# TABLE OF CONTENTS

1. **Introduction** ................................................................. 1 - 1

2. **Getting started** ............................................................... 2 - 1
   2.1 Installation ................................................................. 2 - 1
   2.2 General modelling aspects ............................................. 2 - 1
   2.3 Input procedures .......................................................... 2 - 3
      2.3.1 Input of geometry objects ....................................... 2 - 3
      2.3.2 Input of text and values ......................................... 2 - 3
      2.3.3 Input of selections ................................................ 2 - 4
      2.3.4 Structured input .................................................... 2 - 5
   2.4 Starting the program ................................................... 2 - 6
      2.4.1 General settings .................................................... 2 - 6
      2.4.2 Creating a geometry model ..................................... 2 - 8

3. **Settlement of circular footing on sand (Lesson 1)** ............... 3 - 1
   3.1 Geometry ................................................................. 3 - 1
   3.2 Rigid footing ............................................................. 3 - 2
      3.2.1 Creating the input ................................................ 3 - 2
      3.2.2 Performing calculations ........................................ 3 - 14
      3.2.3 Viewing output results ......................................... 3 - 18
   3.3 Flexible footing .......................................................... 3 - 21

4. **Submerged construction of an excavation (Lesson 2)** ............ 4 - 1
   4.1 Geometry ................................................................. 4 - 2
   4.2 Calculations .............................................................. 4 - 11
   4.3 Viewing output results ................................................ 4 - 14

5. **Undrained river embankment (Lesson 3)** .............................. 5 - 1
   5.1 Geometry model ......................................................... 5 - 1
   5.2 Calculations .............................................................. 5 - 4
   5.3 Output ..................................................................... 5 - 9

6. **Dry excavation using a tie back wall (Lesson 4)** .................... 6 - 1
   6.1 Input ..................................................................... 6 - 1
   6.2 Calculations .............................................................. 6 - 5
   6.3 Output ..................................................................... 6 - 9

7. **Construction of a road embankment (Lesson 5)** .................... 7 - 1
   7.1 Input ..................................................................... 7 - 1
   7.2 Calculations .............................................................. 7 - 4
   7.3 Output ..................................................................... 7 - 5
   7.4 Safety analysis .......................................................... 7 - 7
8 Construction of a shield tunnel (Lesson 4) .......................................................8 - 1
8.1 Geometry ........................................................................................................8 - 2
8.2 Calculations ....................................................................................................8 - 6
8.3 Output ...........................................................................................................8 - 7

A Appendix A - Menu tree ...................................................................................A - 1
A.1 Input menu .....................................................................................................A - 1
A.2 Calculations menu ..........................................................................................A - 2
A.3 Output menu ...................................................................................................A - 3
A.4 Curves menu ..................................................................................................A - 4

B Appendix B - Calculation scheme for initial stresses due to soil weight ...........................................B - 1
8 CONSTRUCTION OF A SHIELD TUNNEL (LESSON 6)

PLAXIS has special facilities for the generation of circular and non-circular tunnels and the simulation of a tunnel construction process. In this chapter the construction of a shield tunnel in medium soft soil is considered. A shield tunnel is constructed by excavating soil at the front of a tunnel boring machine (TBM) and installing a tunnel lining behind it. In this procedure the soil is generally over-excavated, which means that the cross sectional area occupied by the final tunnel lining is always less than the excavated soil area. Although measures are taken to fill up this gap, one cannot avoid stress redistributions and deformations in the soil as a result of the tunnel construction process. In order to avoid damage to existing buildings or foundations on the soil above, it is necessary to predict these effects and to take proper measures. Such an analysis can be performed by means of the finite element method. This lesson shows an example of such an analysis.

