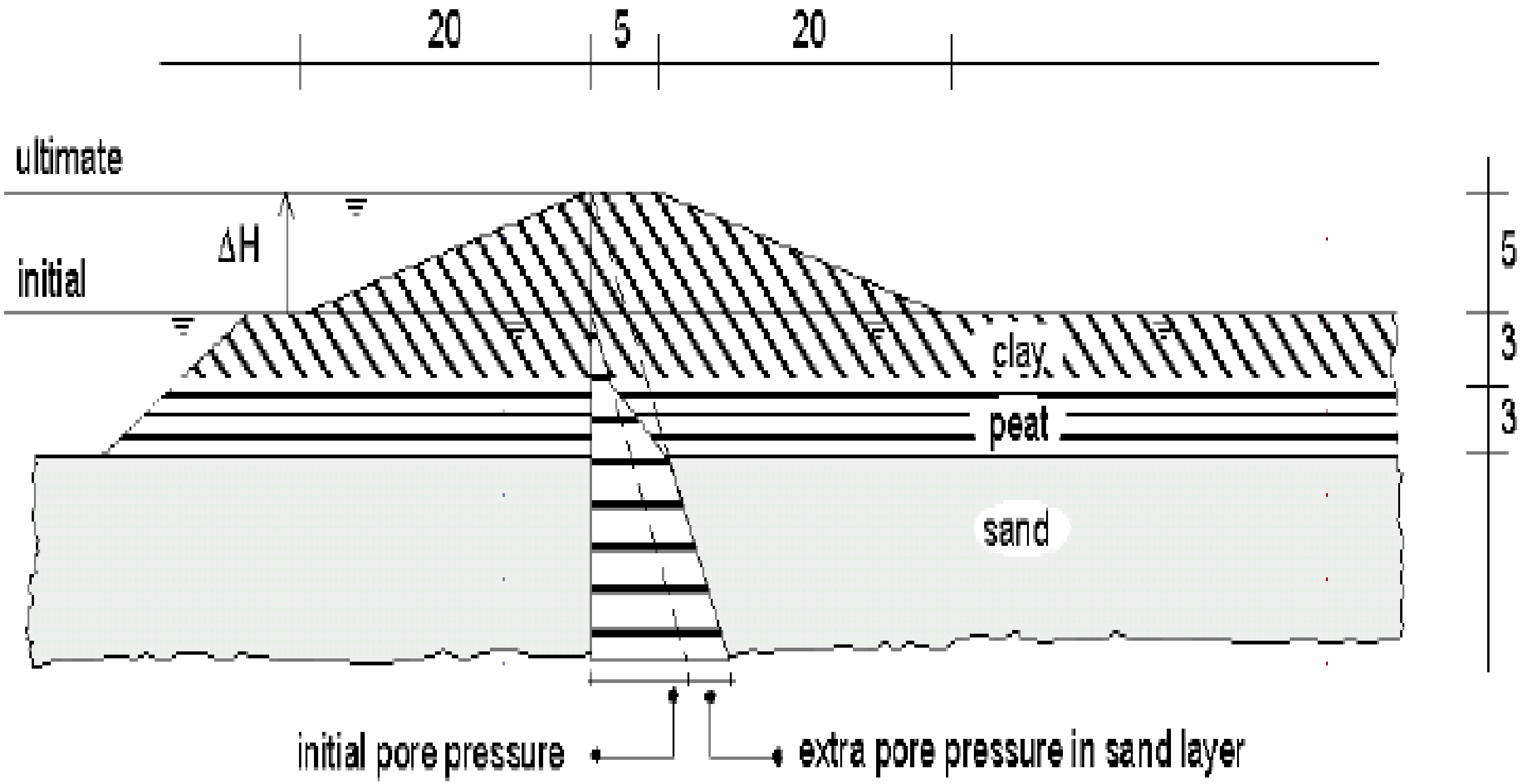


Figure 5.1 Geometry of the river embankment subjected to a changing water level

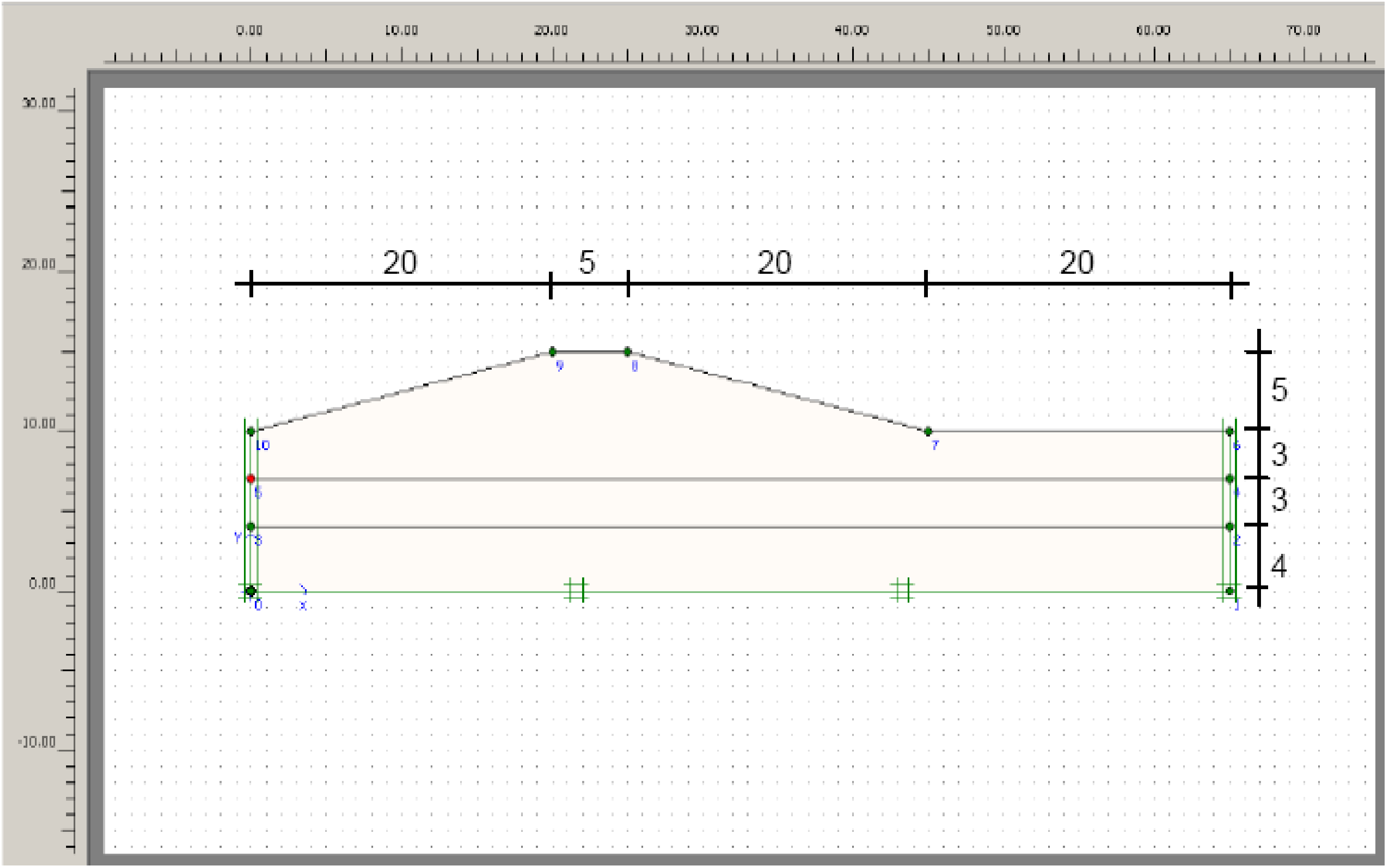


Figure 5.2 Geometry model of the river embankment project

Material sets

Three material layers are adopted for the soil. The properties are given in Table 5.1.

Table 5.1. Material properties of the river embankment and subsoil

Parameter	Name	Clay	Peat	Sand	Unit
Material model	*Model*	MC	MC	MC	-
Type of behaviour	*Type*	Undr.	Undr.	Drained	-
Soil unit weight above p.l.	γ_{unsat}	16	8	17	kN/m^3
Soil unit weight below p.l.	γ_{sat}	18	11.5	20	kN/m^3
Horizontal permeability	k_x	0.001	0.01	1.0	m/day
Vertical permeability	k_y	0.001	0.001	1.0	m/day
Young's modulus	E_{ref}	2000	500	20000	kN/m^2
Poisson's ratio	ν	0.35	0.35	0.3	-
Cohesion	c_{ref}	2.0	7.0	1.0	kN/m^2
Friction angle	φ	24	20	35	°
Dilatancy angle	ψ	0.0	0.0	0.0	°

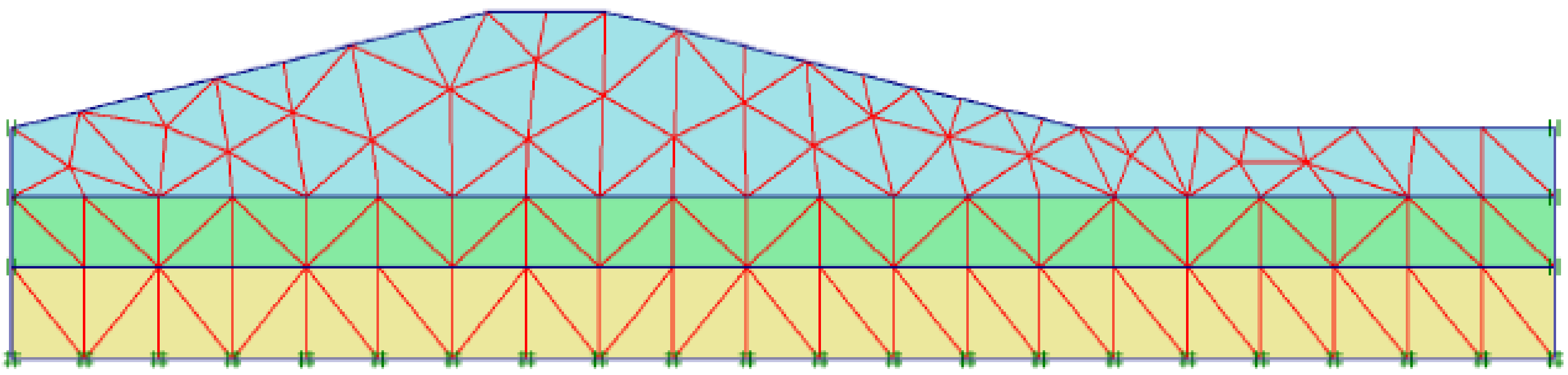


Figure 5.3 Finite element mesh of river embankment project

Hint: If initial stresses were generated by mistake, they can be reset by entering the K_0-*procedure*, entering a value of ΣMweight = 0 and pressing the <Generate> button.

Hint: Since the initial stresses are not subject to undrained behaviour, it is important that undrained behaviour is disabled during gravity loading. This can be done by selecting *Ignore undrained behaviour* in the *Parameters* tab sheet of the *Calculations* window.

> In contrast to the K_0-procedure, the calculation of initial stresses by means of gravity loading results in displacements. These displacements are not realistic, because the embankment is modelled as it appears in reality and the calculation of the initial stresses should not influence the displacements computed later in the analysis. These unrealistic displacements can be reset to zero at the start of the next calculation phase by selecting *Reset displacements to zero* in the next phase.

