6 PHASED EXCAVATION OF A SHIELD TUNNEL

The lining of a shield tunnel is often constructed using prefabricated concrete ring segments, which are bolted together within the tunnel boring machine to form the tunnel lining. During the erection of the tunnel lining the tunnel boring machine (TBM) remains stationary. Once a tunnel lining ring has been fully erected, excavation is resumed, until enough soil has been excavated to erect the next lining ring. As a result, the construction process can be divided into construction stages with a length of a tunnel ring, often about 1.5 m long. In each of these stages, the same steps are repeated over and over again.

In order to model this, a geometry consisting of slices each 1.5 m long can be used. The calculation consists of a number of *Plastic* phases, each of which models the same parts of the excavation process: the support pressure at the tunnel face needed to prevent active failure at the face, the conical shape of the TBM shield, the excavation of the soil and pore water within the TBM, the installation of the tunnel lining and the grouting of the gap between the soil and the newly installed lining. In each phase the input for the calculation phase is identical, except for its location, which will be shifted by 1.5 m each phase.

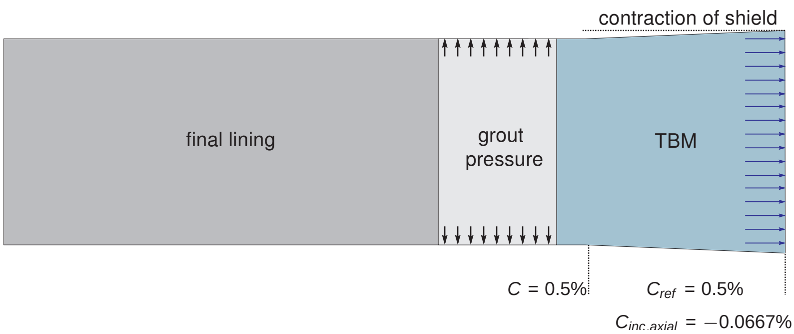

Figure 6.1 Construction stages of a shield tunnel model

Objectives:

- Modelling of the tunnel boring process with a TBM
- Modelling of the cone shape of the TBM
- Using *Tunnel designer* to define geometry, trajectory and sequencing of the tunnel

6.1 GEOMETRY

In the model, only one symmetric half is included. The model is 20 m wide, it extends 80 m in the y-direction and it is 20 m deep. These dimensions are sufficient to allow for any possible collapse mechanism to develop and to avoid any influence from the model boundaries.

When starting PLAXIS 3D set the proper model dimensions in the *Project properties* window, that is x_{min} = -20, x_{max} = 0, y_{min} = 0 and y_{max} = 80.

6.1.1 DEFINITION OF SOIL STRATIGRAPHY

The subsoil consists of three layers. The soft upper sand layer is 2 m deep and extends from the ground surface to Mean Sea Level (MSL). Below the upper sand layer, there is a clay layer of 12 m thickness and this layer is underlain by a stiff sand layer that extends to a large depth. Only 6 m of the stiff sand layer is included in the model. Hence, the bottom of the model is 18 m below MSL. Soil layer is assumed to be horizontal throughout the model and so just one borehole is sufficient to describe the soil layers. The present groundwater head corresponds to the MSL.

- Press the *Create borehole* button and click at the origin of the system of axis to create a borehole at (0 0 0). The *Modify soil layers* window will open.
- Define 3 layers: Upper sand with the top at 2 m and the bottom at 0 m, Clay with the bottom at -12 m and Stiff sand with the bottom at -18 m.
- Open the materials database by clicking the *Materials* button and create the data sets for the soil layers and the final concrete lining in the tunnel as specified in Table 6.1.

Table 6.1 Material properties for the soil layers

Parameter	Name	Upper sand	Clay	Stiff sand	Concrete	Unit
General						
Material model	Model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Linear elastic	–
Drainage type	Type	Drained	Drained	Drained	Non porous	–
Unit weight above phreatic level	γ_{unsat}	17.0	16.0	17.0	27.0	kN/m^3
Unit weight below phreatic level	γ_{sat}	20.0	18.0	20.0	–	kN/m^3
Parameters						
Young's modulus	E'	$1.3 \cdot 10^4$	$1.0 \cdot 10^4$	$7.5 \cdot 10^4$	$3.1 \cdot 10^7$	kN/m^2
Poisson's ratio	ν'	0.3	0.35	0.3	0.1	–
Cohesion	c'_{ref}	1.0	5.0	1.0	–	kN/m^2
Friction angle	φ'	31	25	31	–	°
Dilatancy angle	ψ	0	0	0	–	°
Interfaces						
Interface strength	–	Rigid	Rigid	Rigid	Rigid	–
Initial						
K_0 determination	–	Automatic	Automatic	Automatic	Automatic	–

- Assign the material data sets to the corresponding soil layers (Figure 6.2) and close the *Modify soil layers* window. The concrete data set will be assigned later.

6.1.2 DEFINITION OF STRUCTURAL ELEMENTS

The tunnel excavation is carried out by a tunnel boring machine (TBM) which is 9.0 m long and 8.5 m in diameter. The TBM already advanced 25 m into the soil. Subsequent phases will model an advancement by 1.5 m each.

Create tunnel

In *Structures* mode both the geometry of the tunnel and the TBM will be defined.

- Click the *Create tunnel* button in the side toolbar.
- Click anywhere on the drawing area to define the insertion point. The *Tunnel*

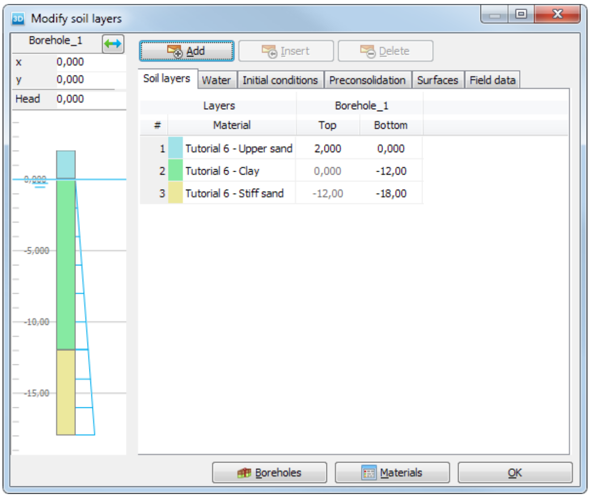

Figure 6.2 Soil layer distribution

designer window pops up.

- In the *Selection explorer* set the insertion point of the tunnel to (0 0 -13.25) (Figure 6.3).
- In the *General* tabsheet select the *Circular* option in the drop-down menu for the *Shape type*.
- The left half of the tunnel is generated in this example. Select the *Define left half* option in the drop-down menu for the *Whole or half tunnel*. A screenshot of the *General* tabsheet after the proper assignment is given in Figure 6.4.
- Click the *Segments* tabsheet to proceed to the corresponding tabsheet. A segment is automatically created. A new box is shown under the segment list where the properties of the segment can be defined.
- In the *Segment* box set *Radius* to 4 m. This is the inner radius of the tunnel.
- Proceed to the *Subsections* tabsheet.

Click the *Generate thick lining* button in the side toolbar. The *Generate thick lining* window pops up.

- Assign a value of 0.25 m and click *OK*. A screenshot of the *Cross section* tabsheet after the proper assignment is given in Figure 6.5.
- Proceed to the *Properties* tabsheet. Here we define the properties for the tunnel such as grout pressure, surface contraction, jack forces and the tunnel face pressure.
- In the *Slice* tabsheet, right-click the outer surface and select *Create plate* from the appearing menu (Figure 6.6).