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

The tunnel considered in this lesson has a diameter of 5.0 m and is located at an average depth of 20 m. The soil profile indicates four distinct layers: The upper 13 m consists of soft clay type soil with stiffness that increases approximately linearly with depth. Under the clay layer there is a 2.0 m thick fine sand layer. This layer is known as a foundation layer for old wooden piles on which traditional houses were built. Deformations of this layer may cause damage to traditional houses, which is highly undesirable. Below the sand layer there is a 5.0 m thick deep loamy clay layer.
This is one of the layers in which the tunnel is constructed. The other part of the tunnel is constructed in the deep sand layer, which consists of dense sand and some gravel. This layer is very stiff. As a result only 5.0 m of this layer is included in the finite element model; the deeper part is considered to be fully rigid and modelled by appropriate boundary conditions. The pore pressure distribution is hydrostatic. The phreatic line is located 3 m below the ground surface (at a level of y = 0 m). Since the situation is more or less symmetric, only one symmetric half (the right half) is taken into account in the plane strain model. From the center of the tunnel the model extends for 20 m in horizontal direction. The 15-node element is adopted for this example.

8.1 GEOMETRY

The basic geometry including the four soil layers, as shown in Fig. 8.1 (but excluding the tunnel), can be created using the geometry line option. Since the ground surface is located at 3.0 m above the reference level, the Top parameter is taken at +3.0 m in the General settings and the Bottom at -22.0 m. For the generation of the tunnel we will use the tunnel designer, which is a special tool within PLAXIS that enables the use of circle segments to model the geometry of a tunnel. The tunnel considered here is the right half of a circular tunnel and will be composed of four segments. After generating the basic geometry, follow these steps to design the circular tunnel:

- Click on the Tunnel button in the toolbar. A window appears with three options for creating tunnel shapes. Select Half a tunnel - Right half and press the <OK> button.
- The tunnel designer appears, as indicated in Fig. 8.2, showing a default (half) tunnel shape composed of three sections of which the lower one (Section 1) is selected. The right side of the window shows some geometrical values.
- Make sure that the lower tunnel section is selected (if not, select it by clicking with the mouse in the lower section).
- The upper value in the table represents the tunnel radius. Enter a radius of 2.5 m. The result of this action is directly visible in the drawing.
- The second value represents the angle over which the section extends. Enter an angle of 90 degrees (which is the maximum angle of one section).
- The local x-coordinate of the first arc center point is always located at the local y-axis (x=0). The local y-coordinate can be specified. Use the default value of y = 0.
- Select the options Tunnel lining and Tunnel interface.
- Proceed to the next section (1). When pressing the <Enter> key after each input value, the program guides you to the next section. Alternatively, you may click on the upper tunnel section.
- Enter a radius of 2.5 m and an angle of 90 degrees. Except for the first arc center point, all other center points are calculated by the program and are not accessible.
The current tunnel has coinciding center points in the origin of the local system of axes.

- Select for Section 1 the *Tunnel lining* and *Tunnel interface* options.

![Tunnel designer with default model of a right tunnel half](image)

**Figure 8.2** Tunnel designer with default model of a right tunnel half

**Hints:**

- A lining 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 beams. The lining properties can be specified in the material database for beams. 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 tunnels in rock).
Below the Section group box there are three parameters. The Symmetric tunnel parameter is always selected for half tunnels. The Contraction parameter can be used to simulate the volume loss in the soil due to the construction of the tunnel. This procedure can be activated in a plastic calculation (see 8.2 Calculations). Select the Contraction parameter and enter an input value of 1.0 percent. The Homogeneous lining parameter is now automatically selected.

Hints: Activation of the contraction procedure during a plastic calculation results in a homogeneous ‘shrinkage’ of the tunnel lining, which reduces the cross section area of the tunnel. The Contraction parameter is defined as the reduction of the tunnel area as a percentage of the original tunnel area. A contraction can only be specified for circular tunnels with a homogeneous tunnel lining.

The tunnel has now been completely defined. Press the <OK> button to close the tunnel designer.

Back in the draw area, the tunnel must be included in the geometry model. This is done by entering the global position of the origin of the local tunnel axes. Click on the existing point at position (0.0; -17.0) (5.0 m above the bottom of the geometry model). The tunnel will be drawn and indicated by an ‘A’ in the center point. The ‘A’ actually refers to the contraction rather than to the tunnel.

Boundary conditions

Click on the Standard fixities button to apply the appropriate boundary conditions. In addition to the standard displacement fixities, fixed rotations are introduced to the upper and lower point of the tunnel lining.