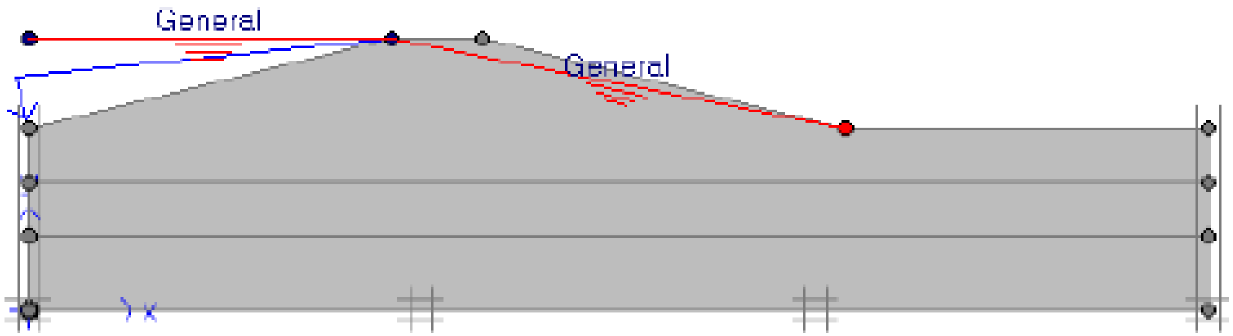


Figure 5.4 General phreatic level for generation of external water pressures

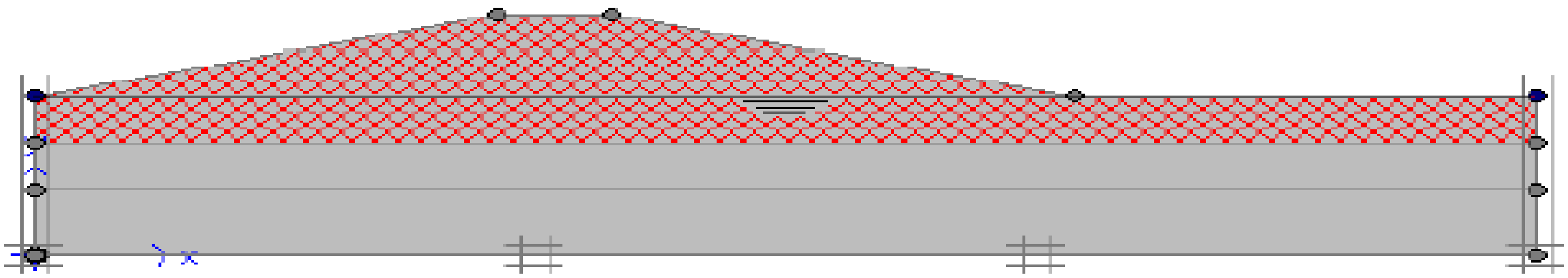


Figure 5.5 Phreatic level for clay layer

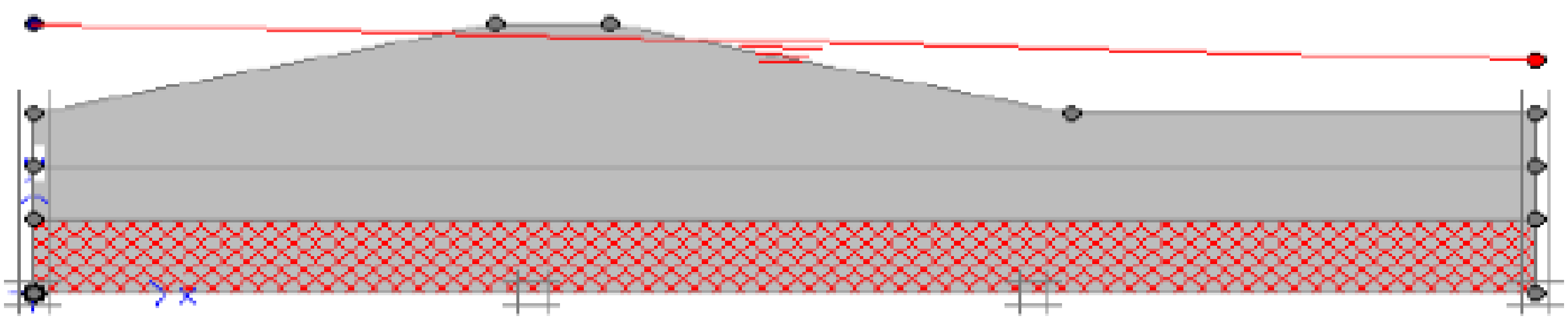


Figure 5.6 Phreatic level for sand layer

Hint: The phreatic level corresponding to a particular cluster is indicated in red as soon as the cluster is selected. Clicking outside the geometry results in an indication of the general phreatic level. If a cluster is selected where the *Interpolate...* option applies, no phreatic level is indicated.

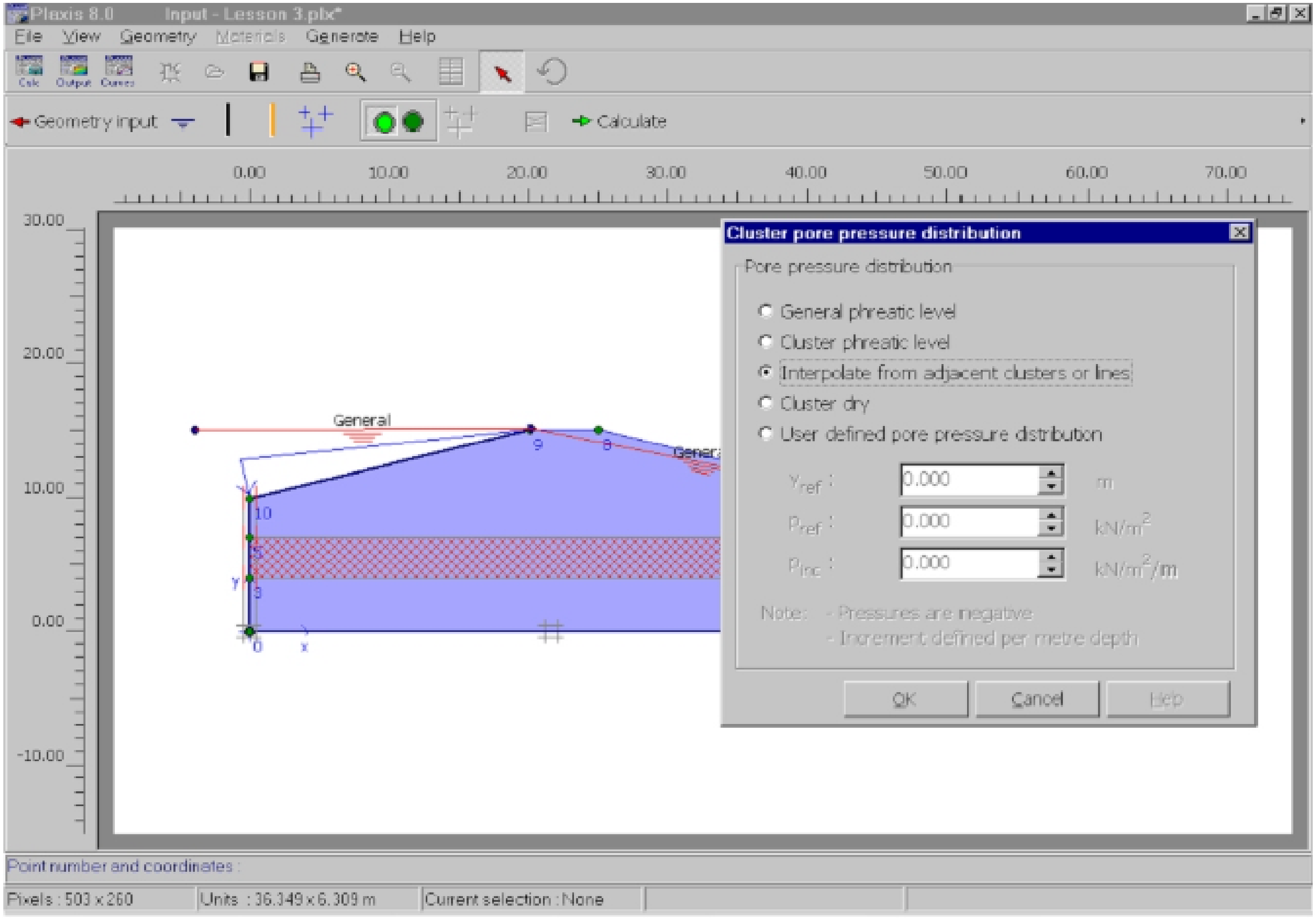


Figure 5.7 Definition of pore pressures for peat layer

Hint: A cross section can be drawn perfectly horizontal or vertical by holding down the <Shift> key while drawing the cross section.

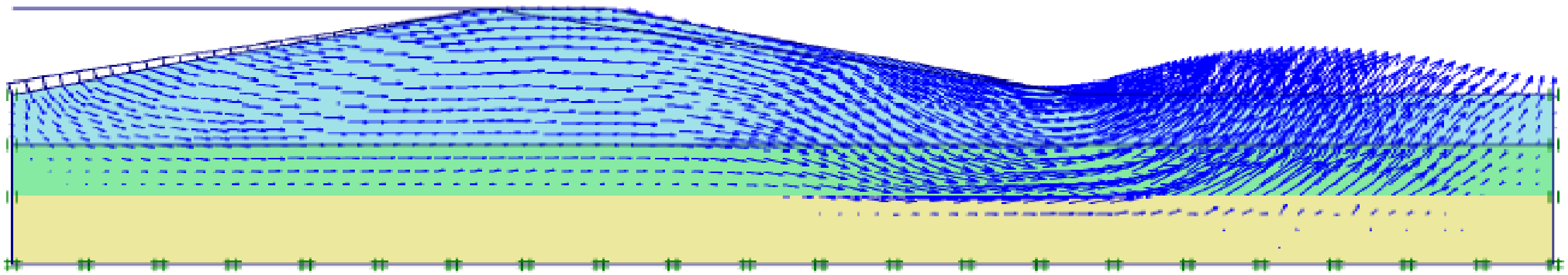


Figure 5.8 Displacement increments due to the change in water level

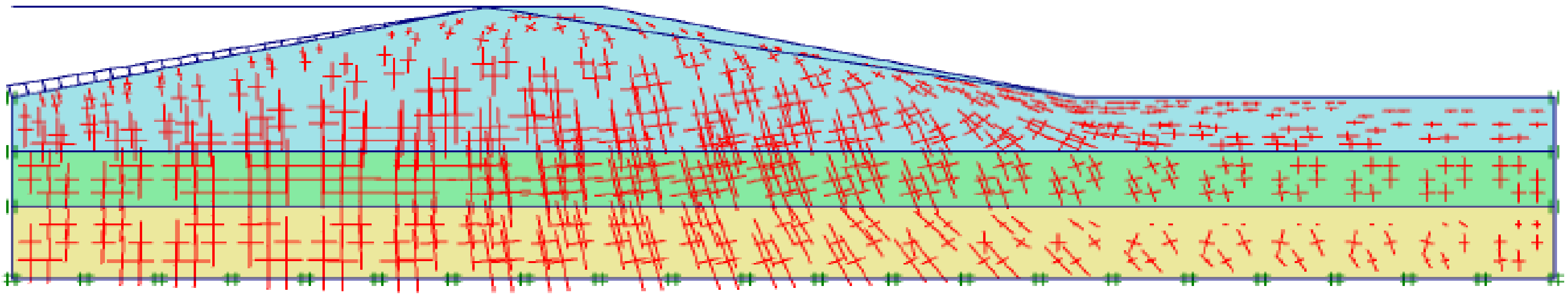


Figure 5.9 Effective stresses in embankment after the increase of the water level