Click on the *Material* in the lower part of the explorer. Create a new material dataset. Specify the material parameters for the TBM according to Table 6.2.

A soil-structure interaction has to be added on the outside of the tunnel due to the slight cone shape of the TBM. Typically, the cross-sectional area at the tail of the TBM is about

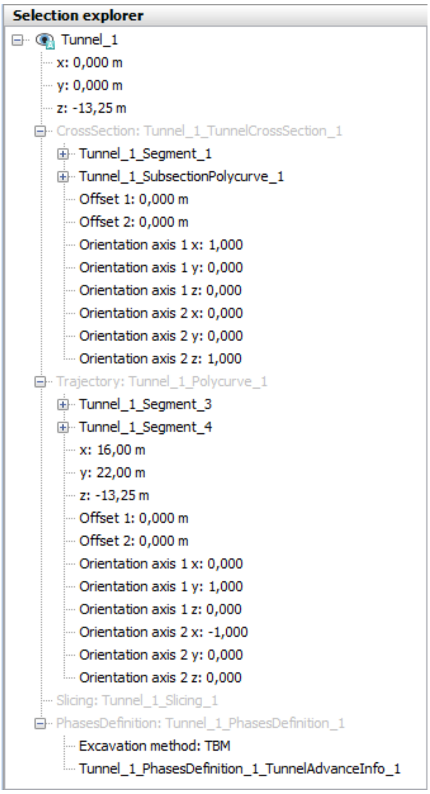

Figure 6.3 Insertion point of the tunnel

Hint: In the tunnel, as considered here, the segments do not have a specific meaning as the tunnel lining is homogeneous and the tunnel will be constructed at once. In general, the meaning of segments becomes significant when:

- It is desired to excavate or construct the tunnel (lining) in different stages.
- Different tunnel segments have different lining properties.
- One would consider hinge connections in the lining (hinges can be added after the design of the tunnel in *Staged construction* mode, Section 7.9.4 of the Reference Manual).
- The tunnel shape is composed of arcs with different radii (e.g. NATM tunnels).

Table 6.2 Material properties of the plate representing the TBM

Parameter	Name	TBM	Unit
Type of behaviour	-	Elastic; Isotropic	-
Thickness	d	0.17	m
Material weight	γ	247	kN/m^3
Young's modulus	E_1	$200 \cdot 10^6$	kN/m^2
Poisson's ratio	ν_{12}	0	-
Shear modulus	G_{12}	$100 \cdot 10^6$	kN/m^2

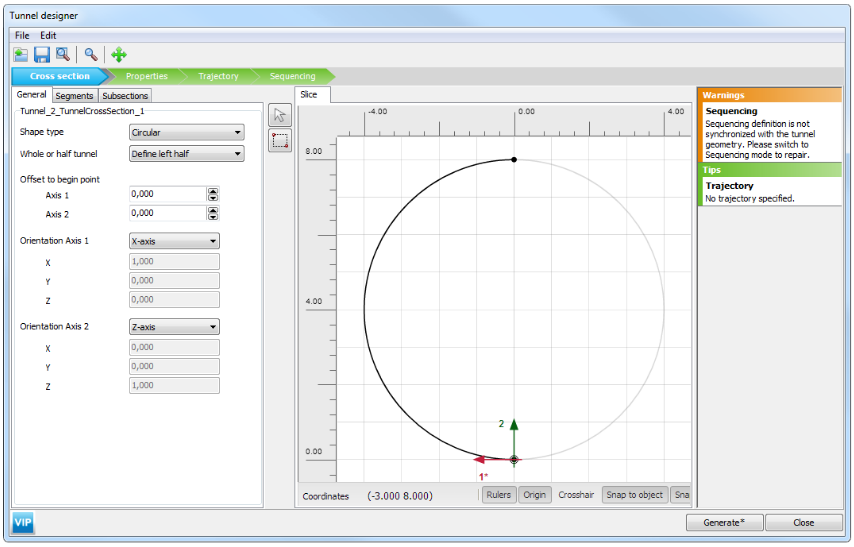

Figure 6.4 *General* tabsheet of the *Tunnel designer*

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

0.5% smaller than the front of the TBM. The reduction of the diameter is realised over the first 7.5 m length of the TBM while the last 1.5 m to the tail has a constant diameter. This means that the section tail has a uniform contraction of 0.5% and the remaining 5 sections have a linear contraction with a reference value C_{ref} = 0.5% and an increment $C_{inc,axial}$ = -0.0667%. The reference is set on the front surface of the excavated slice in the tunnel during the tunnel construction. This is done while setting the Sequencing steps. The $C_{inc,axial}$ = -0.0667%/m and remains the same in every step (1_1 to 1_5). For further information on *Surface contraction* refer Section 5.5.4 of the Reference Manual.

- Right-click the same outer surface and select *Create negative interface* from the appearing menu to create a negative surface around the entire tunnel.
- Next step is to create *Surface contraction* for the tunnel. Right-click the outer surface and select *Create surface contraction*.
- In the properties box, select the *Axial increment* option for the contraction distribution and define C_{ref} = 0.5% and $C_{inc,axial}$ = -0.0667%/m. The increment must be a negative number because the contraction decreases in the direction of the positive local *1-axis*. The reference location is (0 28 0).

Grout pressure

The surface load representing the grout pressure is constant during the building process. In the specifications of the tunnel boring process, it is given that the grout pressure should be -100 kN/m^2 at the top of the tunnel (z = -4.75) and should increase with -20 kN/m^2/m depth. To define the grout pressure:

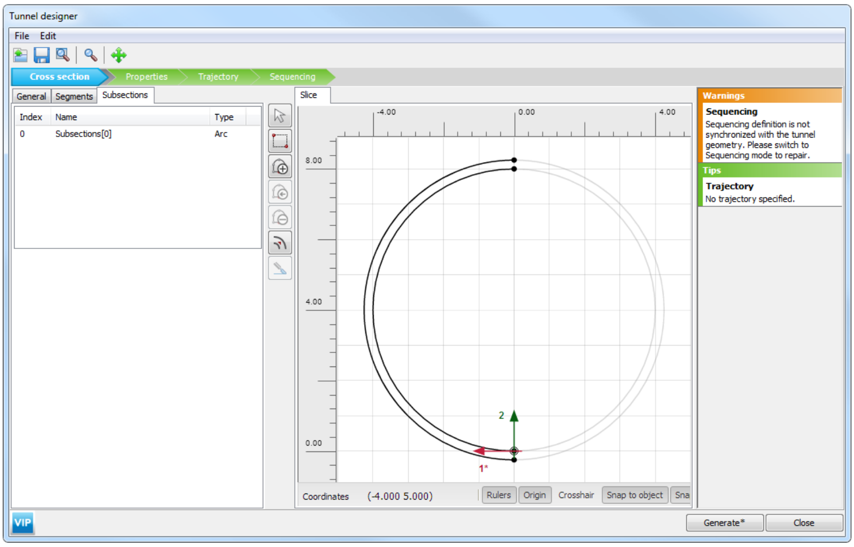

Figure 6.5 The *Cross section* tabsheet of the *Tunnel designer*

Hint: A surface contraction of the tunnel contour of 0.5% corresponds approximately to a volume loss of 0.5% of the tunnel volume (applicable only for small values of surface contractions).

