PLAXIS-GiD
Tutorial Manual
Version 1

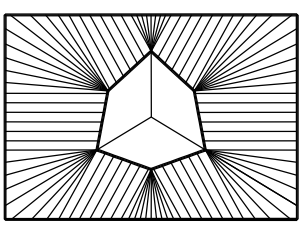

TABLE OF CONTENTS

1 STABILITY OF A TUNNEL (LESSON 1)

In this lesson the stability of a bored tunnel will be considered. While excavating the tunnel, the tunnel heading needs support in the form of liquid, air, ground pressure or mechanical pressure. This pressure must be bounded between a minimum and maximum depending on the soil properties, the depth of the tunnel and the groundwater pressure at the face of the tunnel heading. The minimum face pressure is determined mainly by the need to compensate for the groundwater pressure. A too low pressure can lead to inward collapse of the tunnel heading (active failure). A too high pressure can result in a blow out of the heading (passive failure); at the same time, large deformations may occur at the ground surface.

In this lesson we will model the excavation of a bored tunnel with a face pressure of 110 kN/m^2. In addition, a safety analysis will be applied to determine the safety factor. Water will not be considered yet.

The proposed geometry for this exercise is 60 m wide, 60 m high and 30 m long, as shown in Figure 1.1.

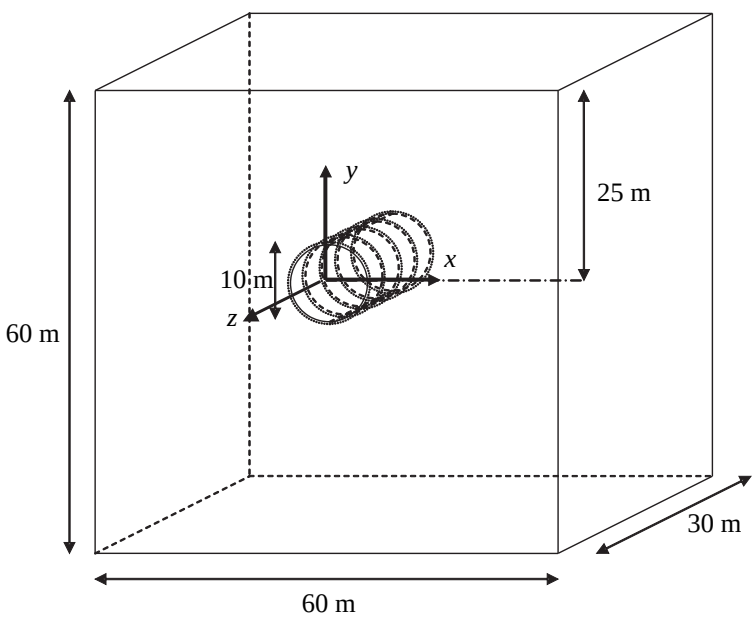

Figure 1.1 Geometry model of tunnel excavation

Objectives:

- Modelling in GiD (for the manual and tutorial lessons of GiD see http://gid.cimne.upc.es/)

- Defining local coordinates for plates and interfaces
- Adding materials to volumes
- Adding materials to plates
- Defining interfaces
- Defining a distributed load
- Defining K0-values
- Activation and inactivation of soil, plates and interfaces
- Modifying the value of a distributed load
- Defining a Phi/c-reduction
- Creating a mesh
- Selection of points to generate curves
- Viewing the results in the output program

1.1 GEOMETRY INPUT

Start the PLAXIS-GiD program. GiD will be opened with an additional data tree for PLAXIS-GiD, see Figure 1.2.

Hint: If this additional data tree is not available make sure that the PLAXIS problem type is selected in the *Data → Problem type → Plaxis1.1* menu. If this problem type is selected and the additional PLAXIS data tree is still not available, press the *Define conditions* button (see Section 3.2 of the Reference Manual).

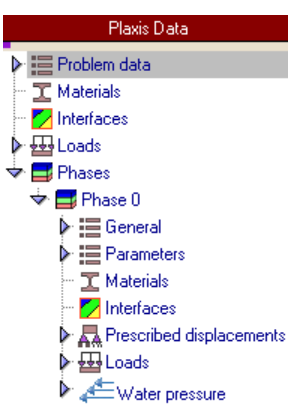

Figure 1.2 Data tree in PLAXIS-GiD

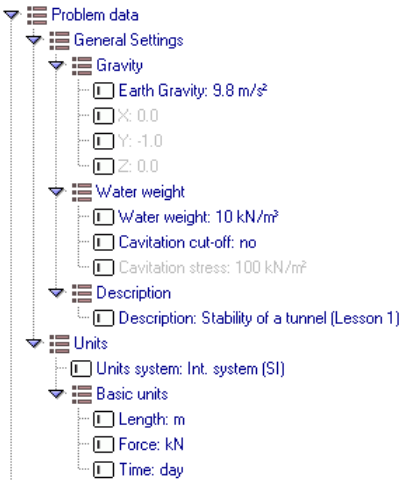

Figure 1.3 *Problem Data* tree in PLAXIS-GiD

Problem data

The first step in every analysis is to set the basic parameters of the finite element model. This is done in the *Problem data* tree, see Figure 1.3. These settings include the description of the problem and the basic units. To enter the appropriate settings for the tunnel calculation follow these steps:

- In the *Units* tree leave the *Units system* to *Int. system(SI)*.
- Leave the *Basic units* to their defaults (unit of *Length* = m; unit of *Force* = kN; unit of *Time* = day).
- Double click the *General Settings* item to open the *General Settings* window (see Figure 1.4). In the *Gravity* tab sheet leave the *Earth gravity* to its default value of 9.8 m/s^2 and the *Water weight* in the *Water weight* tab sheet to its default value of 10 kN/m^3.
- Type "Stability of a tunnel (Lesson 1)" in the *Description* box of the last tab sheet.
- Click the *OK* button to confirm the input.