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

Material properties

The material properties for the four different soil layers are listed in Table 8.1. For all layers the material behaviour is set to drained since we are interested in the long term deformations.
For the upper clay layer we use the advanced option to let the stiffness increase with depth. Therefore a value of $E_{\text{increment}}$ is entered in the Advanced parameters window. The value of $E_{\text{ref}}$ becomes a reference value at the reference level $y_{\text{reference}}$. The actual value of $E$ increases with depth according to: $E(y) = E_{\text{ref}} + E_{\text{increment}}(y_{\text{reference}} - y)$.

The data sets of the two lower soil layers include appropriate parameters for the tunnel interfaces. In the other data sets the interface properties just remain at their default values. Enter four data sets with the properties as listed in Table 8.1 and assign them to the corresponding clusters in the geometry model. In order to enter the advanced parameters for the Clay data set, click on the <Advanced> button in the Parameters tab sheet.

### Table 8.1. Material properties of soil in the tunnel project

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Clay</th>
<th>Sand</th>
<th>Dp. Clay</th>
<th>Dp.sand</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material model</td>
<td>Model</td>
<td>MC</td>
<td>MC</td>
<td>MC</td>
<td>MC</td>
<td>-</td>
</tr>
<tr>
<td>Material beh.</td>
<td>Type</td>
<td>Drained</td>
<td>drained</td>
<td>Drained</td>
<td>drained</td>
<td>-</td>
</tr>
<tr>
<td>Dry soil weight</td>
<td>$\gamma_{\text{dry}}$</td>
<td>15</td>
<td>16.5</td>
<td>16</td>
<td>17</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>Wet soil weight</td>
<td>$\gamma_{\text{wet}}$</td>
<td>18</td>
<td>20</td>
<td>18.5</td>
<td>21</td>
<td>kN/m$^3$</td>
</tr>
<tr>
<td>H. permeability</td>
<td>$k_x$</td>
<td>$10^{-4}$</td>
<td>1.0</td>
<td>$10^{-3}$</td>
<td>0.5</td>
<td>m/day</td>
</tr>
<tr>
<td>V. permeability</td>
<td>$k_y$</td>
<td>$10^{-4}$</td>
<td>1.0</td>
<td>$10^{-3}$</td>
<td>0.5</td>
<td>m/day</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>$E_{\text{ref}}$</td>
<td>1000</td>
<td>80000</td>
<td>10000</td>
<td>120000</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Increase $E$</td>
<td>$E_{\text{incr}}$</td>
<td>650</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Reference level</td>
<td>$y_{\text{ref}}$</td>
<td>0.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>m</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu$</td>
<td>0.33</td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>Cohesion</td>
<td>$c_{\text{ref}}$</td>
<td>5.5</td>
<td>1.0</td>
<td>4.0</td>
<td>1.0</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Friction angle</td>
<td>$\phi$</td>
<td>24</td>
<td>31</td>
<td>25</td>
<td>33</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle</td>
<td>$\psi$</td>
<td>0.0</td>
<td>1.0</td>
<td>0.0</td>
<td>3.0</td>
<td>°</td>
</tr>
<tr>
<td>Interface strength</td>
<td>$R_{\text{inter}}$</td>
<td>rigid</td>
<td>rigid</td>
<td>0.7</td>
<td>0.7</td>
<td>-</td>
</tr>
<tr>
<td>Interface perm.</td>
<td>Perm</td>
<td>neutral</td>
<td>neutral</td>
<td>Imperm</td>
<td>Imperm</td>
<td>-</td>
</tr>
</tbody>
</table>

### Table 8.2. Material properties of the tunnel lining (beam)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of behaviour</td>
<td>Material type</td>
<td>Elastic</td>
<td>kN/m</td>
</tr>
<tr>
<td>Normal stiffness</td>
<td>$EA$</td>
<td>$1.4 \cdot 10^7$</td>
<td>kN/m</td>
</tr>
<tr>
<td>Flexural rigidity</td>
<td>$EI$</td>
<td>$1.43 \cdot 10^4$</td>
<td>kN/m/m</td>
</tr>
<tr>
<td>Equivalent thickness</td>
<td>$d$</td>
<td>0.35</td>
<td>m</td>
</tr>
<tr>
<td>Weight</td>
<td>$w$</td>
<td>8.4</td>
<td>kN/m/m</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>$\nu$</td>
<td>0.15</td>
<td>-</td>
</tr>
</tbody>
</table>
In addition to the four data sets for the soil and interfaces, a beam data set has to be created with the properties of the tunnel lining. These properties are listed in Table 8.2. Assign the data set to the tunnel lining.