» The entered value of contraction is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

- Right-click the outer surface and select *Create surface load* from the appearing menu to create a surface load around the entire tunnel.
- In the properties box, select *Perpendicular, vertical increment* from the drop-down menu for *Distribution*.
- Set the $\sigma_{n,ref}$ to -100 and $\sigma_{n,inc}$ to -20 and define (0 0 -4.75) as the reference point for the load by assigning the values to x_{ref}, y_{ref} and z_{ref} (Figure 6.7).

Tunnel face pressures

The tunnel face pressure is a bentonite pressure (Bentonite Slurry, BS) or an earth pressure (Earth Pressure Balance, EPB) that increases linearly with depth. For the initial position of the TBM and the successive four positions when simulating the advancement of the TBM, a tunnel face pressure has to be defined.

- Select the *Plane* tabsheet above the displayed tunnel cross section.
- Multi-click both the surfaces, right-click and select *Create surface load* from the appearing menu to create a surface load around the entire tunnel.
- In the properties box, select *Perpendicular, vertical increment* from the drop-down menu for *Distribution*.
- Set the $\sigma_{n,ref}$ to -90 and $\sigma_{n,inc}$ to -14 and define (0 0 -4.75) as the reference point for the load by assigning the values to x_{ref}, y_{ref} and z_{ref} (Figure 6.8).

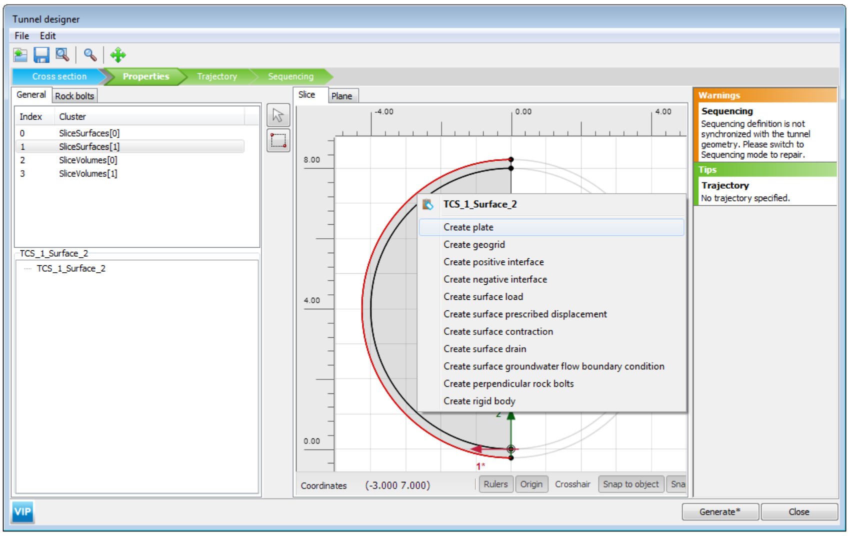

Figure 6.6 The *Properties* tabsheet of the *Tunnel designer* for the creation of a *Plate*

Jack forces

In order to move forward during the boring process, the TBM has to push itself against the existing tunnel lining. This is done by hydraulic jacks. The force applied by the jacks on the final tunnel lining has to be taken into account. This will be assigned to the tunnel lining in *Sequencing* tab.

Trajectory

The next step is to create the path of the boring process. The TBM already advanced 25 m into the soil and then proceed from 25 m to 41.5 m excavating slices of 1.5 m each:

- Click the *Trajectory* tab to proceed to the corresponding tabsheet.

 In the *Segments* tabsheet, click on the *Add segment* on the left toolbar.

- In the properties box set the length to 25.
- Add the next segment and set the length to 16.5.
- To create the slices, proceed to the *Slices* tabsheet.
- Click on the second created segment. In the properties box, select *Length* as the *Slicing method* and set the *Slice length* as 1.5 (Figure 6.9).

Sequencing

In order to simplify the definition of the phases in *Staged construction* mode, the sequencing of the tunnel is defined. The soil in front of the TBM will be excavated, a support pressure will be applied to the tunnel face, the TBM shield will be activated and the conicity of the shield will be modelled, at the back of the TBM the pressure due to the back fill grouting will be modelled as well as the force the hydraulic jacks driving the TBM exert on the already installed lining, and a new lining ring will be installed.

- Click the *Sequencing* tab to proceed to the corresponding tabsheet.

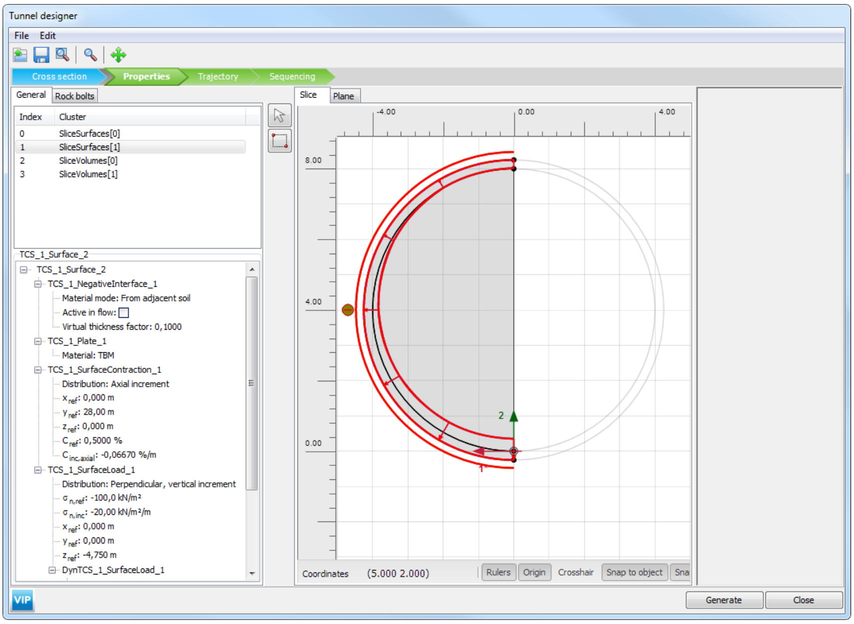

Figure 6.7 *Slice* tabsheet in the *Tunnel designer*

- In the *Sequencing* tabsheet, set the *Excavation method* as *TBM*.

Step_1_1, face excavation: Select the *Slice* tabsheet above the displayed tunnel cross section and select the volumes inside the tunnel. In the *Selection explorer*, deactivate the soil and set the *WaterConditions* to *Dry*. In the *Slice* tabsheet again, select the outer surface. In the *Selection explorer*, activate the negative interface, the plate and the surface contraction (Figure 6.10).

Set C_{ref} = 0% (since this is on the front of the excavation).

Select the *Plane Front* and select all the surfaces. Activate the surface load corresponding to the face pressure (Figure 6.11).

Step_1_2, TBM with conicity: The difference with the front of the TBM is only the face pressure. Select the *Plane Front* and select all the surfaces. In the *Selection explorer*, the surface load corresponding to the face pressure is deactivated by default. Set C_{ref} = 0.1% (on the front surface of the second slice).

From Step_1_3 to Step_1_5, TBM with conicity: click on the *Add Step* button thrice to add 3 new steps. These steps are necessary to define the remaining cone part of the TBM shield (Figure 6.12). Set the following values for the different steps: Step 1_3: C_{ref}= 0.2% (on the front surface of the third slice) Step 1_4: C_{ref}= 0.3% (on the front surface of the fourth slice) Step 1_5: C_{ref}= 0.4% (on the front surface of the fifth slice)

Step_1_6, tail of the shield: The last slice of the shield has a constant diameter. Hence, from the *Slice* tabsheet select the outer surface and select the surface contraction. In the *Selection explorer* select the *Uniform* option with C_{ref} = 0.5% (Figure 6.13).