Modelling the geometry

In GiD a geometry can be modelled by points, lines, surfaces and volumes. Points have to be used to define lines, lines have to be used to define surfaces and surfaces have to be used to define volumes. This manual will not describe the modelling of geometries in the GiD program in detail. Manuals and examples of the GiD program can be found on the website http://gid.cimne.upc.es/.

In order to model the geometry, follow these steps:

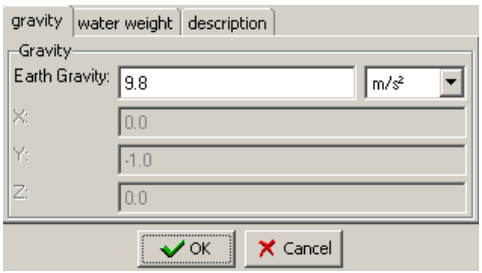

Figure 1.4 The *Project* window

Select the *Create line* in the *Geometry & View* toolbar.

- Enter "30 25 0" in the Command line to define the first point at location (30, 25, 0). Click the <Enter> key to confirm this input. This will create the first point.

As the first point will be created outside the drawing area, it is needed to zoom out. To show everything inside the drawing area select the option *Frame* from the *View* → *Zoom* menu or click the *Zoom frame* button in the toolbar. The first point will be indicated by a black point.

- Enter "30 -35 0" in the Command line to define the second point at (30, -35, 0). Clicking <Enter> will automatically create a second point and a line between both points, indicated by a blue line. Use the *Zoom frame* tool again to show the entire geometry.
- In the same way define the rest of the contour of the geometry by entering the points (-30, -35, 0) and (-30, 25, 0).
- Enter the first point at (30, 25, 0) again to close the contour. The *Create point procedure* window will pop up to ask whether or not two points should be joined. Click the *Join* button to close the contour.
- Press the <Esc> key to stop drawing lines.
- To show the entire geometry again, select the option *Frame* from the *View* → *Zoom* menu or click the *Zoom frame* button in the toolbar.

So far only the contour of the geometry in the *xy*-plane has been created. To model the contour of the tunnel, follow next steps:

- Select the *Point* option from the *Geometry* → *Create* menu.
- Enter "0 5 0" in the Command line to define the top point of the tunnel. Press the <Enter> key to confirm this input.
- Define the bottom of the tunnel at (0, -5, 0) and press <Enter>.
- Press the <Esc> key to stop drawing points.
- To generate a circle out of these newly created points, select the *Copy* option

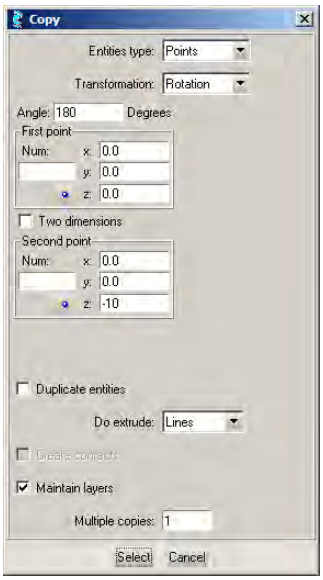

Figure 1.5 *Copy* window

from the *Utilities* menu or press <Ctrl+c>. This will open the *Copy* window (see Figure 1.5).

- Select *Points* as *Entities type* and choose *Rotation* as *Transformation*.
- Enter "180" as the *Angle*.
- As the two nodes of the tunnel should rotate around the *z*-axis, enter the coordinates (0, 0, 0) as the coordinates of the first point and enter the coordinates (0, 0, -10) as the coordinates of the second point.
- Select *Lines* in the *Do extrude* combo box to generate lines.
- Click the *Select* button and select both points of the tunnel by clicking them once. In the GiD program, it is not necessary to hold any key on the keyboard to do a multiple selection.
- Press the <Esc> key or click the *Finish* button to confirm the selection. A circle consisting of two lines will be drawn.
- Close the *Copy* window by clicking the *Cancel* button.

A surface should be created out of these lines to model the tunnel header:

Select the *Create NURBS surface* in the *Geometry & View* toolbar.

- Select both lines of the tunnel and press the <Esc> key. The purple line indicates that a surface has been created.

After creating the surface of the tunnel in the *xy*-plane, the tunnel should be extruded to

the *z*-direction:

- Open the *Copy* window again by choosing this option in the *Utilities* menu.
- Select *Surfaces* as *Entities type*, *Translation* as *Transformation* and enter the coordinates (0, 0, -3) as the coordinates of the second point as one tunnel segment is 3 m long. Select *Volumes* in the *Do extrude* combo box to extrude to volumes.
- Click the *Select* button and select the surface corresponding to the tunnel header. Press <Esc> to confirm the selection.

