5

Phased excavation of a shield tunnel [GSE]

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 1.5m 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 1.5m each phase.

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.

Geometry

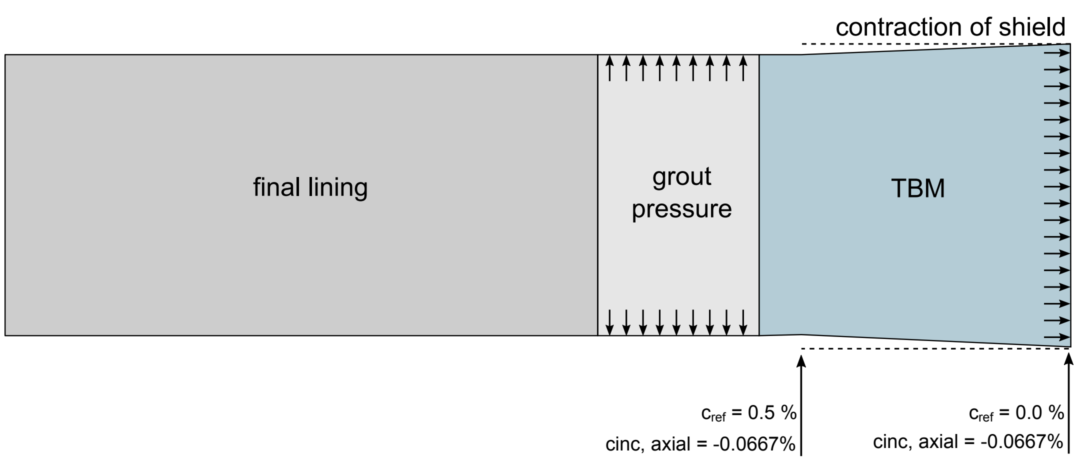

Figure 59: Construction stages of a shield tunnel model

5.1 Create a new project

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.

1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. Keep the default units and set the model dimensions to
   a. x_{min} = -20 and x_{max} = 0,
   b. y_{min} = 0 and y_{max} = 80.

5.2 Define the soil stratigraphy

The subsoil consists of three layers. The soft upper sand layer is 2m deep and extends from the ground surface to Mean Sea Level (MSL). Below the upper sand layer, there is a clay layer of 12m thickness and this layer is underlain by a stiff sand layer that extends to a large depth. Only 6m of the stiff sand layer is included in the model. Hence, the bottom of the model is 18m 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.

1. 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.
2. Define 3 layers: Upper sand with the top at 2m and the bottom at 0m, Clay with the bottom at -12m and Stiff sand with the bottom at -18m.

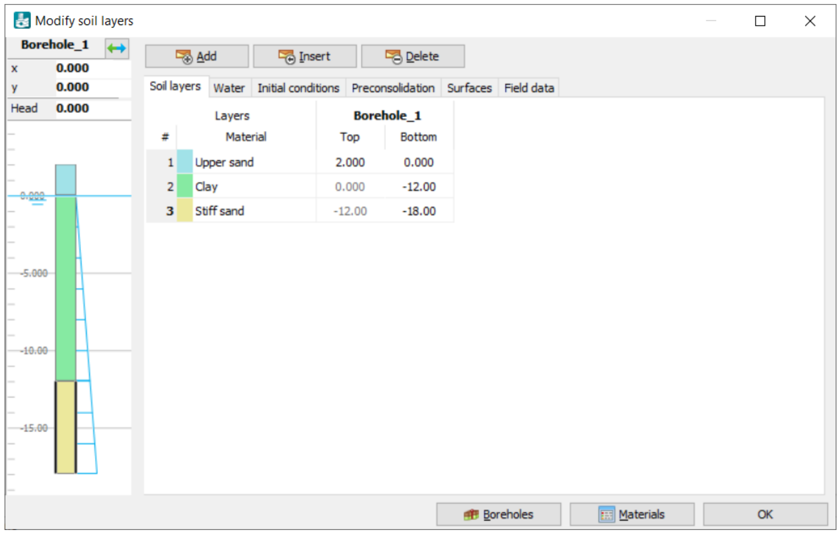

Figure 60: Soil layer distribution

5.3 Create and assign the material data sets

The material properties for the data sets are shown in Table 14 (on page 86).