Step_1_7, grouting and jack thrusting: Select the *Slice* tabsheet and select the

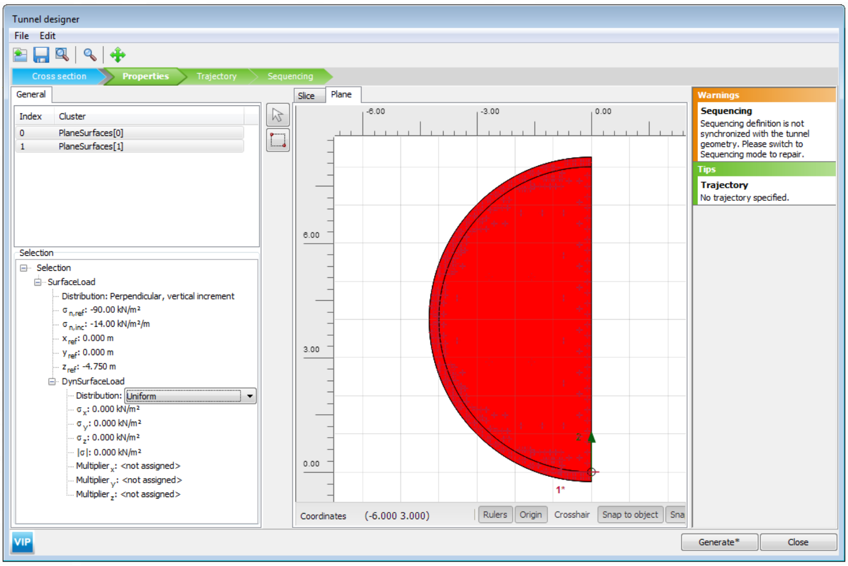

Figure 6.8 *Plane* tabsheet in the *Tunnel designer*

outer surface. Deactivate the negative interface, the plate and the surface contraction. In the *Selection explorer*, activate the surface load corresponding to the grout pressure (Figure 6.14).

Select the *Plane rear* tabsheet and select the outer surface to define the jack thrusting against the final lining. In the *Selection explorer*, activate the surface load and select the *Perpendicular* option for the distribution with $\sigma_{n,ref}$ = 635.4 (Figure 6.15).

Step_1_8, final lining: Select the *Slice* tabsheet and select the outer surface. In the *Selection explorer*, deactivate the grout pressure and activate the negative interface. In the *Slice* tabsheet again, select the outer volume. Activate it, click the material and select the *Concrete* option from the drop-down menu (Figure 6.16).

Select the *Plane rear* tabsheet and select the outer surface. In the *Selection explorer*, deactivate the surface load corresponding to the thrusting jacks (Figure 6.17).

- Click on *Generate* to include the defined tunnel in the model.
- Close the *Tunnel designer* window.

This concludes the model creation in *Structures* mode (Figure 6.18).