As the model is still shown in the *xy*-plane, it should be rotated to see the model in three dimensions. Click the *Rotate trackball* to do this. The points, lines, surfaces and volume of the first tunnel segment are now shown.

- Without closing the *Copy* window, click the *Select* option in this window and select the tunnel header surface at z = -3 m. Press the <Esc> key to confirm the selection. Now the entities of the second tunnel segment are created.
- In the same way, create the surfaces of the third and fourth tunnel segment by selecting the surface at z = -6 m and z = -9 m respectively.

Hint: Zooming, rotating and panning can also be performed by using <Shift> key on the keyboard and the mouse. Holding the <Shift> key while holding down the left mouse button will rotate the view, holding the <Shift> key while using the mouse wheel will zoom in and out and holding the <Shift> key while holding the right mouse button will pan.

> When a selection has to be made, e.g. to define a NURBS surface, it is still possible to use the zoom tools. When leaving the zoom tool, the cursor will automatically change back to selection mode.

Hint: To make it easier to deal with a very complex geometry it is very convenient to add volumes, surfaces, lines or nodes to different layers. Select the *Layers* option from the *Utilities* menu or press <Ctrl+l>. More information is given in the manuals provided by GiD. Layers can be created, frozen and turned off independently.

All points, lines, surfaces and volumes of the tunnel are now created. To create the surfaces of the surrounding geometry, follow these steps:

- If the *Copy* window has been closed, open it again. Select *Lines* as *Entities type*, *Translation* as *Transformation* and enter the coordinates (0, 0, -30) as the coordinates of the second point as the total geometry is 30 m long. Select *Lines* in the *Do extrude* combo box to extrude to lines.
- Click the *Select* button and select all four lines at the contour of the geometry. Press <Esc> to confirm the selection and close the *Copy* window.

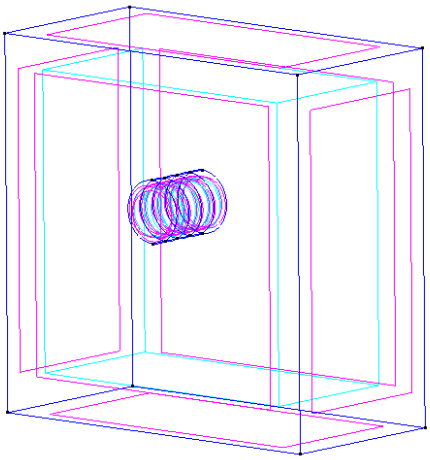

Figure 1.6 Geometry after creating all volumes

- Select the *Create NURBS surface* button, select all lines at y = -35 m and press the <Esc> key to define the bottom surface.
- In the same way define the top surface by selecting all lines at y = 25 m, the left surface by selecting all lines at x = -30 m, the right surface by selecting all lines at x = 30 m, the rear surface by selecting all lines at z = -30 m and the front surface by selecting all lines at z = 0 m including the two tunnel lines.

The next step is to define the volume of the soil around the tunnel:

Click the *Create volume* button to create a volume out of surfaces.

- Select the surfaces at x = -30 m, x = 30 m, y = -35 m, y = 25 m and z = -30 m, the large surface at z = 0 m, all surfaces corresponding to the tunnel contour (8 in total) and the tunnel header at z = -12 m. Press the <Esc> to confirm the selection. Now one volume will be created.

After creating all volumes, the geometry should look like the geometry in Figure 1.6.

Hint: Instead of showing points, lines, surfaces and volumes it is also possible to render the surfaces of a geometry by selecting one of the options from the *Render* menu in the right mouse menu. The option *Normal* will show the points, lines, surfaces and volumes.

Defining materials

After the geometry has been created, the materials have to be defined and added to the corresponding volumes (soil materials), surfaces (plate materials and geogrid materials), lines (beam materials and anchor materials in case of node-to-node anchor) or points

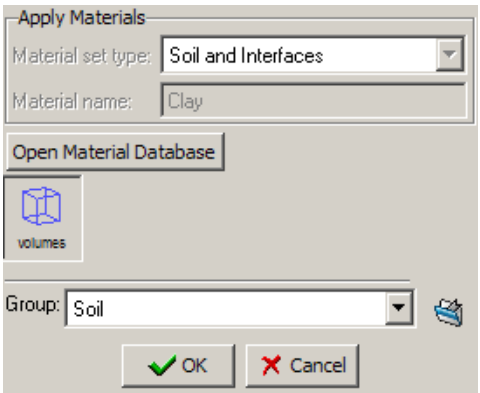

Figure 1.7 The *Apply Materials Database* window

(anchor materials in case of fixed-end anchor):

- Double click at the *Materials* item in the data tree. This will open the *Apply Materials* window, see Figure 1.7.
- First the clay of the soil will be defined. Click the *Open Material Database* button to open the Material Database.
- Click the *New* button and define the material "Clay" according to Table 1.1.
- Close the Material Database.