**Mesh generation**

In this example the 15-node element is used as the basic element type. This means that the mesh is more accurate than when using the 6-node element. The global coarseness parameter can remain at its default value (Coarse). It can be expected that stress concentrations occur around the tunnel. Therefore the mesh should be refined in this area. This can be done by selecting the two clusters inside the tunnel and using the *Refine cluster* option in the *Mesh* menu.

**Initial conditions**

The water weight should be taken 10 kN/m³. The water pressures can be generated on the basis of a general phreatic line at a level of y = 0.0 m.

Before the generation of the initial stresses the tunnel lining should be deactivated. In addition, the *K₀-procedure* can be used to generate the initial effective stresses with the appropriate values of *K₀*.

**8.2 CALCULATIONS**

In order to simulate the construction of the tunnel it is clear that a staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. Deactivating the soil inside the tunnel only affects the soil stiffness and strength and the effective stresses. Without additional input the water pressures remain! In order to remove the water pressure inside the tunnel a low user-defined phreatic line should be introduced for the two soil clusters in the tunnels and the water pressures should be regenerated. To create this input, follow these steps.

- The first calculation phase is a plastic calculation, load advancement ultimate level. For the loading input, select *Staged construction* and click on the define button.
- Within the staged construction mode, activate the tunnel lining and deactivate the two soil clusters inside the tunnel.
- Click on the 'switch' to proceed to the water pressures mode. Click on the *Selection* button and select both soil clusters inside the tunnel simultaneously (using the Shift key). Click on the *Phreatic line* button while the clusters remain selected and draw a phreatic line at the bottom of the geometry model (or at least below the tunnel).
• Click on the Generate water pressures button to generate the water pressures. In the resulting plot it can be seen that there are indeed no water pressures inside the tunnel. Click on the <Update> button to return to the water pressures mode.

• Within the water pressures mode, click on the <Update> button to return to the calculations window.

In addition to the installation of the tunnel lining, the excavation of the soil and the de-watering of the tunnel, the volume loss is simulated by applying a contraction to the tunnel lining. This contraction was defined during the creation of the tunnel in the input program. In order to activate this contraction, follow these steps:

• Click on the <Next> button to introduce a next calculation phase.
• Select a plastic calculation, load advancement ultimate level and select Total multipliers as loading input.
• In the Multipliers tab sheet, enter a value of 2.0 for ΣMcontrA. This is the multiplier that controls the contraction of the tunnel referred to as 'A' in the geometry model.
• Select some characteristic points for load-displacement curves (for example the corner point at the ground surface above the tunnel).
• Start the calculations.

Hint: The contraction of the tunnel lining by itself does not introduces 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.

8.3 OUTPUT

After the calculation, select the two calculation phases and click on the <Output> button. The Output program is started, showing the deformed meshes at the end of the calculation phases.

As a result of the first calculation phase (removing soil and water out of the tunnel) there is some settlement of the soil surface and the tunnel lining shows some deformation. In this phase the axial force in the lining is the maximum axial force that will be reached. The lining forces can be viewed by double clicking the lining and selecting the options from the Force menu (see Fig. 8.3 and 8.4).
The second calculation phase shows the results due to the simulation of the volume loss. The deformed mesh indicates a settlement trough at the ground surface (see Fig. 8.5). The plot of effective stresses shows that arching occurs around the tunnel. This arching reduces the stresses acting on the tunnel lining. As a result, the axial force is lower than after the first calculation phase. The bending moments, however, are larger.
Figure 8.6  Axial forces in the lining after the second phase

Figure 8.7  Bending moments in the lining after the second phase