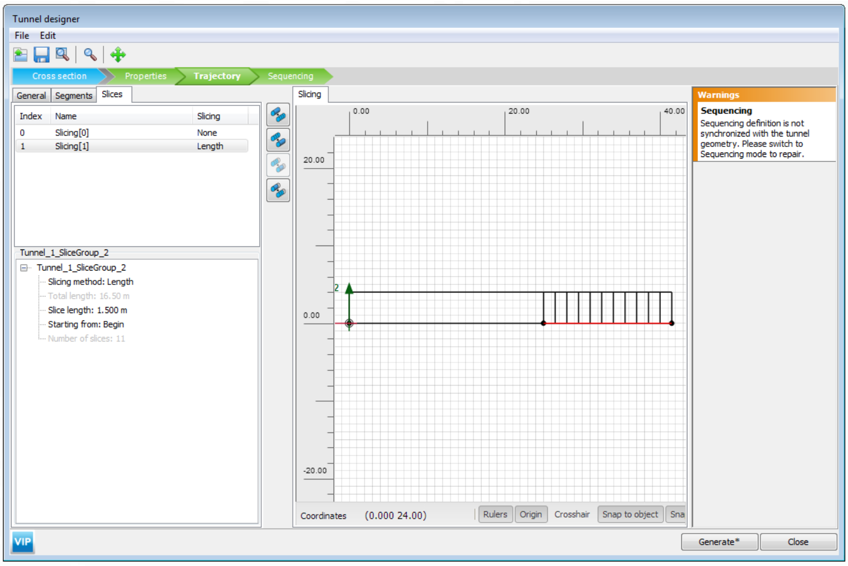

Figure 6.9 *Trajectory* tabsheet in the *Tunnel designer*

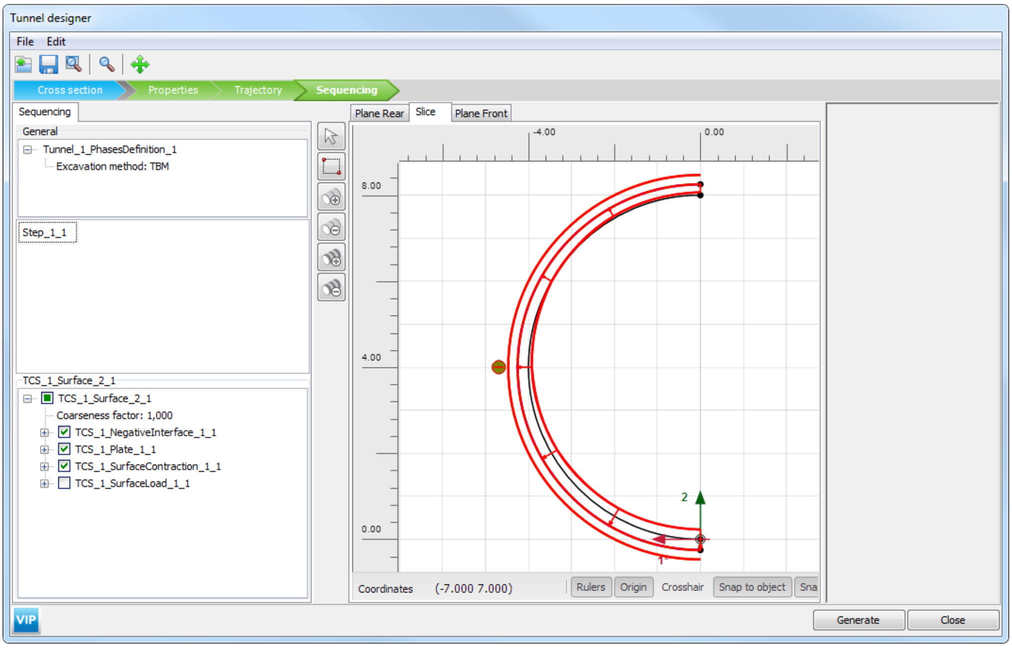

Figure 6.10 *Slice* tabsheet in the *Tunnel designer* for Step_1_1

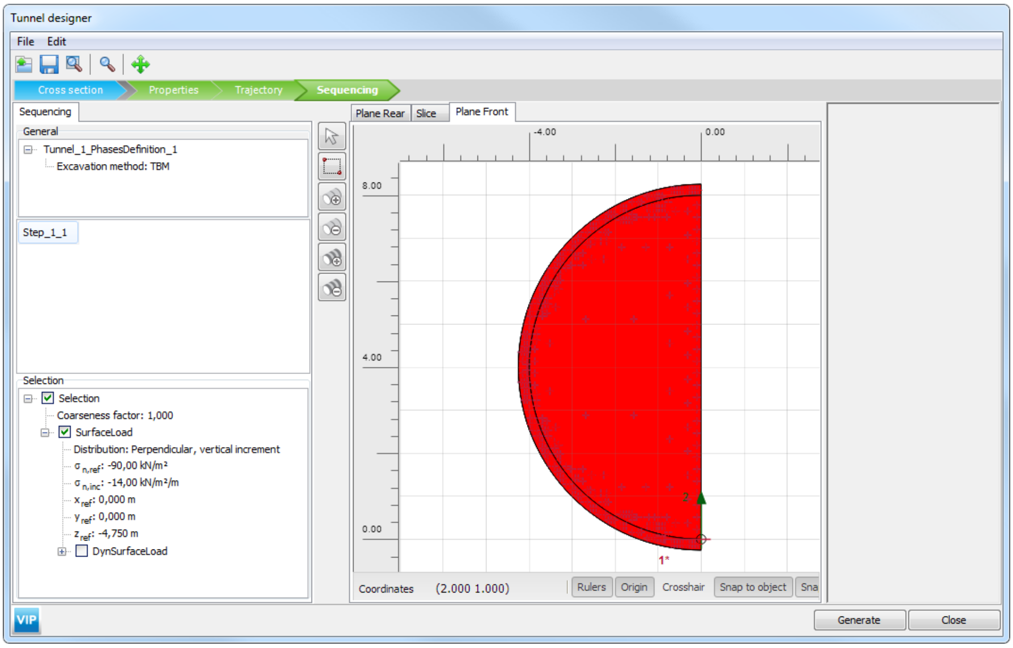

Figure 6.11 *Plane Front* tabsheet in the *Tunnel designer* for Step_1_1

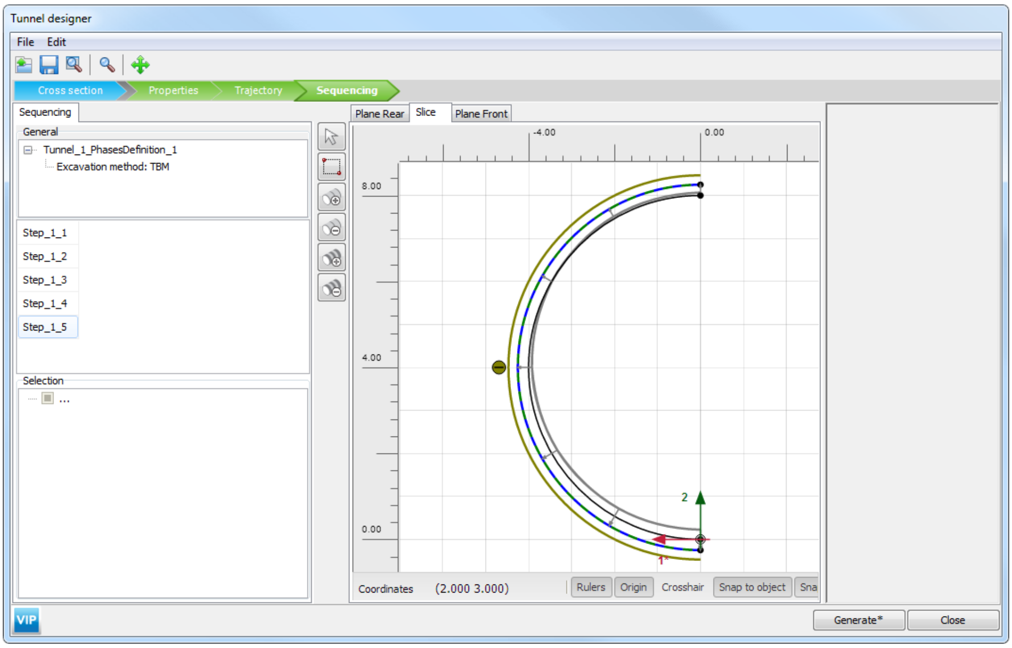

Figure 6.12 *Slice* tabsheet in the *Tunnel designer* from Step_1_3 to Step_1_5

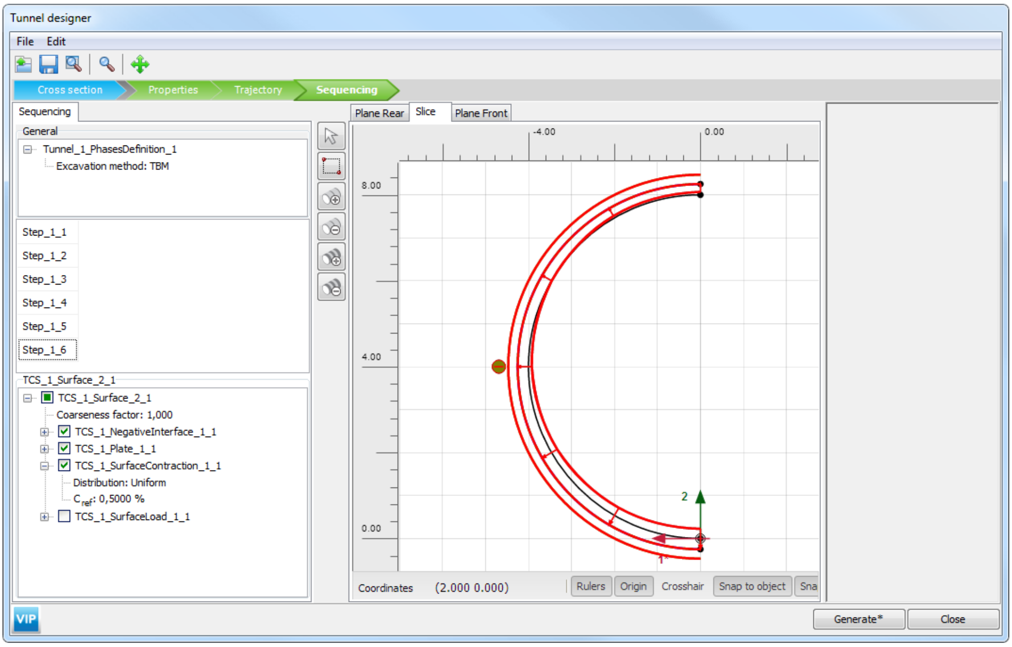

Figure 6.13 *Slice* tabsheet in the *Tunnel designer* for Step_1_6

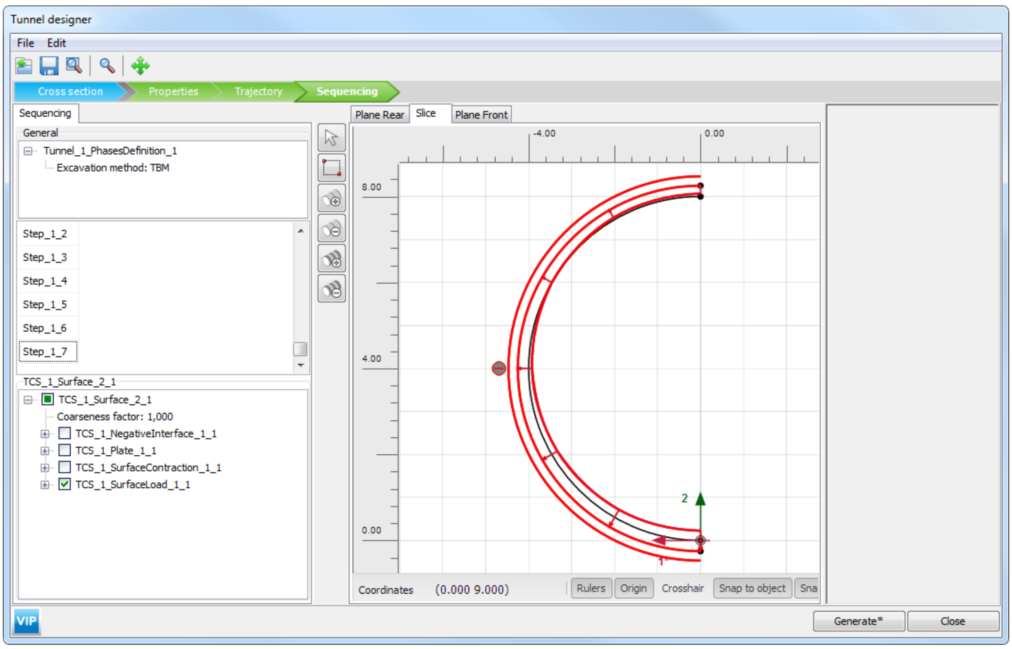

Figure 6.14 *Slice* tabsheet in the *Tunnel designer* Step_1_7

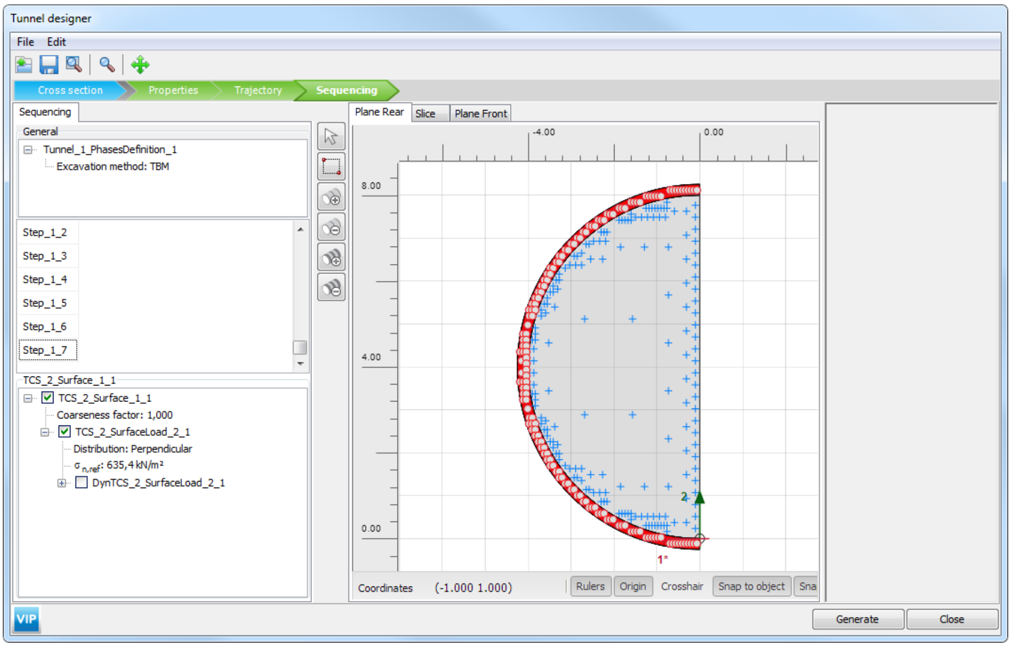

Figure 6.15 *Plane Rear* tabsheet in the *Tunnel designer* Step_1_7

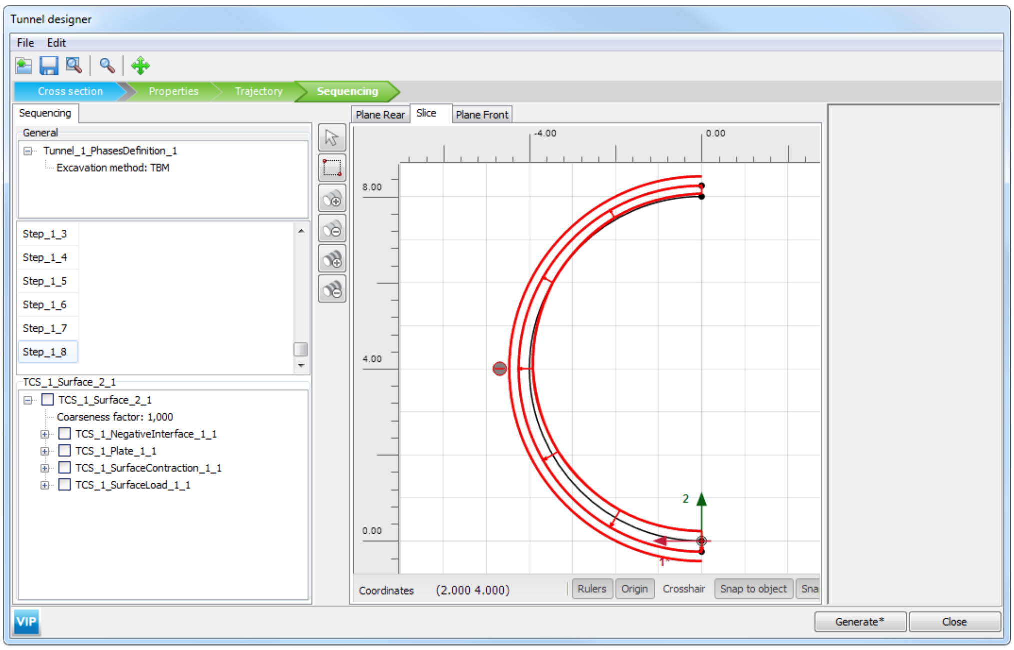

Figure 6.16 *Slice* tabsheet in the *Tunnel designer* Step_1_8

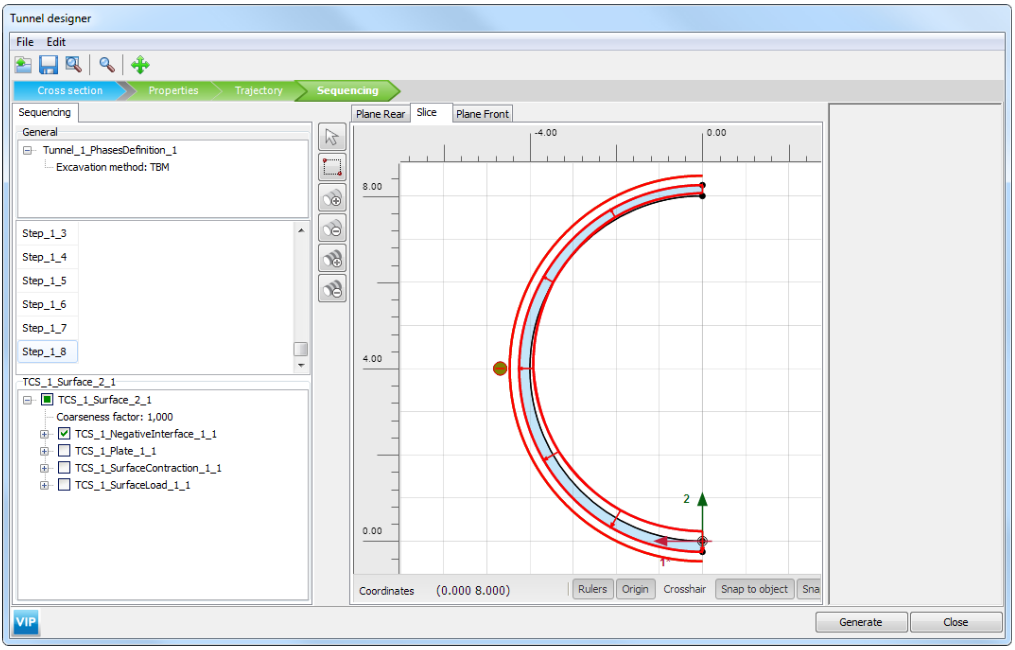

Figure 6.17 *Slice* tabsheet in the *Tunnel designer* Step_1_8

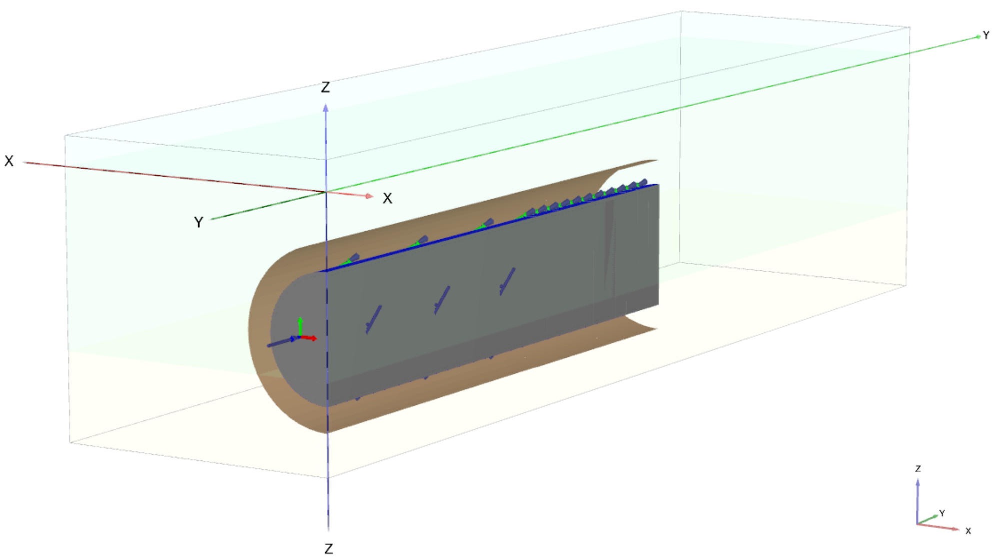

Figure 6.18 The created tunnels in *Structures* mode

6.2 MESH GENERATION

In the *Mesh* mode it is possible to specify global and local refinements and generate the mesh. The default local refinements are valid for this example.

- Click the *Generate mesh* button in order to generate the mesh. The *Mesh options* window appears.
- The default option (*Medium*) will be used to generate the mesh.
- Click the *View mesh* button to inspect the generated mesh (Figure 6.19).

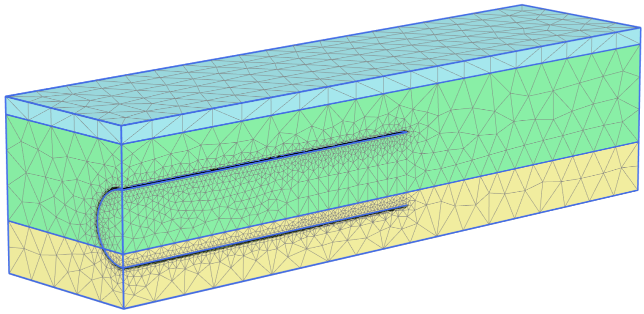

Figure 6.19 The generated mesh

After inspecting the mesh, the output window can be closed. Mesh generation has now been finished, and so creating all necessary input for defining the calculation phases has been finished.

6.3 PERFORMING CALCULATIONS