Table 1.1 Material properties of the soil

Parameter	Name	Clay	Unit
Material model	*Model*	Mohr-Coulomb	-
Type of material behaviour	*Type*	Drained	-
Unit weight of soil	$\gamma_{sat}/\gamma_{unsat}$	16.0	kN/m^3
Young's modulus	E_{ref}	$1 \cdot 10^4$	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion	c	10.0	kN/m^2
Friction angle	φ	30.0	-
Dilatancy angle	ψ	0.0	-
Interface reduction factor	R_{inter}	0.5	-
K_0 determination		Automatic	-

Click the *Create a new group and select entities into it* button to select the volumes this material has to be assigned to. As all entities in a group should have the same properties like material and activation, each segment and the surrounding volume should be part of a different group. Therefore, select only the surrounding soil. Press <Esc> or click again the *Create a new group and select entities into it* button to confirm the selection.

- The window will show the name of the group in blue. Change the name of the group to "Soil".
- Click the *OK* button to confirm the creation of this first group.
- To create a new group, double click the *Materials* option in the data tree. In the same way create a group for the each volume corresponding to a tunnel segment. Assign the "Clay" material and define appropriate names (for example "Segment 1 - soil").

Hint: To see the volumes, surfaces, lines or points in a group select one group (or more groups by holding down the <Ctrl> key) in the data tree and select one of the options in the *Draw* menu of the right mouse menu.

In the same way the material of the tunnel lining modelled by a plate can be defined. However as a plate can have orthotropic properties, first a local coordinate system should be defined:

- Select the *Assign Automatic* option from the *Data* → *Local axes* → *Surfaces* menu.
- Select all 8 surfaces representing the contour of the tunnel.
- Press the <Esc> key to confirm the selection.

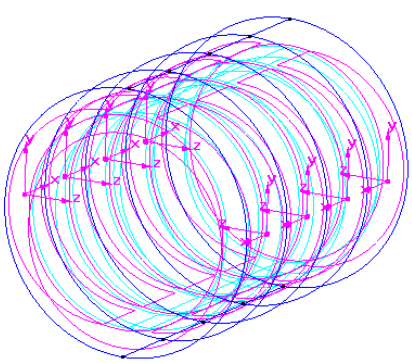

Figure 1.8 The local coordinates at the contour of the tunnel

- The local coordinates can now be shown by selecting the *Draw* option from the same menu, see Figure 1.8.

Hint: To remove a local coordinate system from a surface, select the *Unassign* option from the *Data → Local axes → Surfaces* menu and select the appropriate surface.

The materials for the tunnel lining can be added to the tunnel contour in the same manner as for the soil:

- Double click the *Materials* option in the data tree to generate a new materials group.
- Open the Material Database and select *Plates* as *Set type*.
- Define the "Lining" material as given in Table 1.2.
- Close the material database and select the two surfaces representing the tunnel contour of the first tunnel segment.
- Press the <Esc> key and give this group an appropriate name (for example "Segment 1 - lining").
- Click the *OK* button to create this group.
- In the same way define the other 3 groups representing the tunnel lining of the second, third or fourth segment.

Table 1.2 Material properties of tunnel lining

Parameter	Name	Lining	Unit
Type of material behaviour	*Type*	Isotropic	-
Thickness	d	0.4	m
Weight	γ	16.0	kN/m^3
Young's modulus	E	$3 \cdot 10^7$	kN/m^2
Poisson's ratio	ν	0.15	-
Shear modulus	G	$1.304 \cdot 10^7$	kN/m^2

Defining interface elements

To model the interaction between the soil and the tunnel lining, interface elements have to be added. Looking to the local coordinates of the tunnel contour (see Figure 1.8) the interfaces should be defined in the negative local z-direction.

- Double click the *Interfaces* option in the data tree. The *Apply Interfaces* window will be opened (see Figure 1.9).

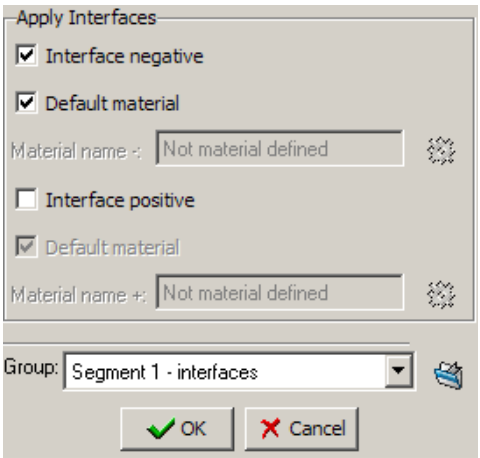

Figure 1.9 The *Apply Interface elements* window

- Select only the *Interface negative* option as only negative interfaces have to be defined.
- Keep the option *Default material* to use the material of the adjacent soil volume for the interface.
- Click the *Create a new group and select entities into it* button and select both surfaces representing the contour of the first tunnel segment.
- Press the <Esc> key to confirm the selection and give the group an appropriate name (for example "Segment 1 - interfaces").
- Click the *OK* button.
- In the same way create three additional groups to represent the interfaces of the other three tunnel segments.

Defining distributed load

Finally, the distributed load on the surfaces between the tunnel segments have to be defined to represent the pressure on the tunnel heading.