Table 14: Material properties for the soil layers

Property	Name	Upper sand	Clay	Stiff sand	Concrete	Unit
General						
Soil model	Model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Linear elastic	-
Drainage type	Type	Drained	Drained	Drained	Non porous	-
Unsaturated unit weight	γ_{unsat}	17.0	16.0	17.0	27.0	kN/m^3
Saturated Unit weight	γ_{sat}	20.0	18.0	20.0	-	kN/m^3
Mechanical						
Young's modulus	E'_{ref}	$1.3 \cdot 10^4$	$1.0 \cdot 10^4$	$7.5 \cdot 10^4$	$3.1 \cdot 10^7$	kN/m^2
Poisson's ratio	$\nu(nu)$	0.3	0.35	0.3	0.1	-
Cohesion	c'_{ref}	1.0	5.0	1.0	-	kN/m^2
Friction angle	$\varphi'(phi)$	31	25	31	-	°

Mechanical						
Dilatancy angle	*ψ(psi)*	0	0	0	-	°
Interfaces						
Interface strength	-	Rigid	Rigid	Rigid	Rigid	-
Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-

1. 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 the Table 14 (on page 86).
2. Assign the material data sets to the corresponding soil layers (Figure 60 (on page 86))and close the **Modify soil layers** window. The concrete data set will be assigned later.

5.4 Definition of structural elements

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

Note: 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** , Reference Manual - *Chapter 7 - Definition of connections*.)
- The tunnel shape is composed of arcs with different radii (e.g. NATM tunnels).

The material properties for the linning tunnel are presented in Table 15 (on page 87).

Table 15: Material properties of the plate representing the TBM

Property	Name	TBM	Unit
General			
Material type	-	Elastic	-
Unit weight	γ	247	kN / m^3

Mechanical			
Isotropic	-	Yes	-
Young's modulus	E_1	$200 \cdot 10^6$	kN / m^2
Poisson's ratio	ν_{12}	0	-
Thickness	d	0.17	m
Shear modulus	G_{12}	$100 \cdot 10^6$	kN / m^2

Note: 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.

5.4.1 Create tunnel

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

1.  Click the **Start designer** button in the side toolbar.
2.  Click the **Create tunnel** button from the list.
3. Click anywhere on the drawing area to define the insertion point. The **Tunnel designer** window pops up.
4. In the **Selection explorer** set the insertion point of the tunnel to (0 0 -13.25) as shown in Figure 61 (on page 88).

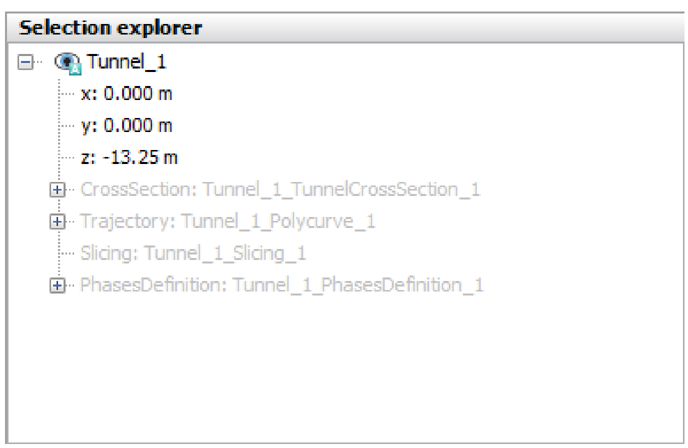

Figure 61: Insertion point of the tunnel

5. In the **General** tabsheet for the **Shape type** select the *Circular* option from the drop-down menu.
6. 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 62 (on page 89).

7. 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.
8. In the **Segment** box set *Radius* to 4 m. This is the inner radius of the tunnel.
9. Proceed to the **Subsections** tabsheet.
10. Click the **Generate thick lining** button  in the side toolbar. The **Generate thick lining** window pops up.
11. 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 63 (on page 90)
12. 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.
13. In the cross section inside the **Slice** tab, right-click on the outer surface and from the appearing menu select **Create** > **Create plate** (see Figure 64 (on page 90)).
14. Click on the **Material** in the lower part of the explorer. Create a new material dataset for TBM and specify the material parameters for the TBM according to Table 15 (on page 87).