The excavation of the soil and the construction of the tunnel lining will be modelled in the *Staged construction* mode. Since water levels will remain constant the *Flow conditions* mode can be skipped. It should be noted that due to the mesh generation the tunnel effectively has been split into an upper part, located in the clay, and a lower part located in the stiff sand. As a result, both the lower and the upper part of the tunnel should be considered.

The first phase differs from the following phases, as in this phase the tunnel is activated for the first time. This phase will model a tunnel that has already advanced 25 m into the soil. Subsequent phases will model an advancement by 1.5 m each.

Initial phase

The initial phase consists of the generation of the initial stresses using the *K0 procedure*. The default settings for the initial phase are valid.

Phase 1 - Initial position of the TBM

In the first phase, it is assumed that the TBM has already advanced 25 m. The section next to the first 25 m (section 25 m - 26.5 m), will represent the area directly behind the TBM were grout is injected in the tail void. In the next 6 sections (26.5 m - 35.5 m) the

TBM will be modelled.

Add the first calculation phase.

- In the *Model explorer* expand *Tunnels* and then expand *Tunnel_1*. Scroll down the *Model explorer* until the option *Advancement step* and set it to 7 in order to simulate the advancement of the first 25 m.

The final lining will be activated in the following phase. In order to consider the conicity of the TBM in the first 25 m, the clusters representing final lining need to be deactivated, the plates representing the TBM are activated and 0.5% contraction is applied.

Select the right view to reorientate the model in order to obtain a clearer view of the inside of the tunnel.