- Double click the *Distributed Load* item in the *Loads* subtree to open the *Apply Distributed Load* window (see Figure 1.10).
- Click the *Create a new group and select entities into it* button and select the surface between the first and the second tunnel segment.
- Give the group an appropriate name and click *OK*.
- In the same way, define the distributed load on the surface between the second and third tunnel segment (second distributed load group), the third and fourth

tunnel segment (third distributed load group) and the fourth tunnel segment and the rest of the soil (fourth group).

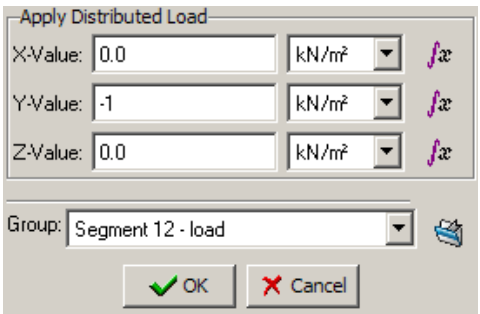

Figure 1.10 The *Apply Distributed Load* window

1.2 PERFORMING CALCULATIONS

Once all groups have been defined, the geometry is complete. Before the actual calculation is started, the calculation stages have to be defined and a finite element mesh has to be created.

When expanding the *Phases* tree *Phase 0* has already been defined. In the *Phases 0* tree all defined groups can be modified, i.e. to activate or inactivate groups, to change the material or to change the value of the load (see Figure 1.11).

Initial conditions

In the initial phase only the soil volumes will be active. In the following calculation phases one tunnel segment will be constructed per phase. Finally, a safety analysis will be performed. To define the initial conditions follow these steps:

- Double click the *General* option in the *Phase 0* tree to open the *General* window (see Figure 1.12). Change the *Calculation type* in the *calculation* tab sheet to *K0 procedure* and click the *OK* button to confirm the input. Using the K_0-procedure, initial stresses will be calculated directly from the soil weight and water pressure. Horizontal effective stresses are based on the K_0-values as specified in the corresponding material data sets (see Defining materials).

As all soil groups will be active by default and all structural groups and interface groups will be inactive by default, no changes are required in the materials and interface groups.

As water will not be considered yet, the phreatic level has to be defined below the geometry model:

- Expand the *Water pressure* subtree in the *Phase 0* tree.

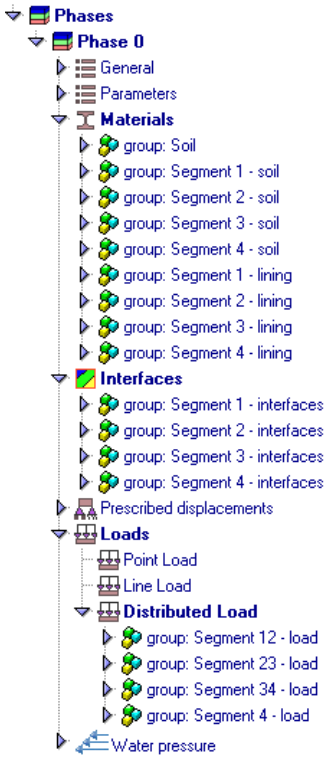

Figure 1.11 The *Phases* tree

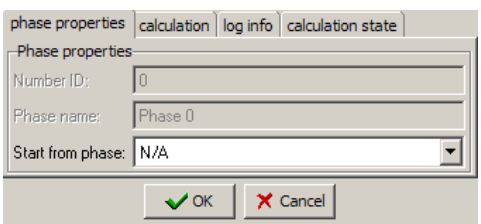

Figure 1.12 The *General* window

Hint:	In addition to change the properties by opening the corresponding window, the properties can also be modified in the data tree directly. Double click the property to be modified and change its value. Press the <Enter> key to confirm the input.

- Double click the *General water level* option. The *General water level* window will be opened (see Figure 1.13).
- Change the *Water level* to -50 m.
- Click the *OK* button to confirm the input.

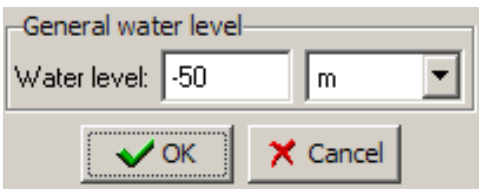

Figure 1.13 The *General water level* window

Installation of the tunnel

After the definition of the initial conditions, the excavation of the first tunnel segment can be modelled. This will be done in a separate calculation phase which needs to be added. To do this, follow these steps:

- Select *Phase 0* in the data tree.
- Select the option *Copy* of the right mouse menu to create phase 1 as a copy of phase 0.
- In the *General* window of the *Phase 1* subtree make sure that *Phase 0* is selected in the *Start from phase* combo box. Make sure that *Plastic* is selected in the *Calculation type* combo box in the *calculation* tab sheet. Click the *OK* button.

In order to model the excavation of the first tunnel segment, the soil of this segment should be de-activated whereas its lining and interfaces should be activated.