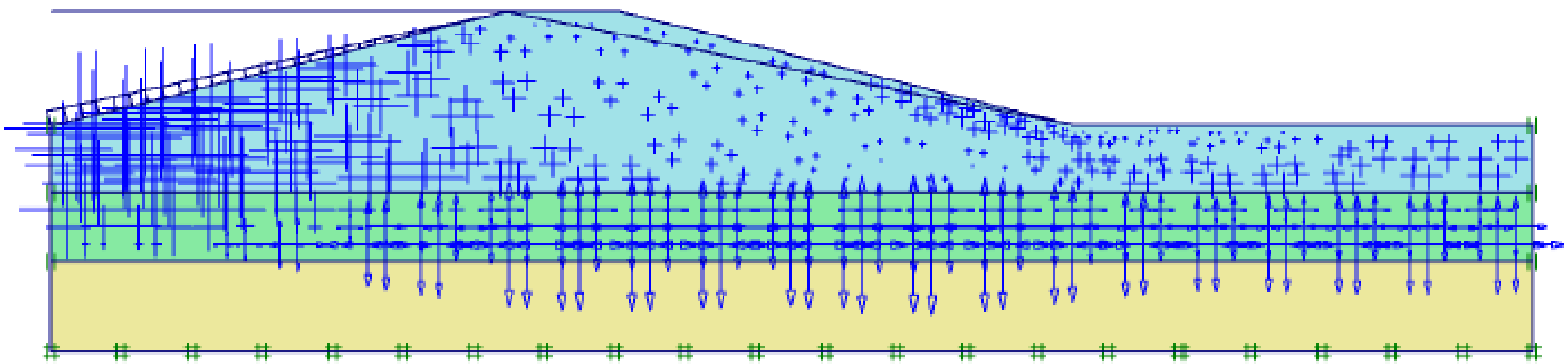


Figure 5.10 Excess pore pressures after the increase of the water level

LESSON 4

DRY EXCAVATION USING TIE BACK WALL

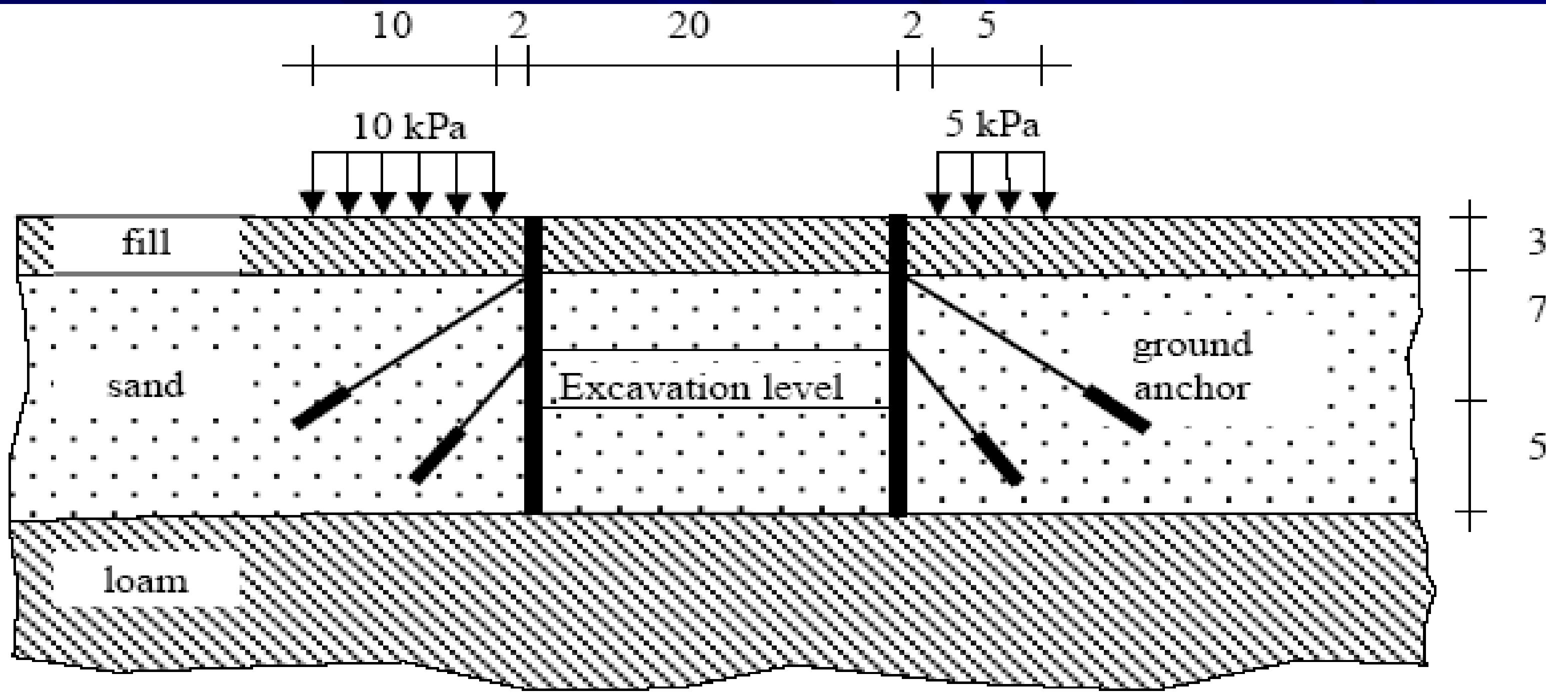


Figure 6.1 Excavation supported by tie back walls

Hint: In general, it is a good habit to extend interfaces around corners of structures to allow for sufficient freedom of deformation and to obtain a more accurate stress distribution. When doing so, make sure that the strength of the extended part of the interface is equal to the soil strength and that the interface does not influence the flow field, if applicable.

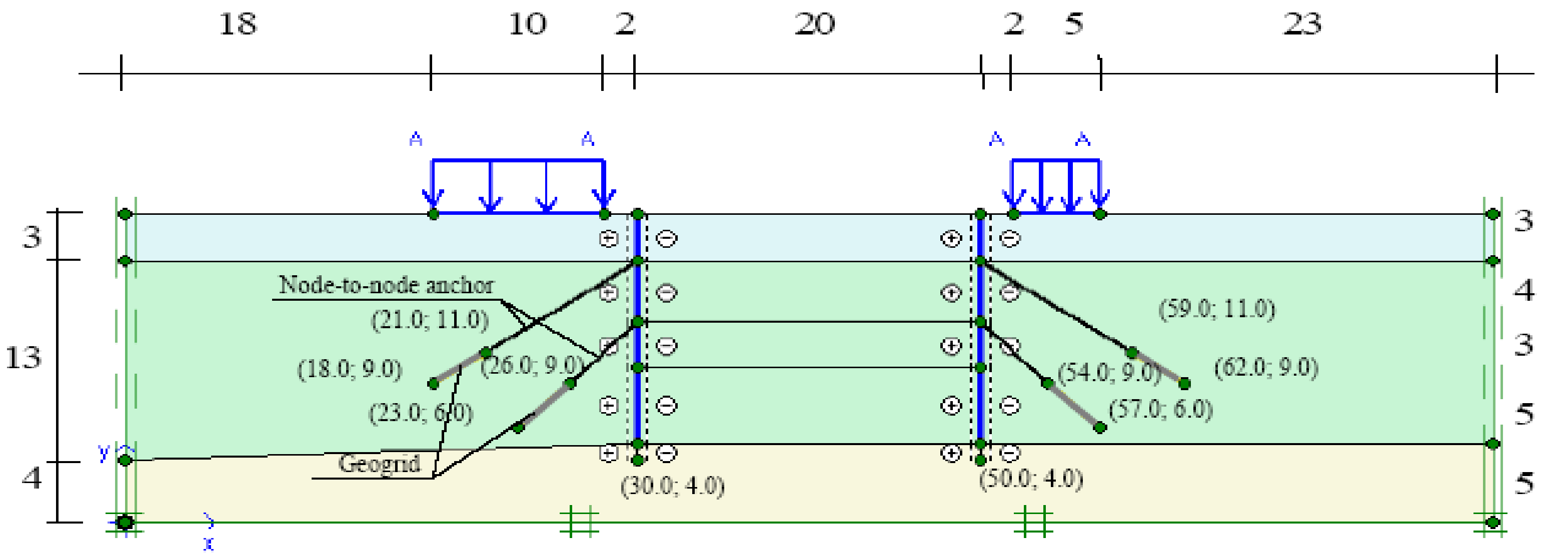


Figure 6.2 Geometry model of building pit

Hint: The extended part of an interface is not used for soil-structure interaction and should therefore have the same strength as the surrounding soil. This can be achieved with a strength reduction factor R_{inter} = 1.0, which is automatically adopted in the *Rigid* selection. If necessary, a separate material data set must be created for the extended part of an interface. In addition, the extended part of an interface should not influence the flow field. This is achieved by deactivating the interface when generating the pore pressures.

Table 6.1. Soil and interface properties

Parameter	Name	Fill	Sand	Loam	Unit
Material model	*Model*	MC	MC	MC	-
Type of material behaviour	*Type*	Drained	Drained	Drained	-
Soil unit weight above p.l.	γ_{unsat}	16	17	17	kN/m^3
Soil unit weight below p.l.	γ_{sat}	20	20	19	kN/m^3
Horizontal permeability	k_x	1.0	0.5	0.1	m/day
Vertical permeability	k_y	1.0	0.5	0.1	m/day
Young's modulus	E_{ref}	8000	30000	20000	kN/m^2
Poisson's ratio	ν	0.30	0.30	0.33	-
Cohesion	c_{ref}	1.0	1.0	8.0	kN/m^2
Friction angle	φ	30	34	29	°
Dilatancy angle	ψ	0.0	4.0	0.0	°
Interface reduction factor	R_{inter}	0.65	0.70	Rigid	-

Table 6.2. Properties of the diaphragm wall (plate)

Parameter	Name	Value	Unit
Type of behaviour	*Material type*	Elastic	-
Normal stiffness	*EA*	$12 \cdot 10^6$	kN/m
Flexural rigidity	*EI*	$0.12 \cdot 10^6$	kNm^2/m
Equivalent thickness	*d*	0.346	m

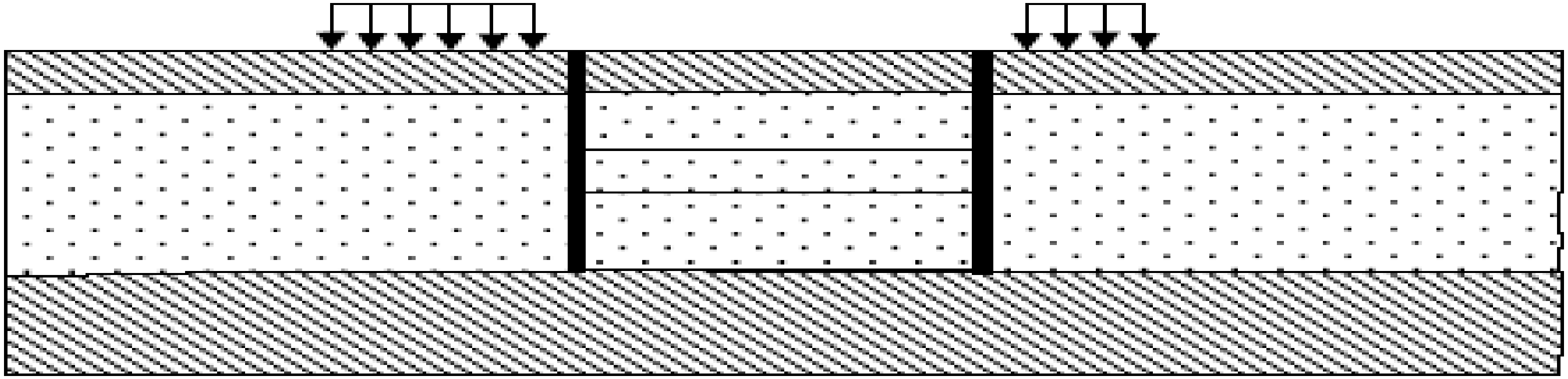
Parameter	Name	Value	Unit
Weight	w	8.3	kN/m/m
Poisson's ratio	ν	0.15	-

Table 6.3. Properties of the anchor rod (node-to-node anchor)

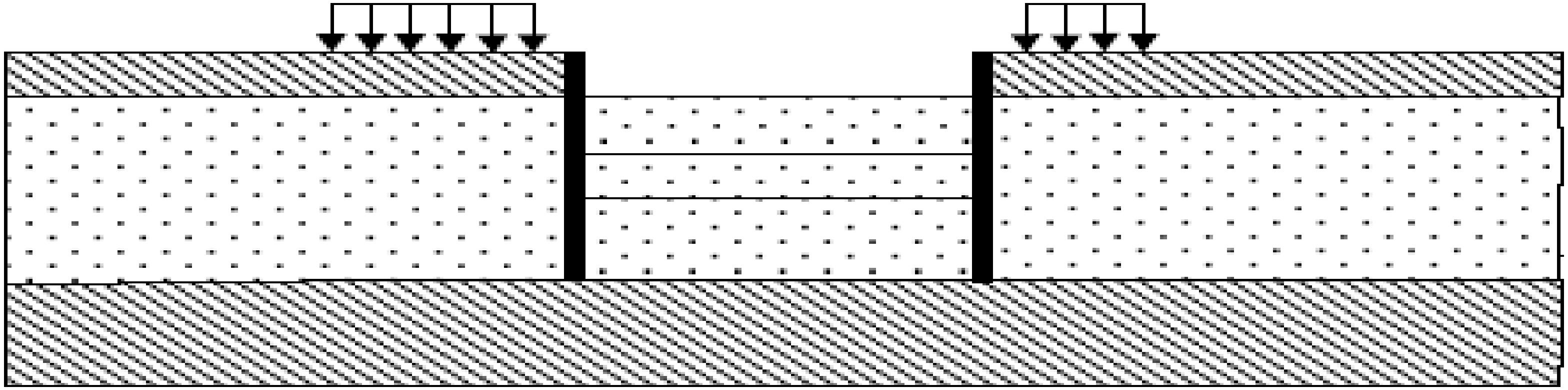
Parameter	Name	Value	Unit
Type of behaviour	*Material type*	Elastic	-
Normal stiffness	EA	$2 \cdot 10^5$	kN
Spacing out of plane	L_s	2.5	m
Maximum force	$F_{max,comp}$	$1 \cdot 10^{15}$	kN
	$F_{max,tens}$	$1 \cdot 10^{15}$	kN

Table 6.4. Property of the grout body (geogrid)

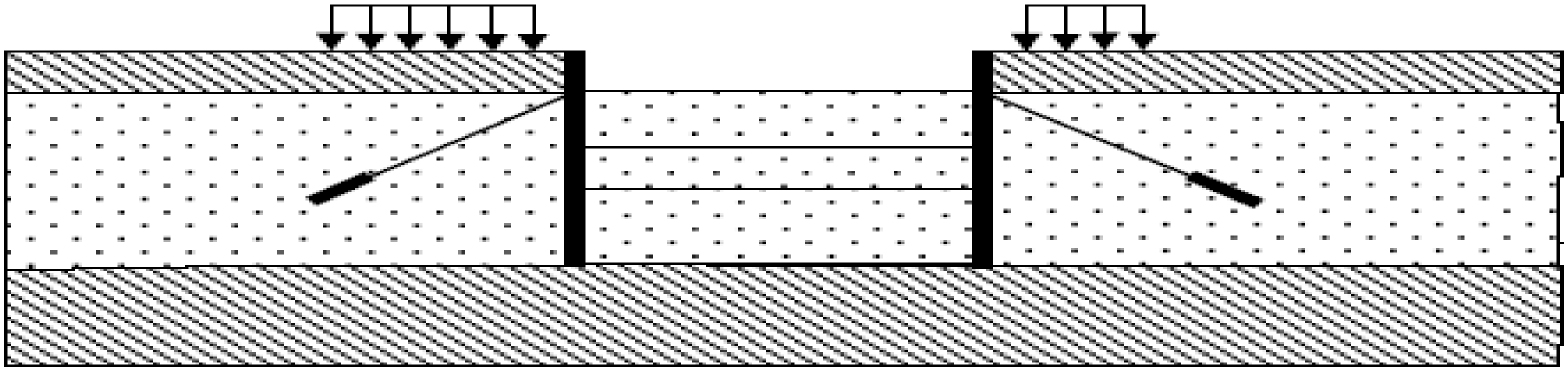
Parameter	Name	Value	Unit
Normal stiffness	EA	$1 \cdot 10^5$	kN/m



Phase 1

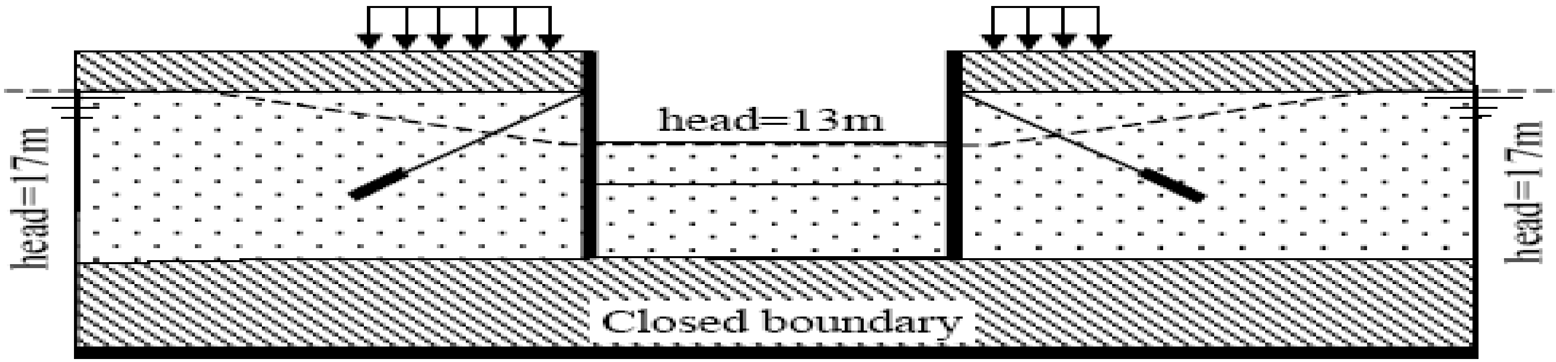


Phase 2



Phase 3

Hint: A pre-stress force is exactly matched at the end of a finished staged construction calculation and turned into an anchor force. In successive calculation phases the force is considered to be just an anchor force and can therefore further increase or decrease, depending on the development of the surrounding stresses and forces.