- In the drawing area select the soil volumes corresponding to the lining in the first 25 m (Figure 6.20).

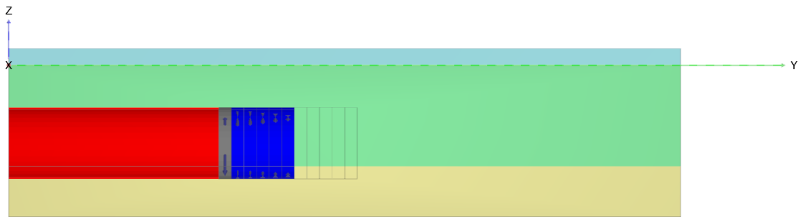

Figure 6.20 Selection of soil volumes (0 m - 25 m)

- In the *Selection explorer* deactivate the soil. The soil is switched off, but the wireframe representing the deactivated soil is still coloured red as the deactivated soil is still selected.

> **Hint:** An object that is deactivated will automatically be hidden as a volume or surface, but a wireframe representing the hidden object will remain. The visibility of the object not active in a calculation phase can be defined in the corresponding tabsheet of the *Visualization settings* window (Section 3.6.3 of the Reference Manual).

The interface is already activated. To activate the plate and the contraction in the first 25 m of the tunnel:

Select the *Select plates* option in the appearing menu. Select the surfaces between 0 m and 25 m in the model to which plates are assigned (Figure 6.21).

- In the *Selection explorer* activate plate and uniform contraction by checking the corresponding boxes.
- In the drawing area select the lateral surfaces of the outer volume, corresponding to the last slice of the TBM (grout and jack thrusting) at 25.0 m (Figure 6.22). In the *Selection explorer*, deactivate the surface load corresponding to the jack thrusting, because the TBM is only placed in this phase and it's not moving.

Click the *Preview* button to get a preview of everything that has been defined

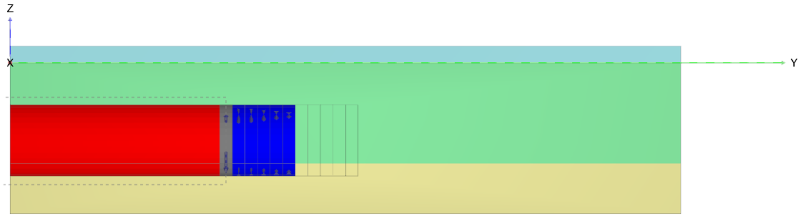

Figure 6.21 Selection of plate (0 m - 25 m)

(Figure 6.23). Make sure that both grout pressure and tunnel face pressure are applied and that both increase from top to bottom.