- To de-activate the soil in the first tunnel segment double click the second group in the *Materials* subtree ("Segment 1 - soil") to open the *Edit Materials* window (see Figure 1.14). Unselect the option *Active material* and click the *OK* button.
- Activate the lining of the first tunnel segment by double clicking the first plate group ("Segment 1 - lining") and selecting the option *Active material*. Click the *OK* button.
- To activate the negative interface of the first tunnel segment double click at the first interface group to open the *Edit Interfaces* window (see Figure 1.15) and select the option *Active interface* -. Click the *OK* button to confirm the input.
- Double click the first load group in the *Distributed Load* subtree to open its *Edit Distributed Load* window (see Figure 1.16). Select the option *Active load* and change the value in z-direction to -110 kN/m^2 and click *OK*.

The other calculation phases can be defined by copying phases:

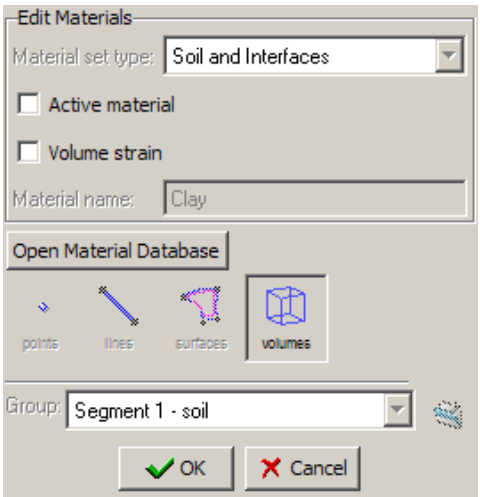

Figure 1.14 The *Edit Materials* window

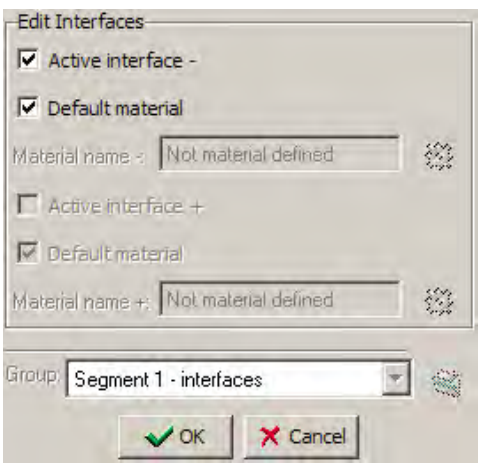

Figure 1.15 The *Edit Interfaces* window

- Copy phase 1 to create phase 2.
- Deactivate the soil in the second tunnel segment and activate both the lining and the negative interface of this tunnel segment.
- Deactivate the load at the surface between the first and second segment and activate and change the load at the surface between the second and third segment to 110 kN/m^2 in the negative z-direction.
- Copy phase 2 to create phase 3 and make the appropriate modifications.

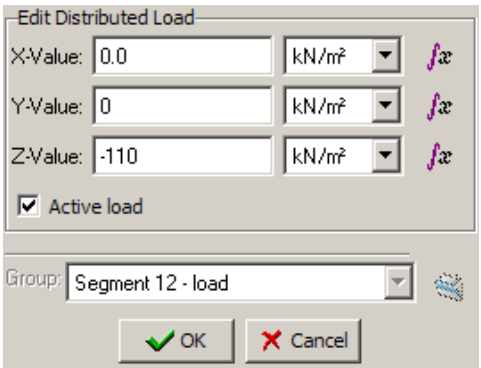

Figure 1.16 The *Edit Distributed Load* window

- Create and define phase 4 in the same way.

Safety analysis

After the construction of all four segments of the tunnel is finished the safety factor of the tunnel has to be determined by doing a phi/c-reduction:

- Copy phase 4 to create phase 5.
- Open the *General* window by double clicking this item in the subtree of phase 5.
- Select *Phi/c-reduction* in the *Calculation type* combo box of the *Calculation* tab sheet and click *OK*. Note that it is not possible anymore to change the staged construction settings.

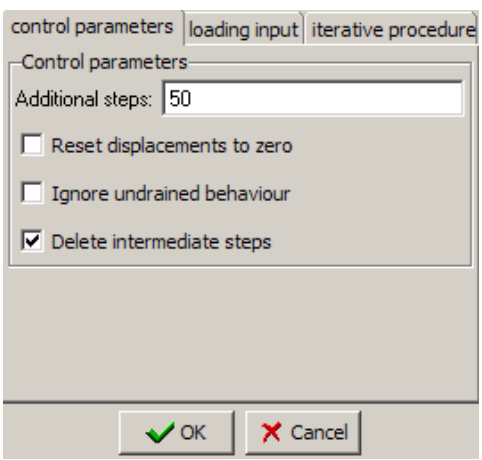

Figure 1.17 The *Parameters* window

- Double click the *Parameters* option to open the *Parameters* window (see Figure 1.17). Change the number of additional steps to 50.

Mesh generation

After all phases have been defined, a finite element mesh has to be created:

- Select the *Generate mesh* from the *Mesh* menu or press <Ctrl+g>. The *Enter value* window will now appear with a suggested element size (see Figure 1.18). Do not change this value and click the *OK* button.
- After the mesh has been created the *Dialog* window will appear showing the number of created triangle elements (corresponding to plate and interface elements), tetrahedral element (corresponding to soil elements) and the total number of nodes (see Figure 1.19). As the mesh will be generated in a non-deterministic way, the number of nodes and elements may be different. Instead of points, lines, surfaces and volumes the created mesh is now shown.