Phase 4

Hint: The results of a groundwater calculation can be viewed as *Pore pressures*, *Flow field* and *Groundwater head*. These options are available from the *Stress* menu.

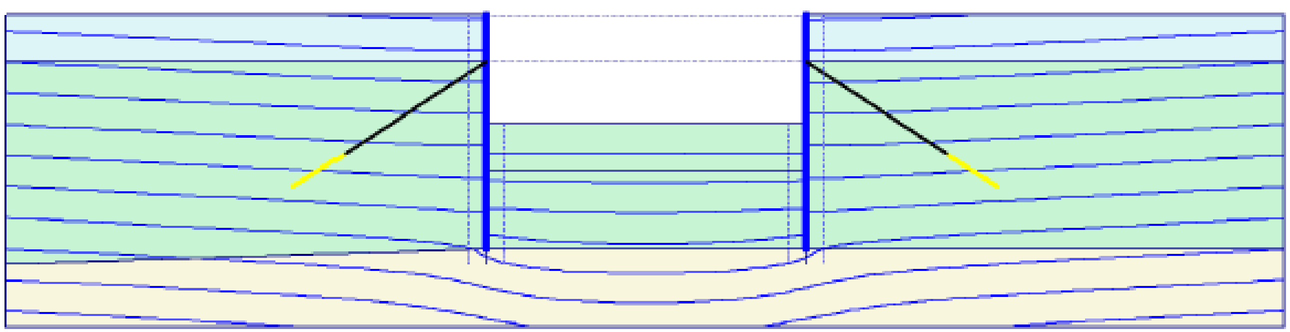
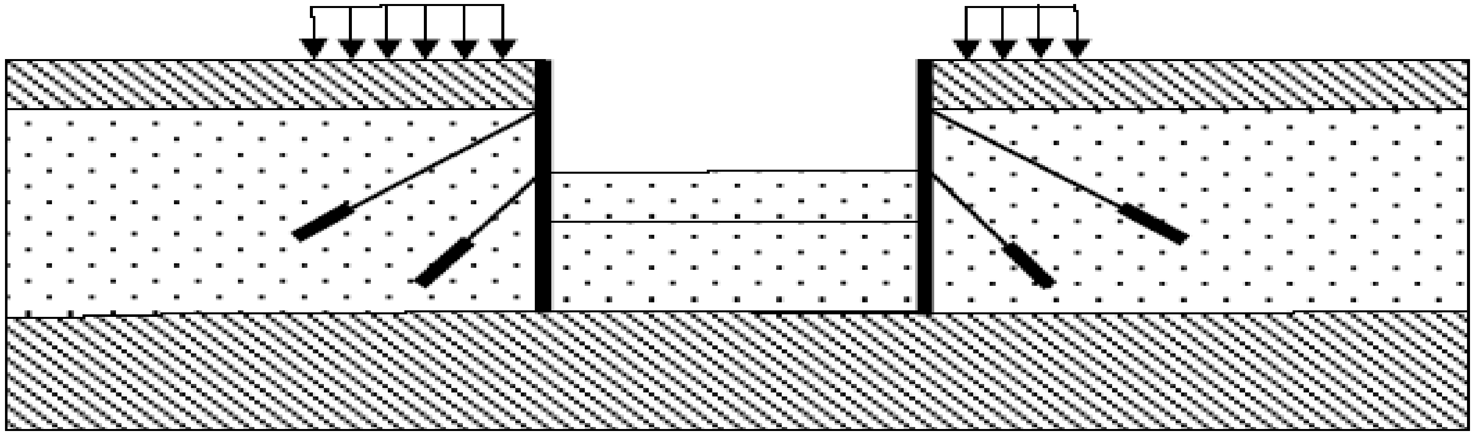
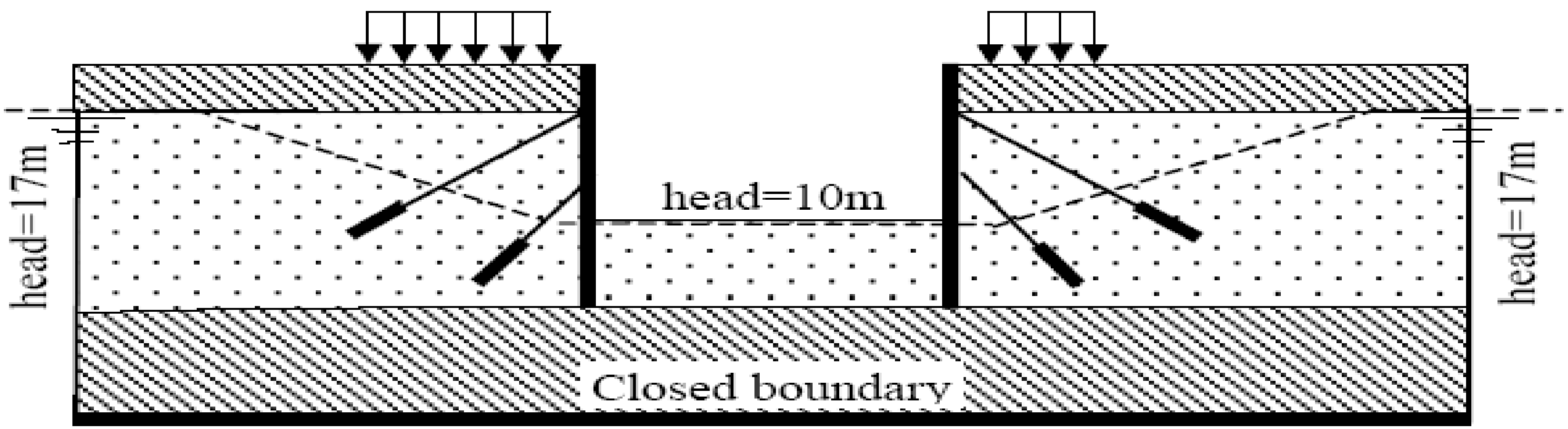


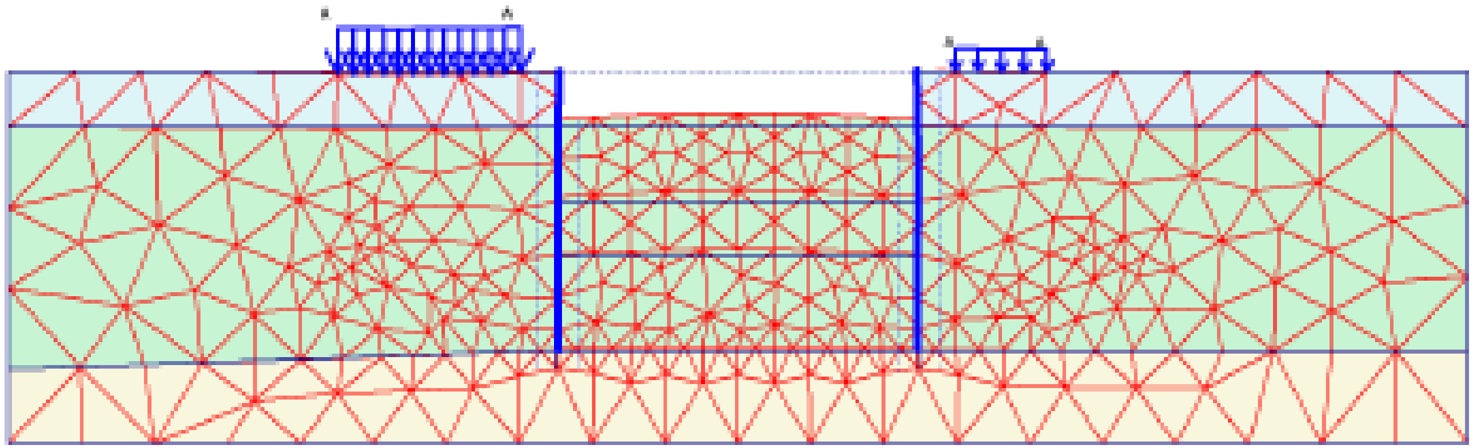
Figure 6.3 Active pore pressure contours resulting from groundwater calculation