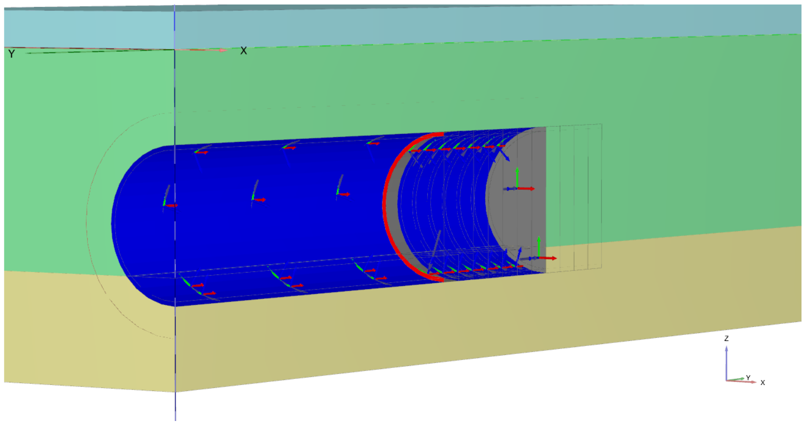

Figure 6.22 Selection of soil surfaces (25.0 m)

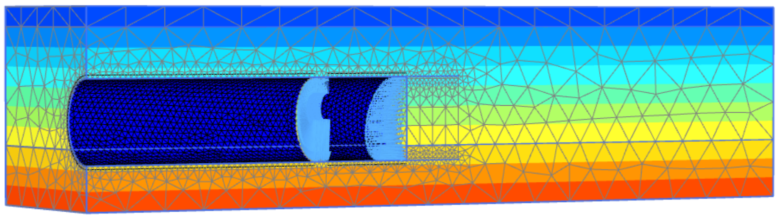

Figure 6.23 Preview of the Phase 1

Phase 2 - TBM advancement 1

In this phase, the advancement of the TBM by 1.5 m (from y = 35.5 to y = 37) will be modelled.

Add a new phase.

- In the *Model explorer* expand *Tunnels* and then expand *Tunnel_1*. Scroll down the *Model explorer* until the option *Advancement step* and set it to 8 in order to simulate the advancement of the first 26.5 m.

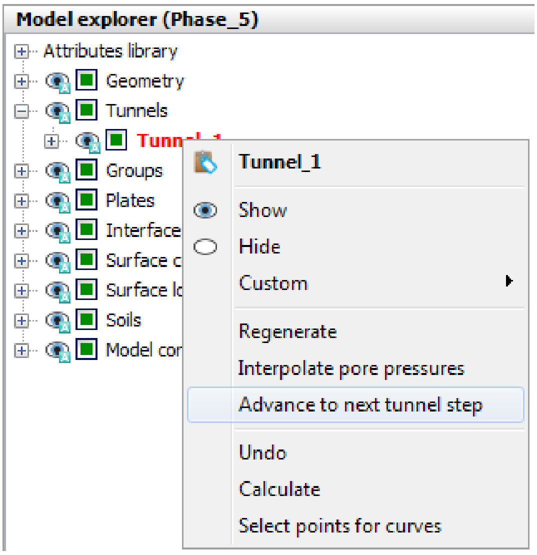

Figure 6.24 The *Advance to next tunnel step* option from *Model explorer*

Phase 3 - TBM advancement 2

In this phase, the TBM advances from y = 37 to y = 38.5.

- Add a new phase.
- In the *Model explorer* expand *Tunnels* and right-click *Tunnel_1*. Then click on *Advance to next tunnel step*.

Phase 4 - TBM advancement 3

In this phase, the TBM advances from y = 38.5 to y = 40.

- Add a new phase.
- In the *Model explorer* expand *Tunnels* and right-click *Tunnel_1*. Then click on *Advance to next tunnel step*.

Phase 5 - TBM advancement 4

In this phase, the final advancement of the TBM is modelled (from y = 40 to y = 41.5).

- Add a new phase.
- In the *Model explorer* expand *Tunnels* and right-click *Tunnel_1*. Then click on *Advance to next tunnel step*.

Press the *Calculate* button to start the calculation. Ignore the message "No nodes or stress points selected for curves" as any load-displacement curves are drawn in this example, and start the calculation.

6.4 VIEWING THE RESULTS

Once the calculation has been completed, the results can be evaluated in the Output program. In the Output program the displacement and stresses are shown in the full 3D model, but the computational results are also available in tabular form. To view the results for the current analysis, follow these steps:

- Select the last calculation phase (Phase 5) in the *Phases explorer*.
- Click the *View calculation results* button in the side toolbar to open the *Output* program. The Output program will by default show the 3D deformed mesh at the end of the selected calculation phase.
- From the *Deformations* menu, select *Total displacements* and then u_z in order to see the total vertical displacements in the model as a shaded plot (Figure 6.25).

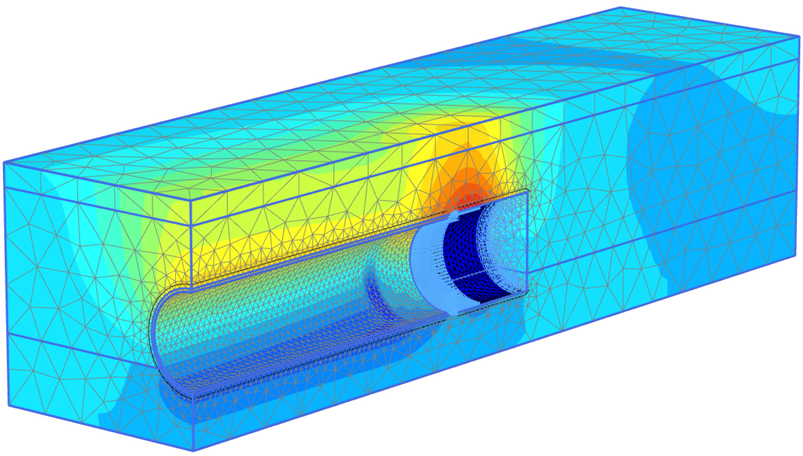

Figure 6.25 Total vertical displacements after the final phase $u_z \approx 3.1cm$

In order to see the settlements at ground level make a horizontal cross section by choosing the *Horizontal cross section* button. In the window that appears fill in a cross section height of 1.95 m. The window with the cross section opens (Figure 6.26). The maximum settlement at ground level is about 1.9 cm.

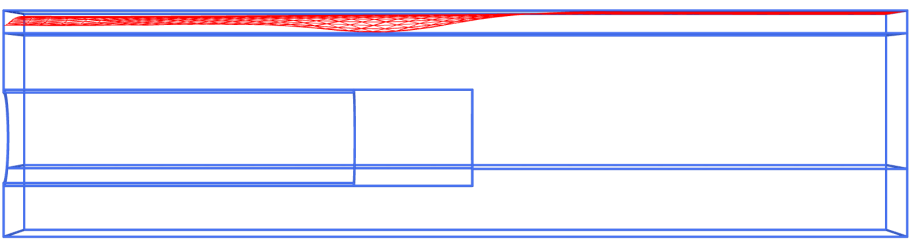

Figure 6.26 Settlement trough at ground level $|u| \approx 1.9cm$