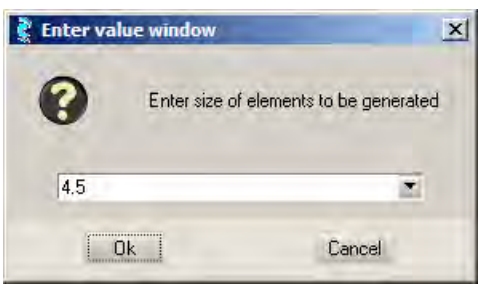

Figure 1.18 The *Enter value* window

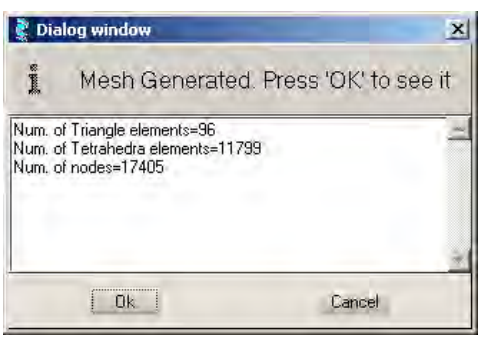

Figure 1.19 The *Dialog* window after generation of the mesh

In addition the mesh around the tunnel contour will be refined a bit:

To toggle back to the geometry view click the *Toggle geometry-mesh view* button

in the toolbar, select the appropriate option from the *View → Mode* menu or press <Ctrl+m>.

- Select *Assign sizes on surfaces* from the *Mesh → Unstructured* menu and change the value to 1.5.
- Click the *Assign* button and select all eight surfaces corresponding to the tunnel contour.
- Press the <Esc> key to confirm the input and close the window by clicking the *Close* button.
- Generate the mesh by selecting this option in the *Mesh* menu or by pressing <Ctrl+g>. Do erase the old mesh and leave the element size to 4.5. The *Dialog* window will now show a larger number of elements.

After the mesh has been created, it is possible to see the preview of each calculation phase, but first the project has to be saved:

Save the project by clicking the *Save GiD project* button in the toolbar or by selecting the *Save* option in the *Files* menu.

- Double click the *Preview / Results* item in the *Phase 4* data tree. Keep the *Mode type* to *Preview* and click the *Output* button. The PLAXIS output program will now be opened, showing a preview of the last calculation phase, see Figure 1.20. It is possible to toggle between all calculation phases by selecting the appropriate option from the *Displayed step* combo box to check the preview of all phases.

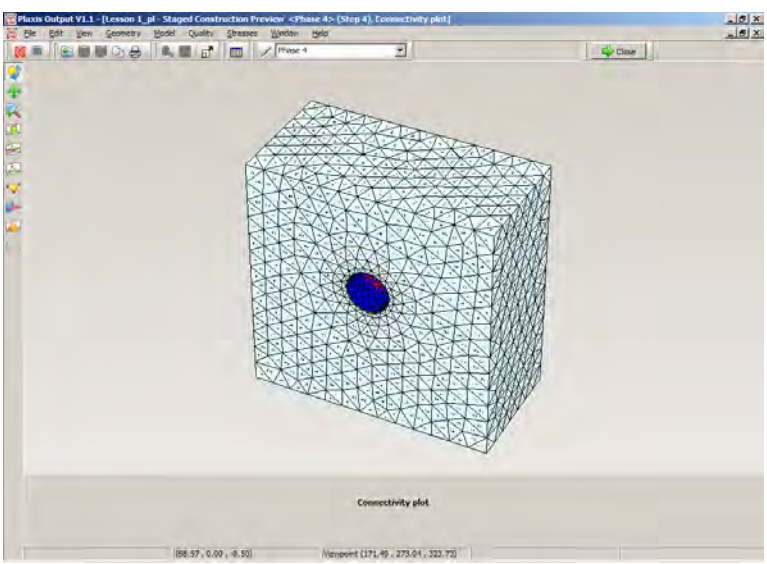

Figure 1.20 Preview of phase 4

Hint: If a calculation has failed, log information about this failure will be displayed in the *Log info* tab sheet of the *General* window in the subtree of this calculation phase.

- Close the Output program by clicking the *Close* button.

Before the calculation can be started it will be useful to select some points for the generation of curves:

- Double click the *Mesh / Point selection* item at the top of the *Phases* data tree and select the *Point Selection for Curves* option as *Mode type*. Click the *Output* button to open the Output program.
- Select *Mesh Point Selection* from the *Edit* menu.
- Enter the coordinates (0, 0, -12) and click *Search closest node and stress point*.
- Click the *Select node* button and close the window.
- Close the Output program by clicking the *Update* button.