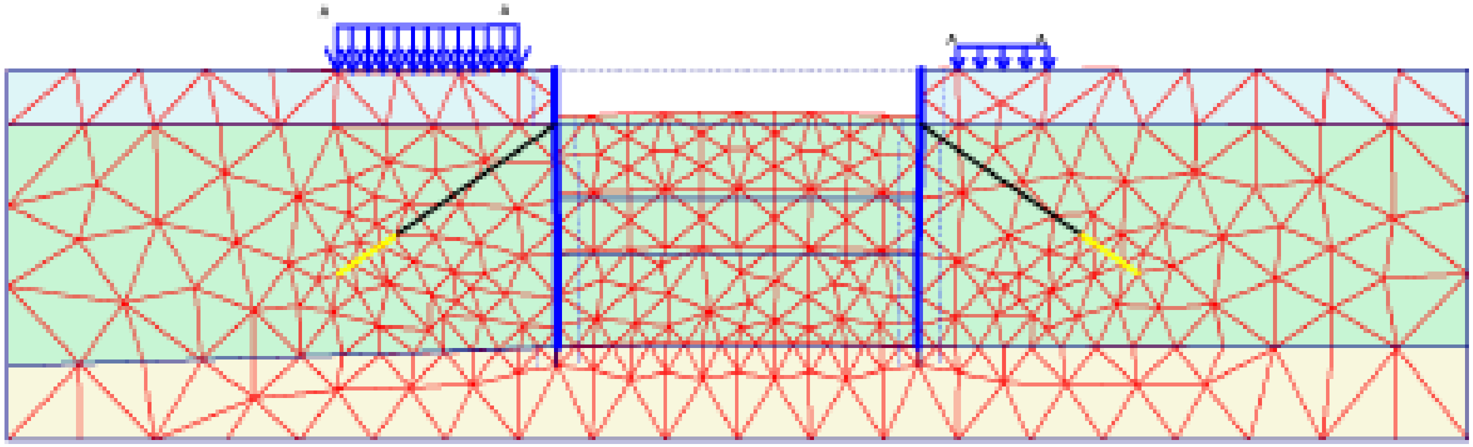
Phase 5



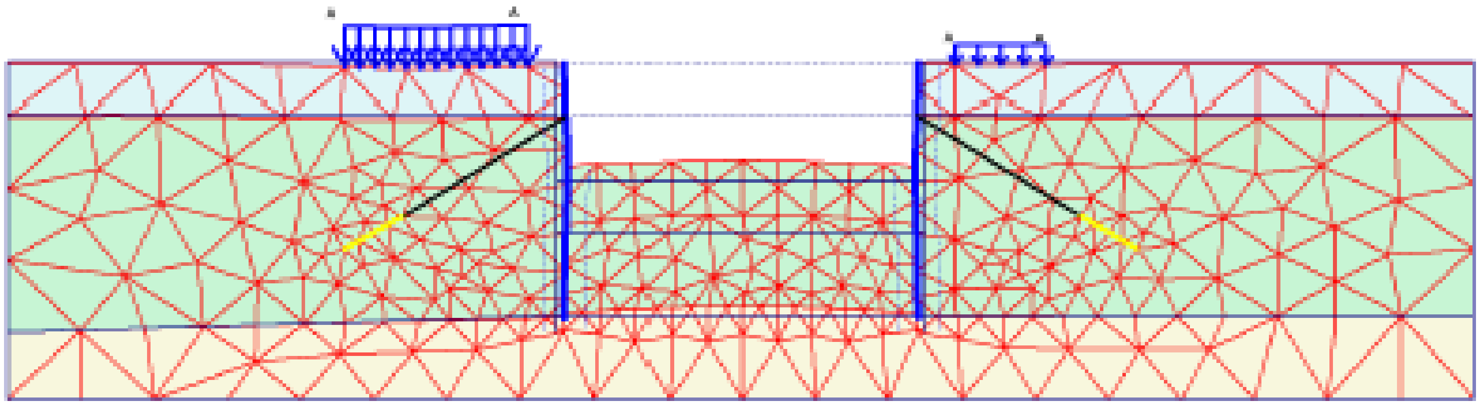
Phase 6



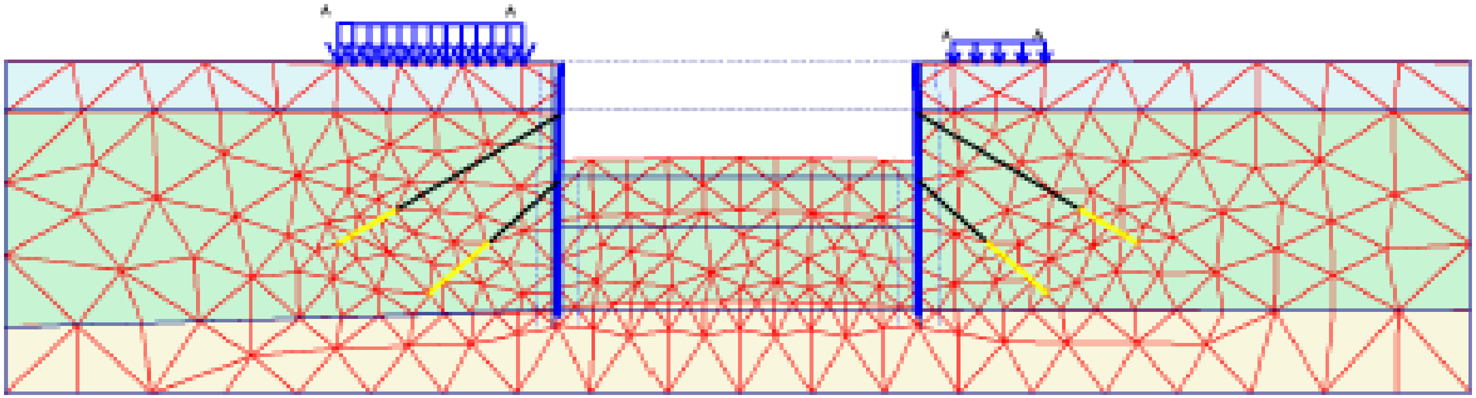
(a) phase 2



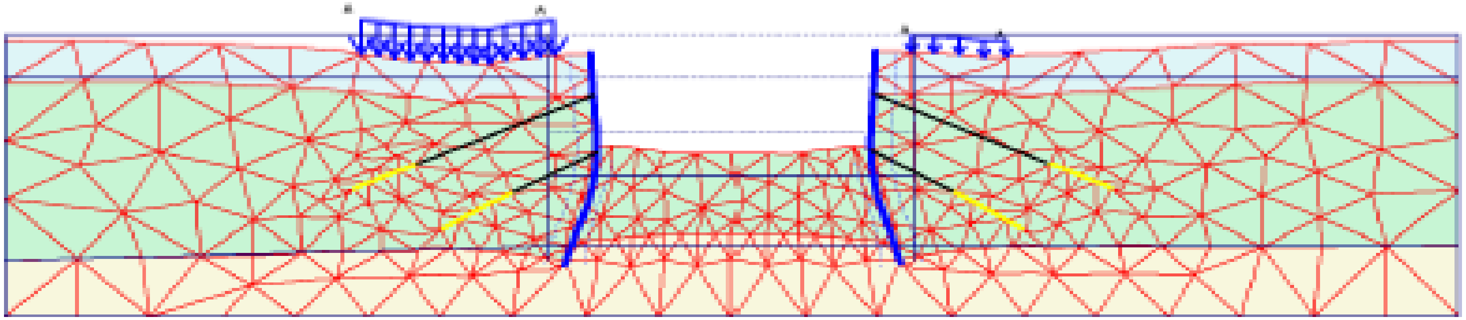
(b) phase 3



(c) phase 4



(d) phase 5



(e) final stage

Figure 6.4 Deformed mesh stages (a) to (e)

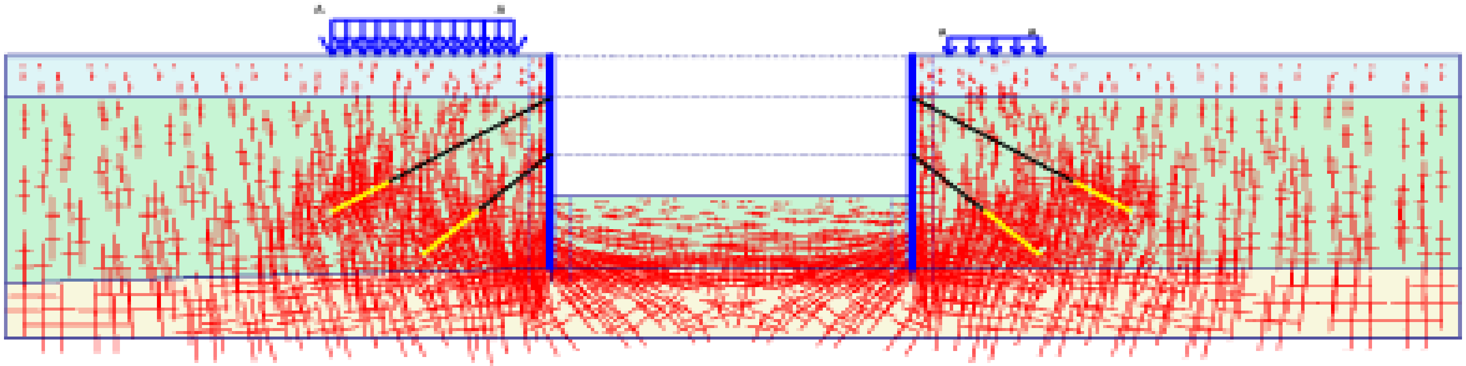


Figure 6.5 Effective stresses, final stage

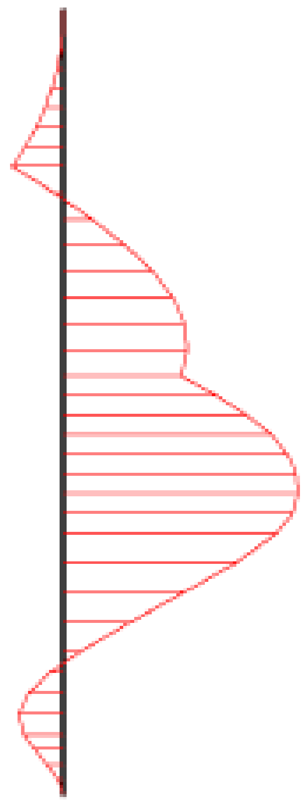


Figure 6.6 Bending moments in the left diaphragm wall in the final stage

LESSON 5

CONSTRUTION OF A ROAD EMBEDMENT

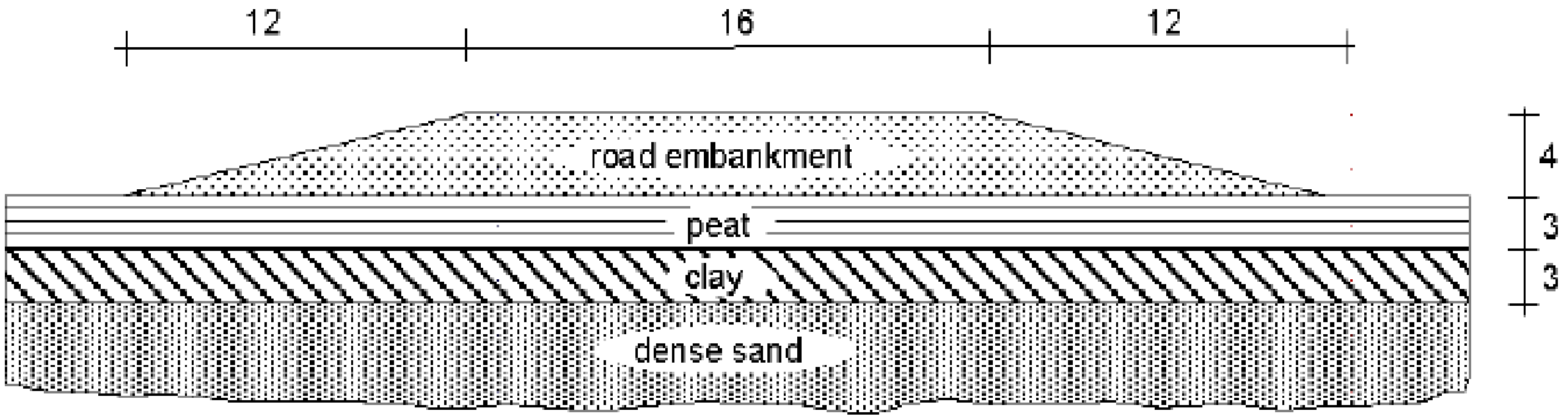


Figure 7.1 Situation of a road embankment on soft soil

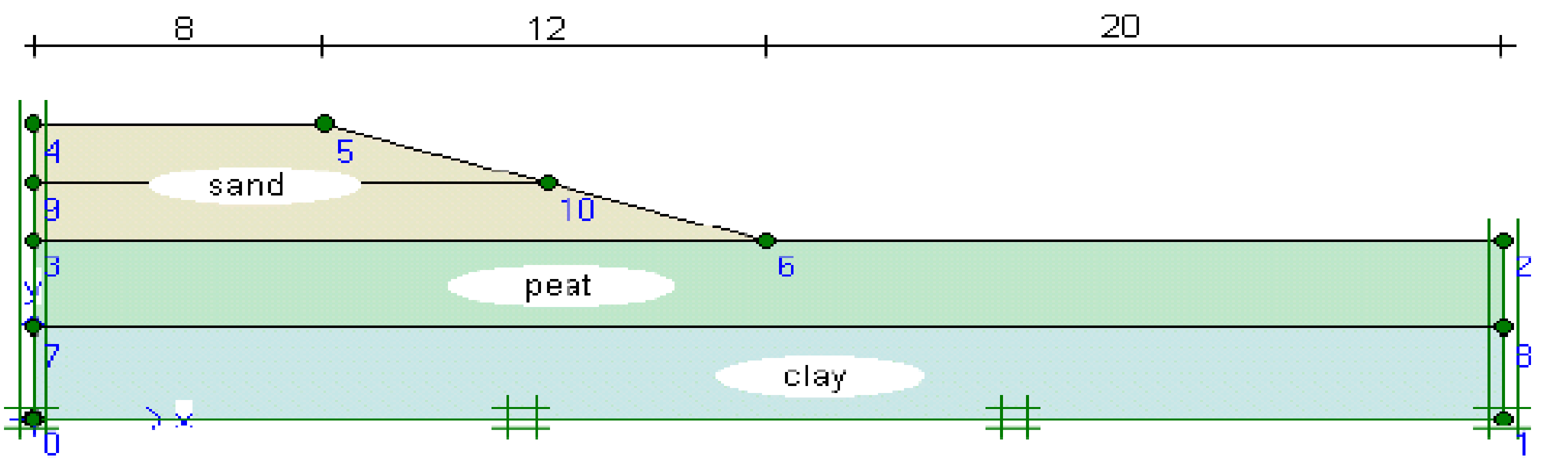


Figure 7.2 Geometry model of road embankment project

Table 7.1. Material properties of the road embankment and subsoil

Parameter	Name	Clay	Peat	Sand	Unit
Material model	*Model*	MC	MC	MC	-
Type of behaviour	*Type*	Undrained	Undrained	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	15	8	16	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	18	11	20	kN/m^3
Horizontal permeability	k_x	$1 \cdot 10^{-4}$	$2 \cdot 10^{-3}$	1.0	m/day
Vertical permeability	k_y	$1 \cdot 10^{-4}$	$1 \cdot 10^{-3}$	1.0	m/day
Young's modulus	E_{ref}	1000	350	3000	kN/m^2
Poisson's ratio	ν	0.33	0.35	0.3	-
Cohesion	c_{ref}	2.0	5.0	1.0	kN/m^2
Friction angle	φ	24	20	30	°
Dilatancy angle	ψ	0.0	0.0	0.0	°

Hint: Closed consolidation boundaries can only be defined by clicking on existing geometry points. The program will automatically find intermediate geometry points.

> Consolidation boundary conditions must be generated in the boundary nodes of the mesh. This is done together with the generation of water pressures. Hence, after introducing or changing consolidation boundaries, always click on the *Generate water pressures* button.

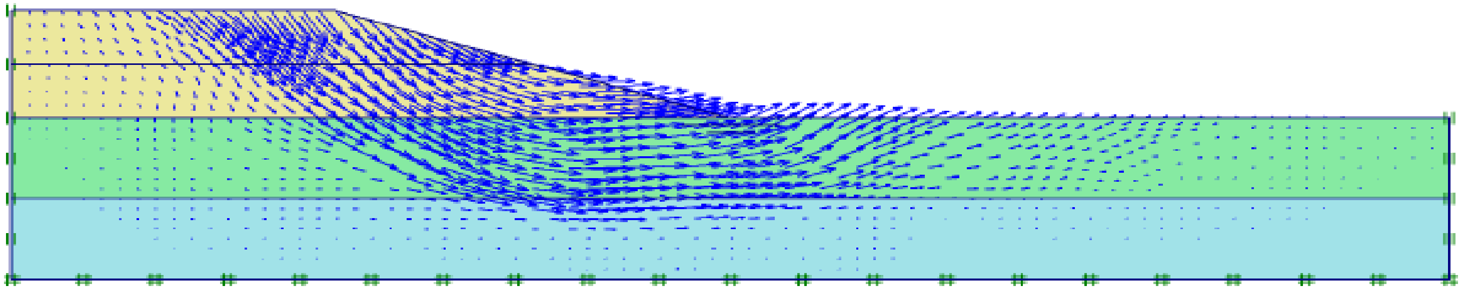


Figure 7.3 Displacement increments after undrained construction of embankment

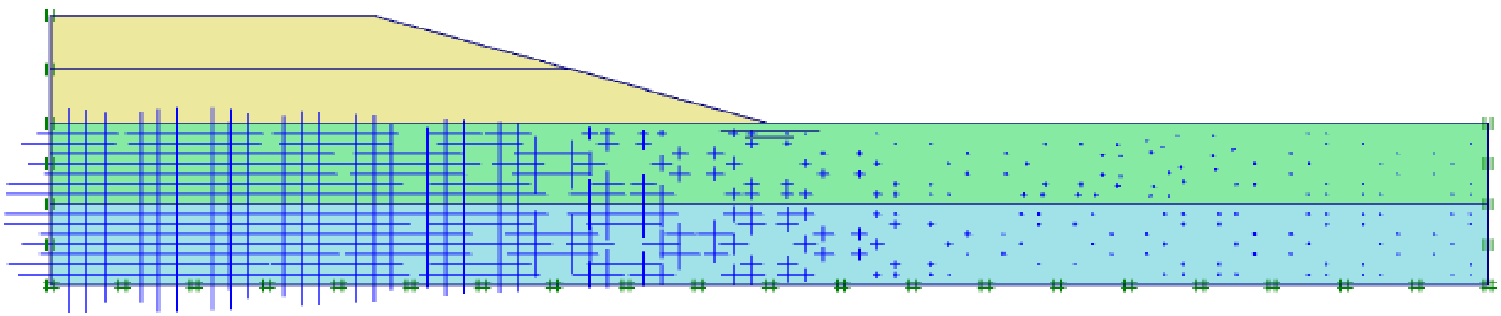


Figure 7.4 Excess pore pressures after undrained construction of embankment

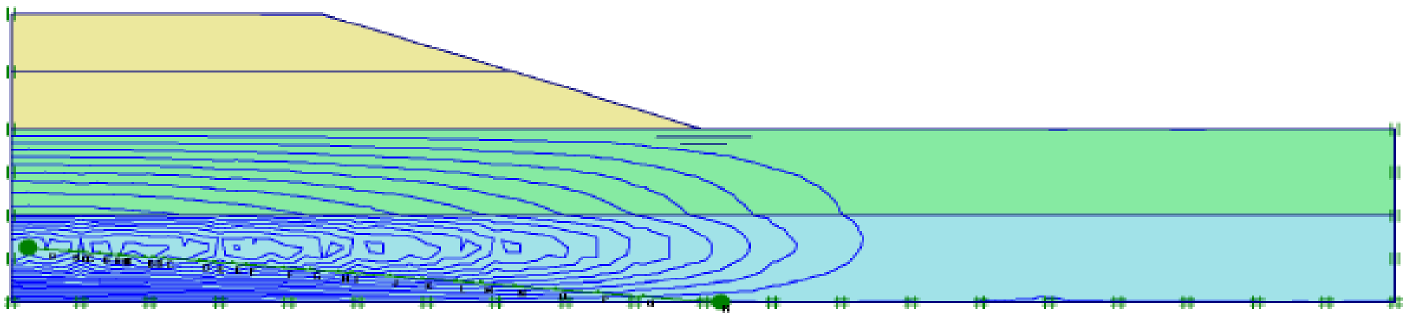


Figure 7.5 Excess pore pressure contours after consolidation to $P_{excess} < 1.0$ kN/m^2

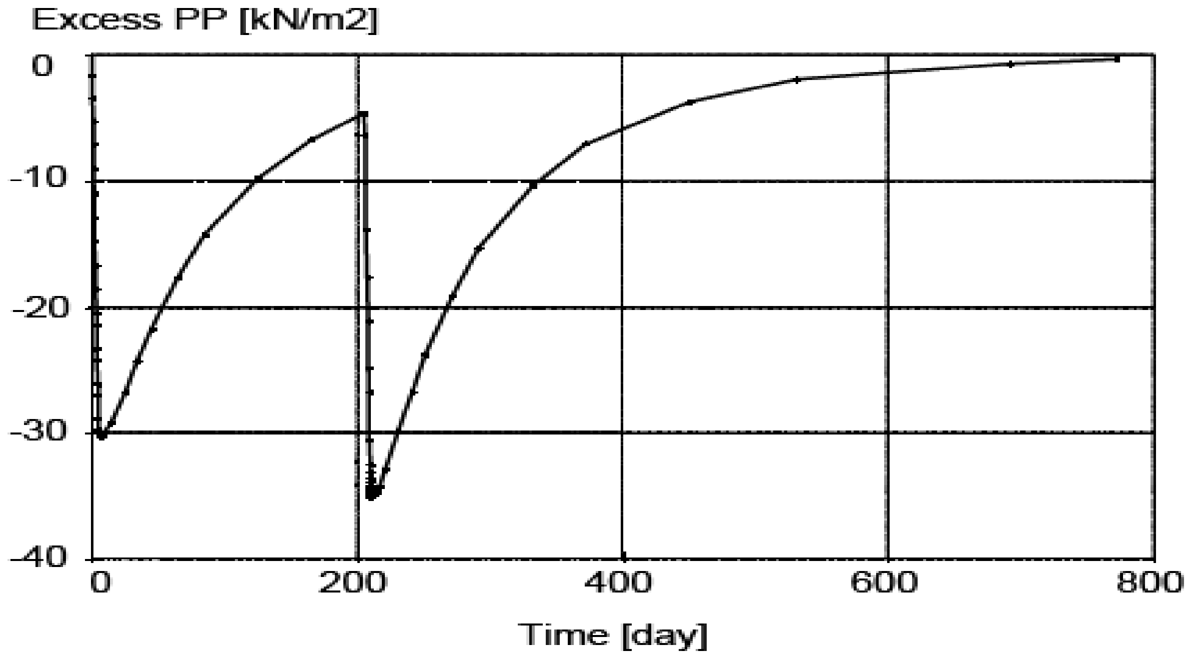


Figure 7.6 Development of excess pore pressure under the embankment

Hint: The default value of *Additional steps* in a *Phi-c-reduction* calculation is 100. In contrast to an *Ultimate level* calculation, the number of additional steps is always fully executed. In most P*hi-c-reduction* calculations, 100 steps are sufficient to arrive at a state of failure. If not, the number of additional steps can be increased to a maximum of 1000.

> For most phi-c-reduction calculations *Msf* = 0.1 is an adequate first step to start up the process. During the calculation process, the development of the total multiplier for the strength reduction, Σ*Msf*, is automatically controlled by the load advancement procedure.

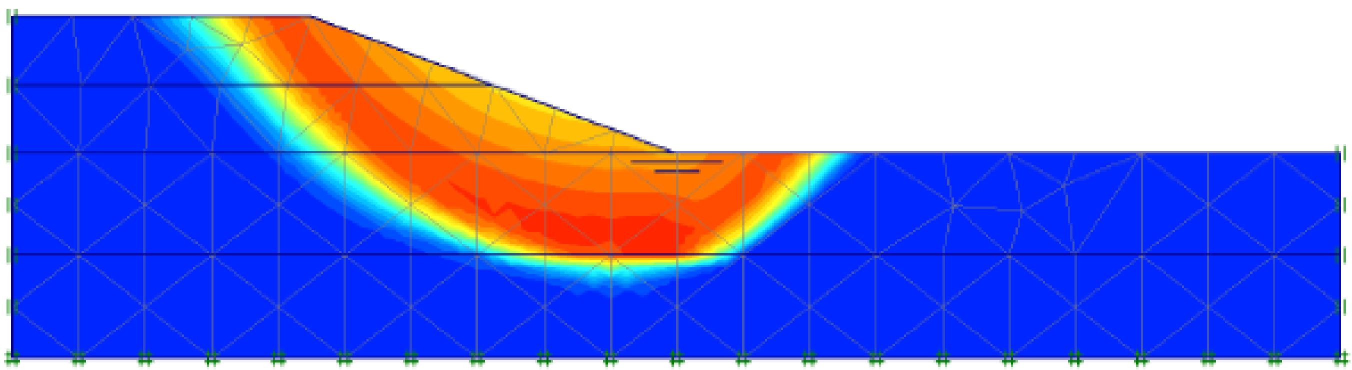


Figure 7.7 Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

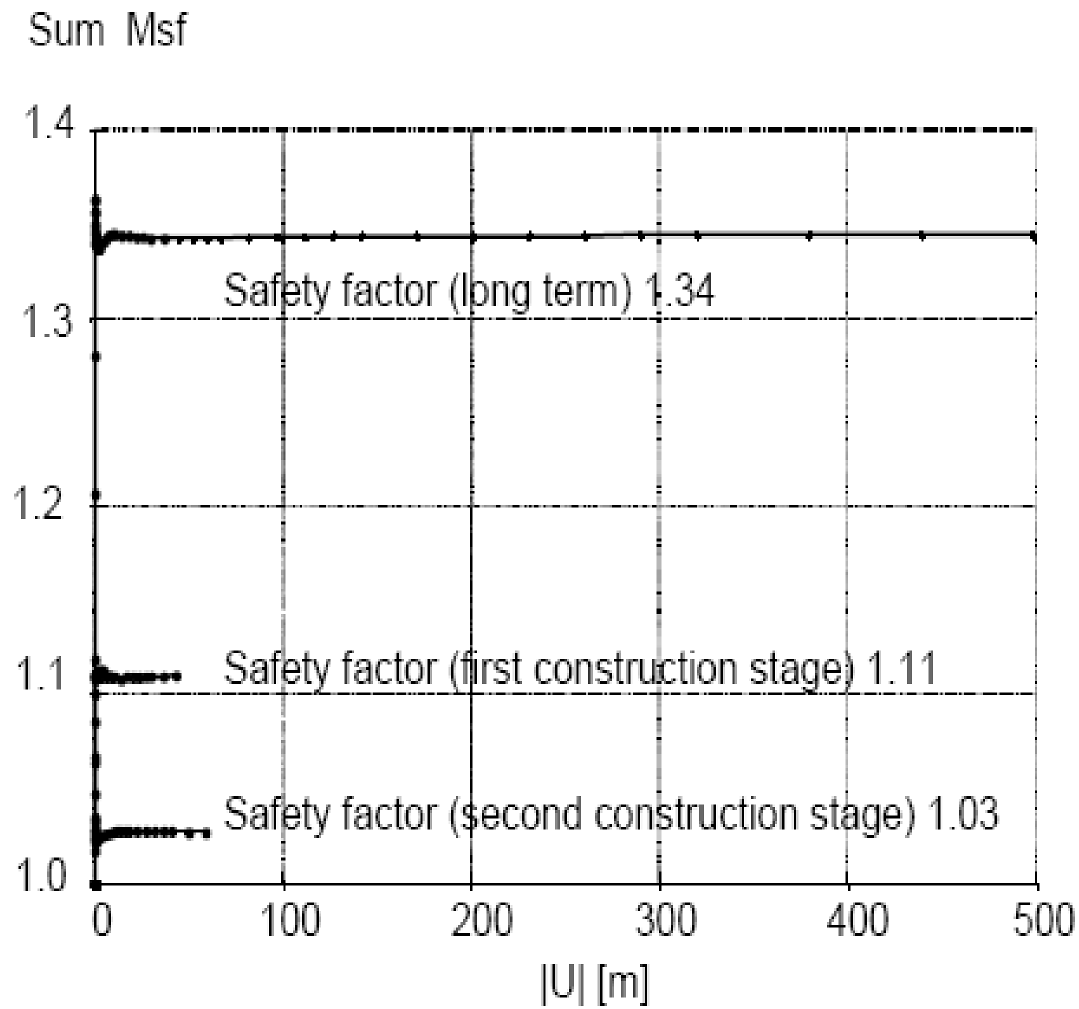


Figure 7.8 Evaluation of safety factor for three stages of the construction process