Starting the calculation

After all calculation phases have been defined, a mesh is created and the definition of the calculation phases have been checked, the calculation can be started:

Click the *Phase manager* button in the toolbar or select the *Phase manager* from the *Calculate* menu to open the *Phase Manager* window (see Figure 1.21). This window will show the status of a calculation phase. A blue arrow indicates that the phase is marked for calculation, whereas a white arrow indicates that the phase is not marked for calculation. After calculation a green tick will appear if the calculation has been finished successfully or a red cross if the calculation has failed. The status of a phase can be changed by clicking this item. Make sure all phases are marked for calculation.

- Close the *Phase Manager* window by clicking the *OK* button.

Click the *Start calculation* button in the toolbar or select the *Calculate* option from the *Calculate* menu in order to start the calculation.

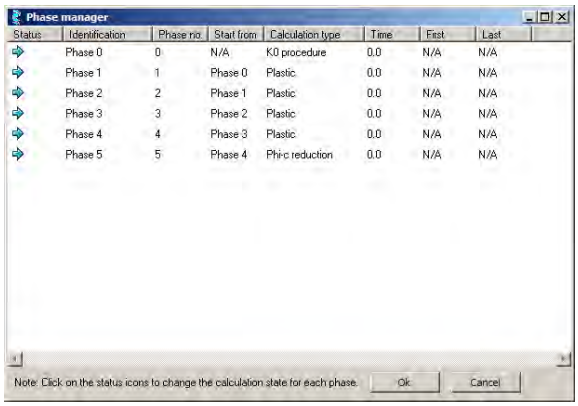

Figure 1.21 The *Phase Manager* window

1.3 VIEWING OUTPUT RESULTS

After the calculation has been finished successfully, the results can be evaluated. As the PLAXIS Output program will be used, it will not be discussed here in detail.

- Double click the *Preview / Results* item in the *Phase 4* subtree and make sure that the option *Results* is selected in the *Mode type* combo box. Click the *Output* button to open the results of phase 4. The deformed mesh will now be shown (see Figure 1.22).

To generate the displacement - safety factor curve, click the *Curves Manager* in the toolbar, select the *Curves* option from the *File* menu or press the <F2> key on the keyboard.

- Click *New* in the *Curves Manager* window. Select the pre-selected node in the combo box of the *X-Axis* and select the option *Deformations* → *Total Displacements* → $|u|$ as input for the x-axis. For the y-axis, keep the combo box at *Project* and select *Multiplier* → $\sum Msf$ as input. Click the *OK* button to generate the curve (see Figure 1.23).

Hint: Clicking at a plate or interface element will select all plate or interface elements which are connected to each other and have the same local coordinate system. In order to select all plate or interface elements at once, click one element first and press <Ctrl+a>. Holding the <Ctrl> key while double clicking an element will open all structural elements of the selected type in a new window.

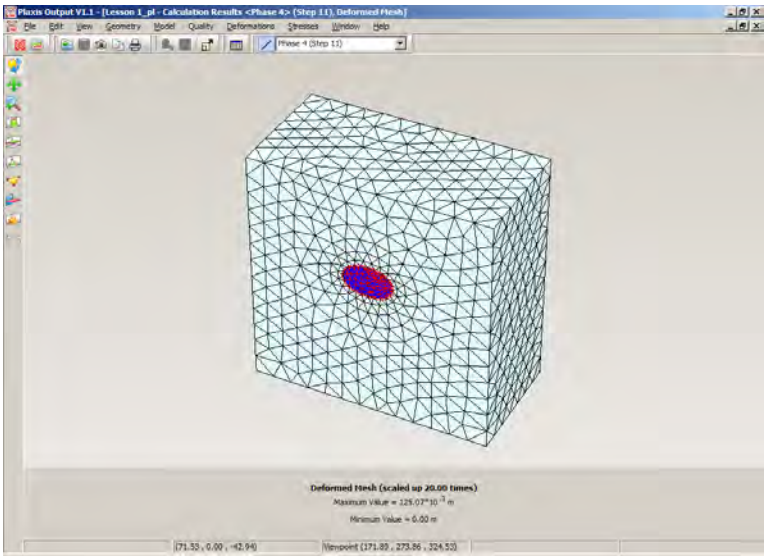

Figure 1.22 The deformed mesh

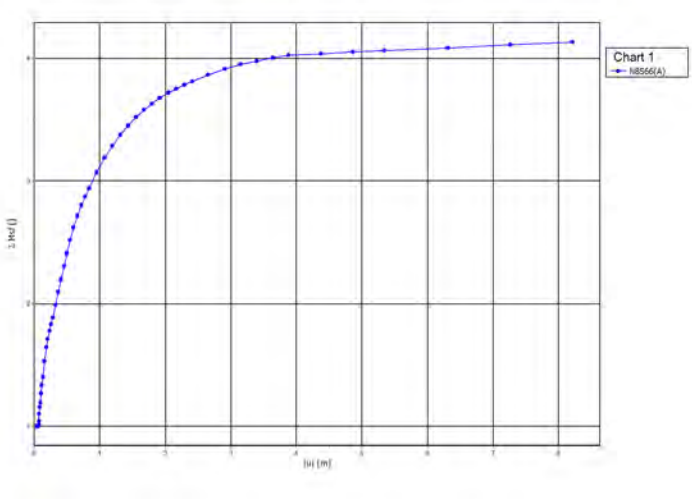

Figure 1.23 Displacement - safety factor curve