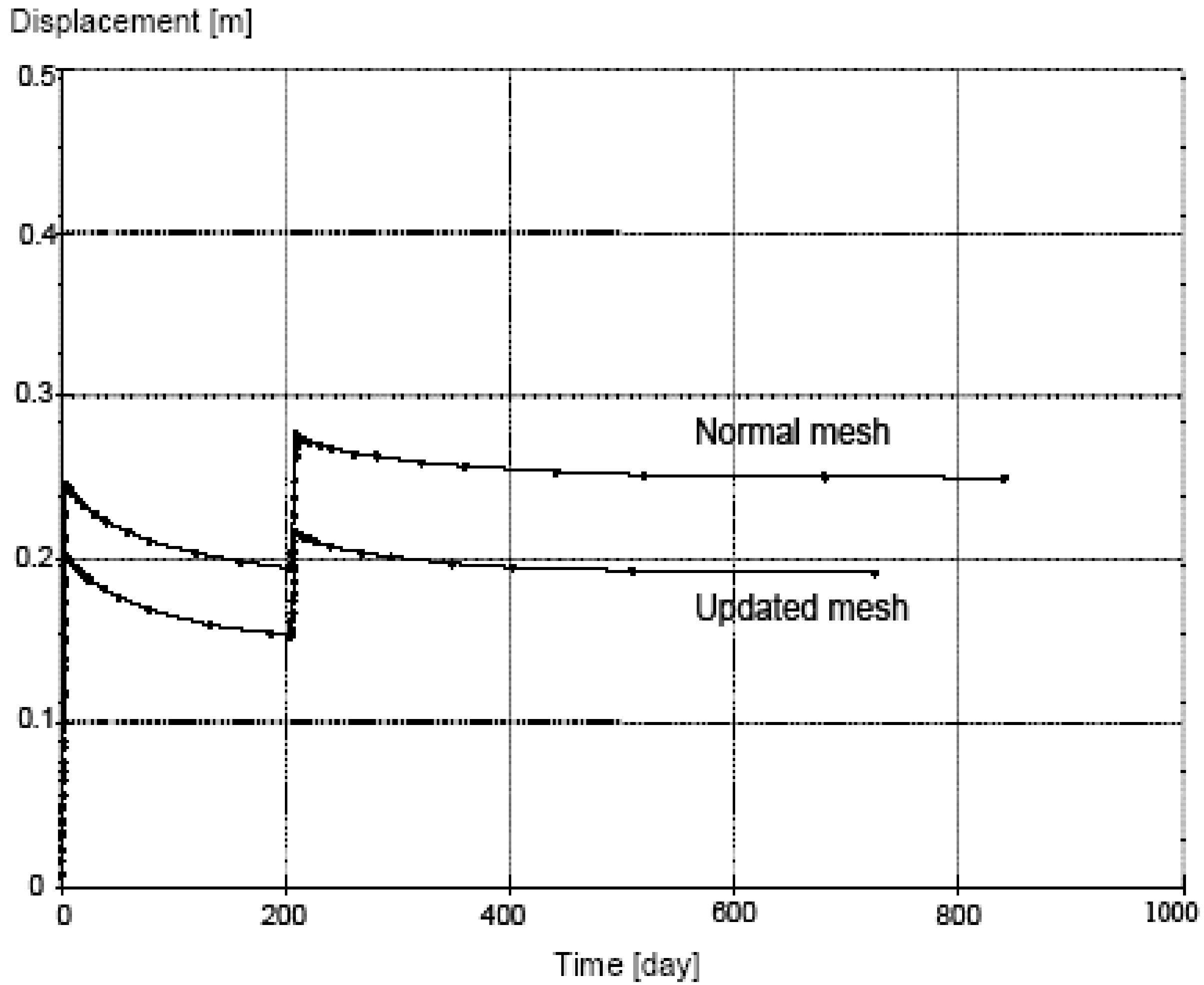


Figure 7.9 Settlements of the toe of the embankment using updated mesh calculation

LESSON 6

SETTLEMENT DUE TO TUNNEL CONSTRUTION

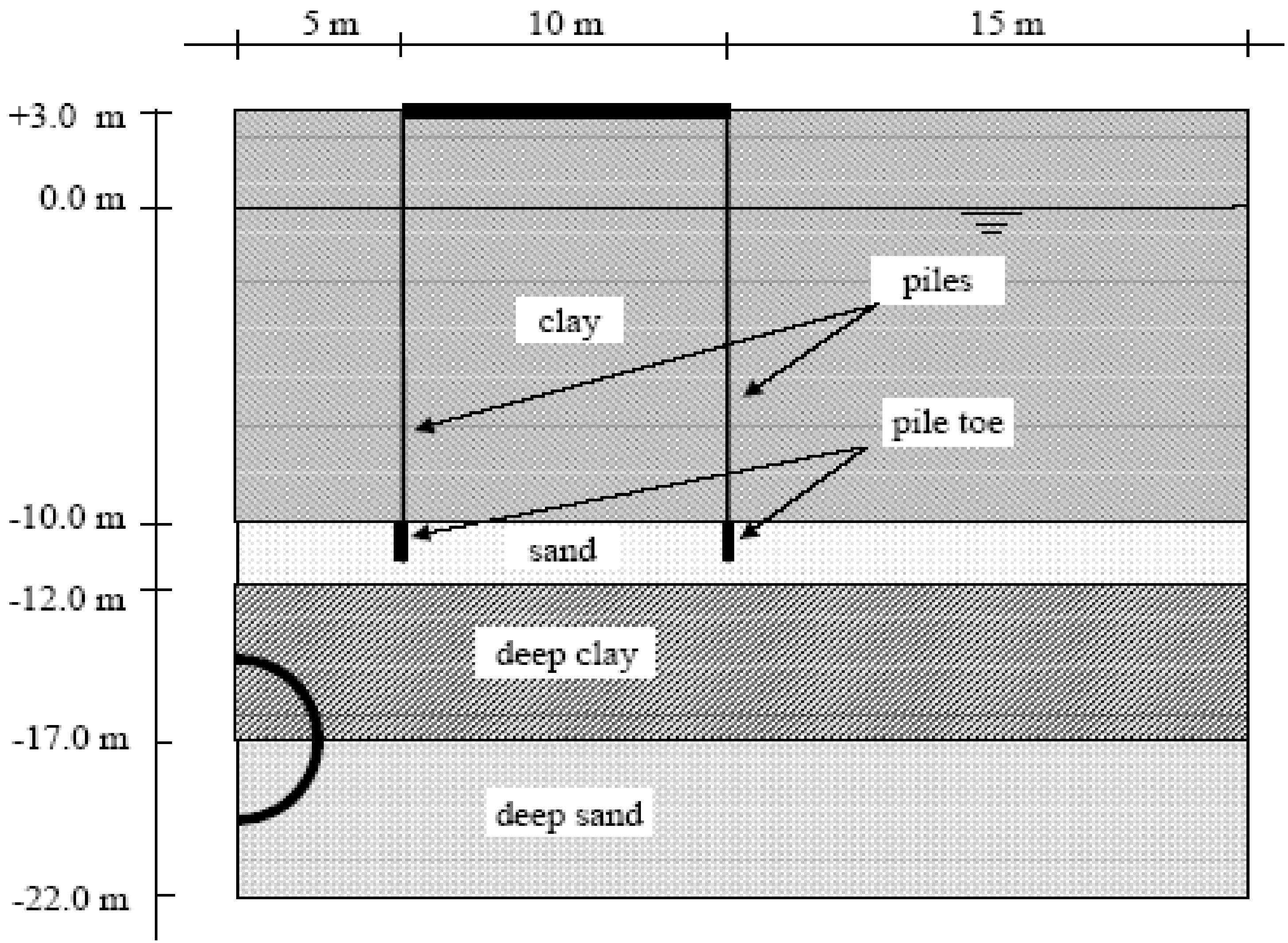


Figure 8.1 Geometry of the tunnel project with an indication of the soil layers

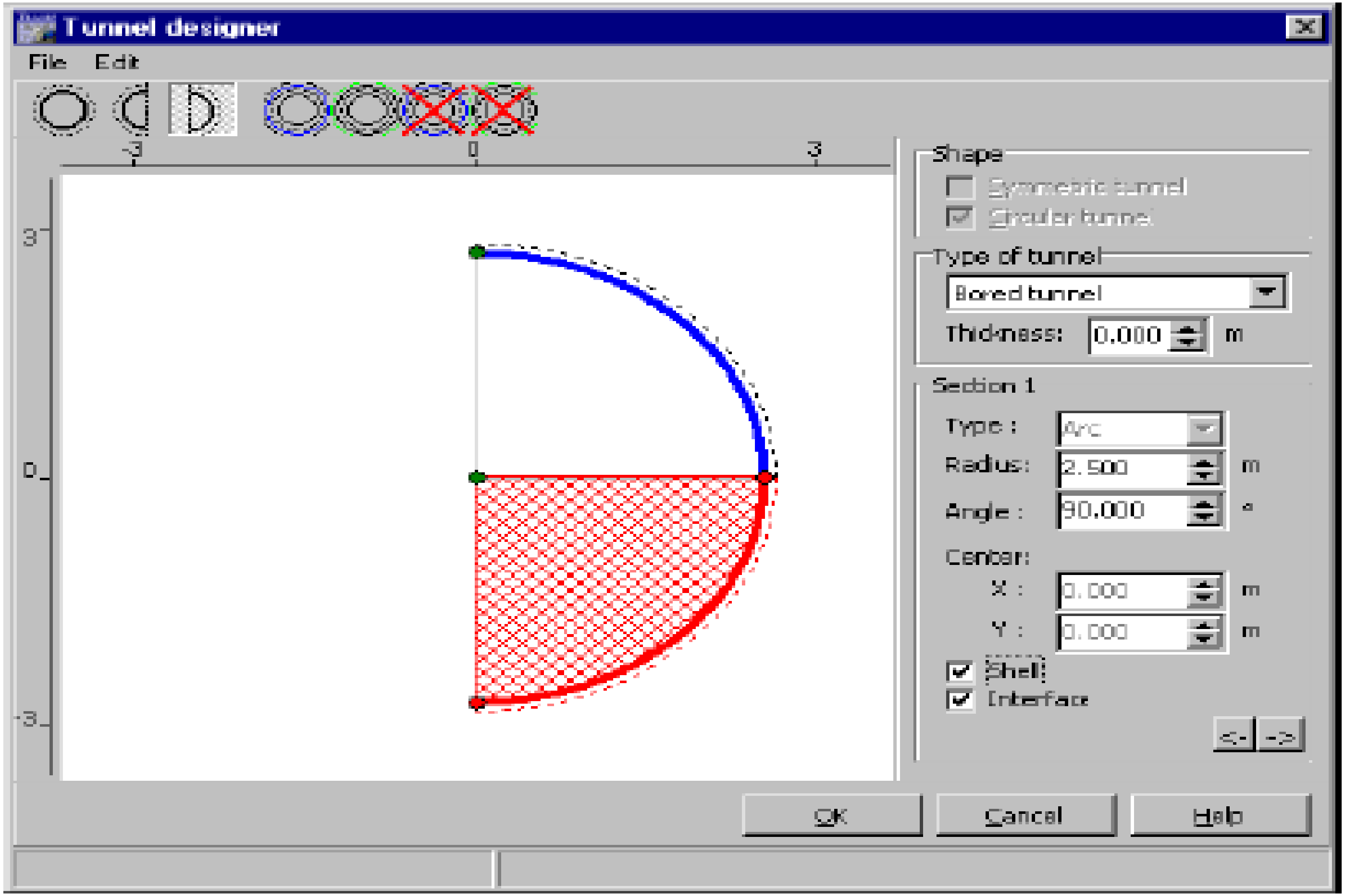


Figure 8.2 Tunnel designer with current tunnel model

Hint: A shell and interface can be assigned directly to all tunnel sections by clicking on the corresponding buttons at the top of the tunnel window.

> A tunnel lining consists of curved plates. The lining properties can be specified in the material database for plates. Similarly, a tunnel interface is nothing more than a curved interface.

> In the tunnel as considered here the sections do not have a specific meaning since the tunnel lining is homogeneous and the tunnel will be constructed at once.

> In general, the meaning of sections becomes significant when:

> It is desired to excavate or construct the tunnel (lining) in different stages.

> Different tunnel sections have different lining properties.

> One would consider hinge connections in the lining (hinges can be added after the design of the tunnel in the general drawing area).

> The tunnel shape is composed of arcs with different radii (for example NATM tunnels).

Hint: In the *Standard fixities* option, a plate that extends to a geometry boundary that is fixed in at least one direction obtains fixed rotations, whereas a plate that extends to a free boundary obtains a free rotation.

Table 8.1. Material properties of soil in the tunnel project

Parameter	Name	Clay	Sand	Dp. Clay	Dp.sand	Unit
Material model	*Model*	MC	MC	MC	MC	-
Material beh.	*Type*	Drained	drained	Drained	drained	-
Soil unit weight above phreatic level	γ_{unsat}	15	16.5	16	17	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	18	20	18.5	21	kN/m^3
H. permeability	k_x	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
V. permeability	k_y	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
Young's modulus	E_{ref}	1000	80000	10000	120000	kN/m^2
Increase *E*	E_{incr}	650	-	-	-	kN/m^3
Reference level	y_{ref}	0.0	-	-	-	m
Poisson's ratio	ν	0.33	0.3	0.33	0.3	-
Cohesion	c_{ref}	5.5	1.0	4.0	1.0	kN/m^2
Friction angle	φ	24	31	25	33	°
Dilatancy angle	ψ	0.0	1.0	0.0	3.0	°
Interface strength	R_{inter}	rigid	rigid	0.7	0.7	-

Table 8.2. Material properties of the plates

Parameter	Name	Lining	Pile toe	Building	Unit
Type of behaviour	*Type*	Elastic	Elastic	Elastic	
Normal stiffness	*EA*	$1.4 \cdot 10^7$	$2 \cdot 10^6$	$1 \cdot 10^{10}$	kN/m
Flexural rigidity	*EI*	$1.43 \cdot 10^5$	$8 \cdot 10^3$	$1 \cdot 10^{10}$	kNm^2/m
Equivalent thickness	*d*	0.35	0.219	3.464	m
Weight	*w*	8.4	2.0	25	kN/m/m
Poisson's ratio	ν	0.15	0.2	0.0	-

Table 8.3. Material properties of the anchors

Parameter	Name	Pile	Unit
Material type	*Type*	Elastic	
Normal stiffness	*EA*	$2 \cdot 10^6$	kN
Spacing between anchors	$L_{spacing}$	1	m

Hint: The contraction of the tunnel lining by itself does not introduce forces in the tunnel lining. Eventual changes in lining forces as a result of the contraction procedure are due to stress redistributions in the surrounding soil or to changing external forces.

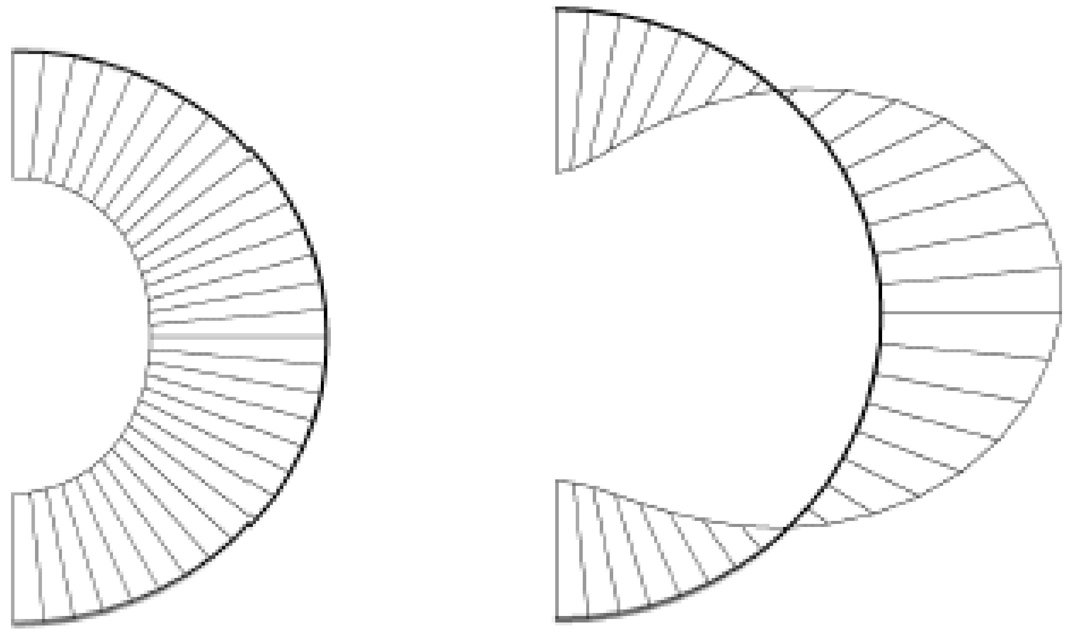


Figure 8.3 Axial forces and Bending moments in the lining after the second phase.

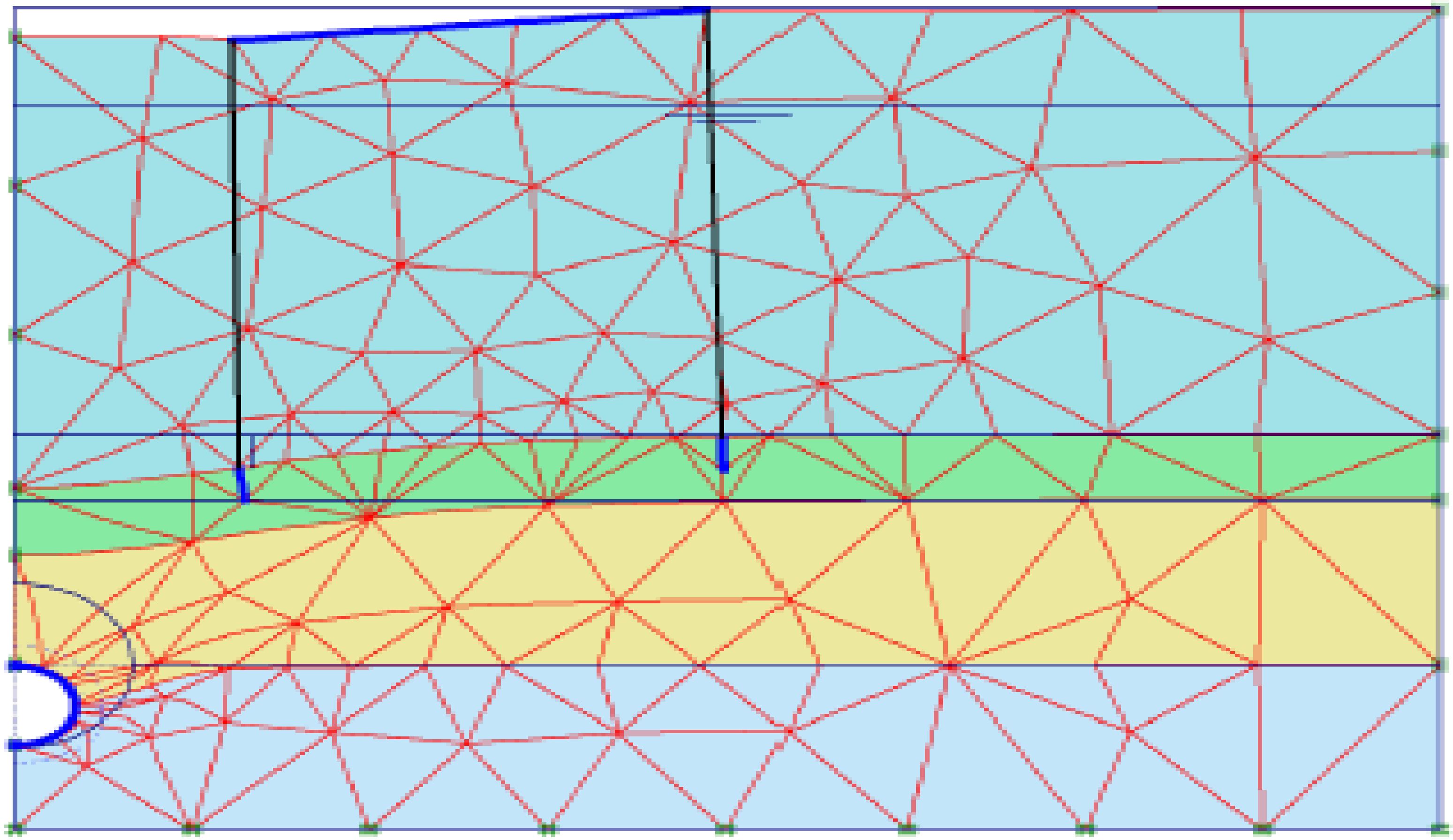


Figure 8.4 Deformed mesh after construction of the tunnel

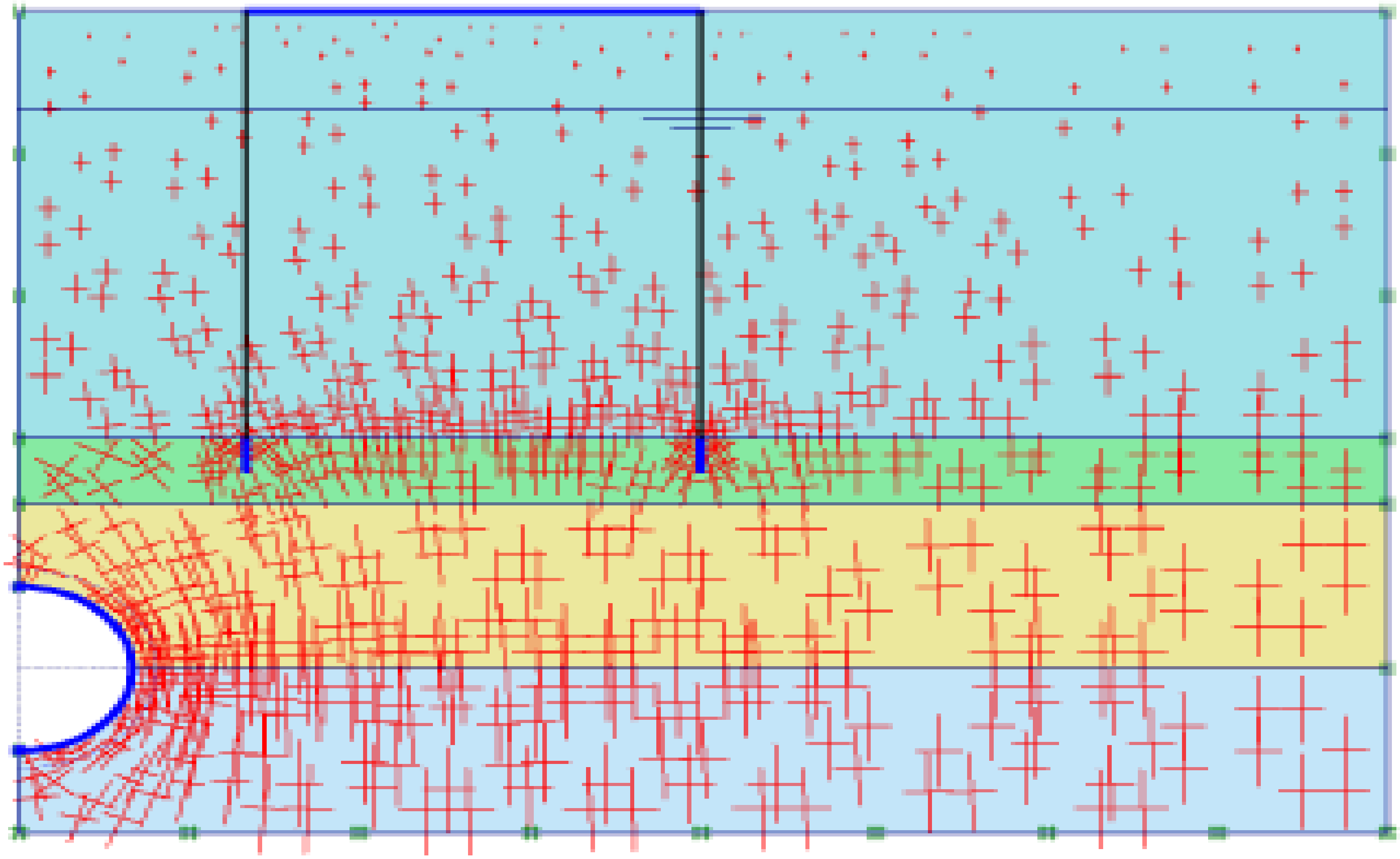


Figure 8.5 Effective stresses after construction of the tunnel

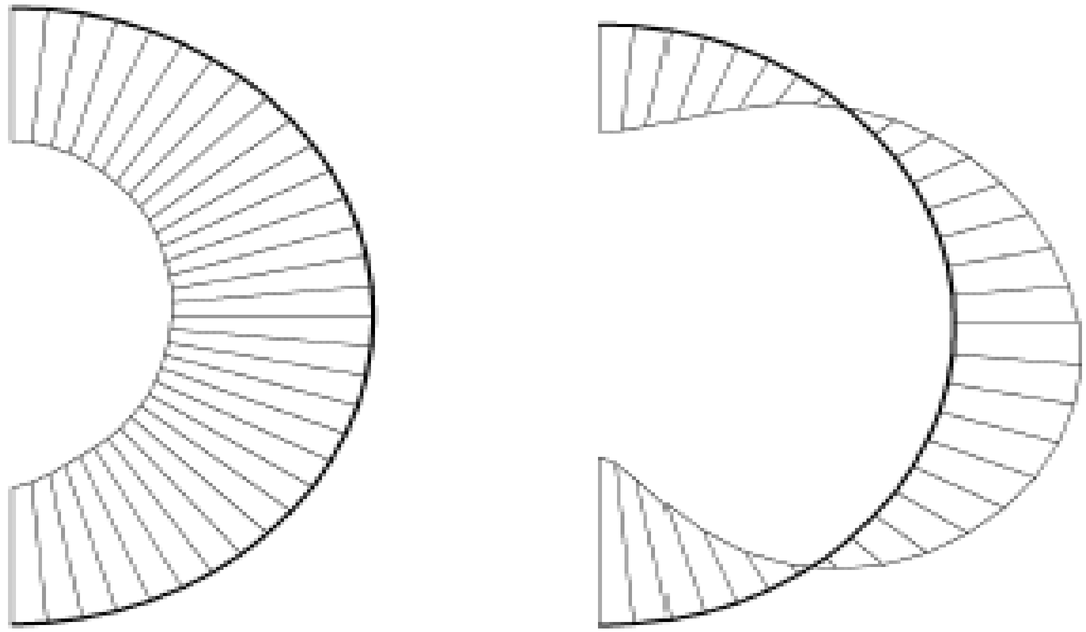


Figure 8.6 Axial forces and bending moments in the lining after the third phase

THANK YOU