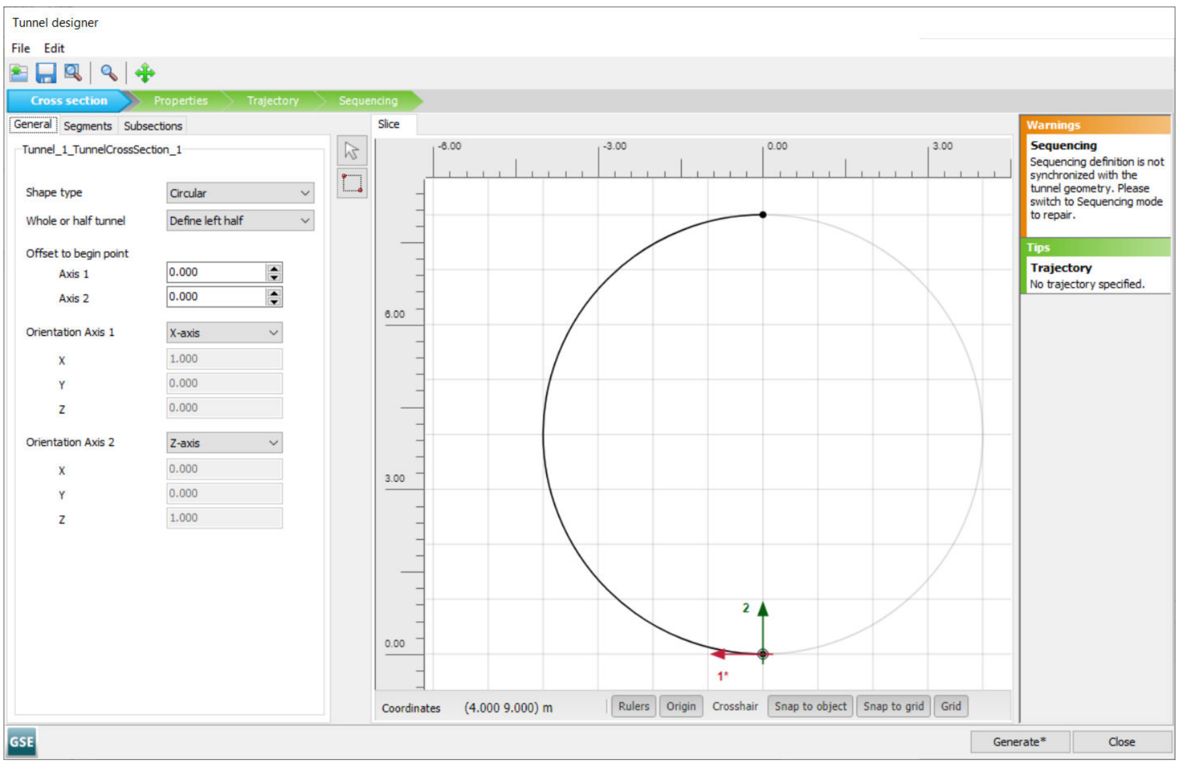

Figure 62: General tabsheet of the Tunnel designer

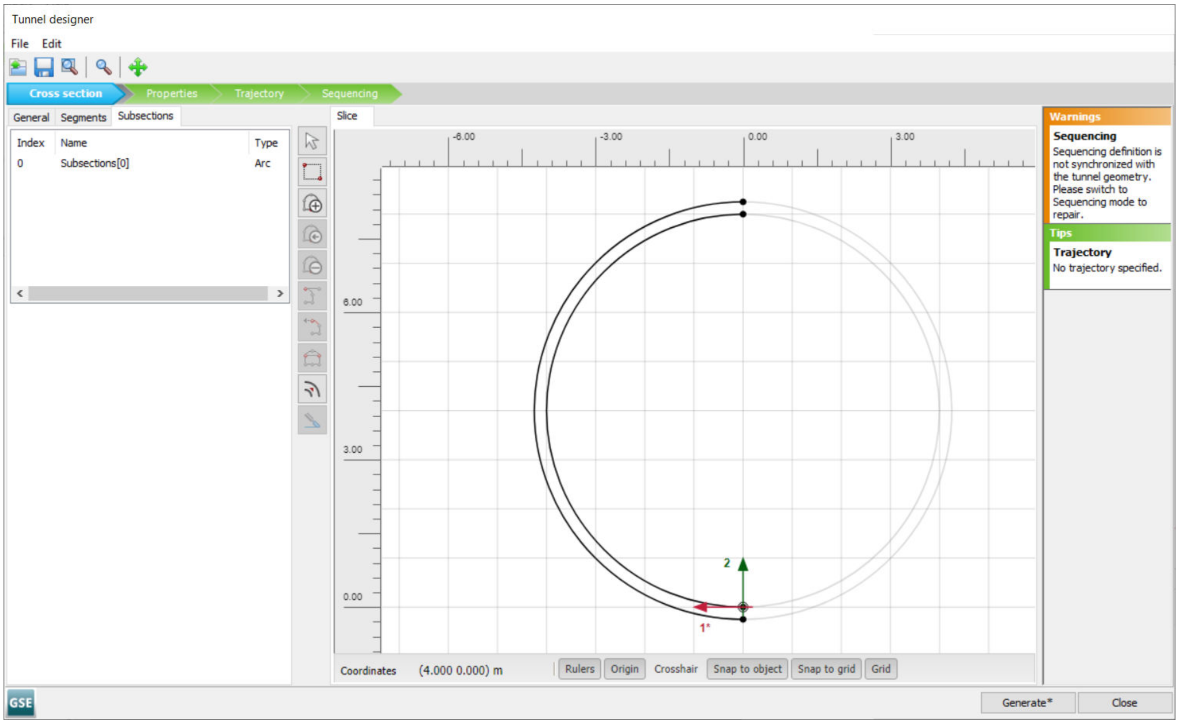

Figure 63: The Cross section tabsheet of the Tunnel designer

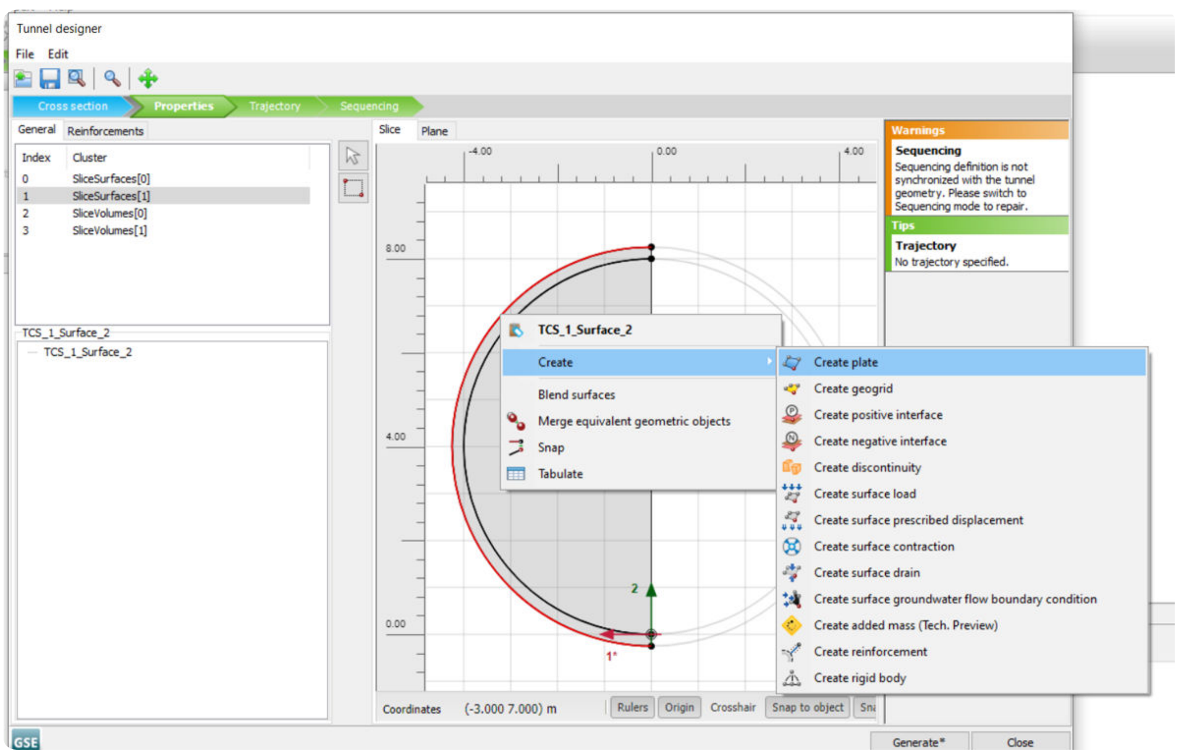

Figure 64: The Properties tabsheet of the Tunnel designer for the creation of a Plate

5.4.2 Surface contraction

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 0.5% smaller than the front of the TBM. The reduction of the diameter is realised over the first 7.5m length of the TBM while the last 1.5m 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 to the Reference Manual .

1. Right-click the same outer surface and select **Create negative interface** from the appearing menu to create a negative surface around the entire tunnel.
2. Next step is to create **Surface contraction** for the tunnel. Right-click the outer surface and select **Create** > **Create surface contraction**.
3. In the **Selection explorer** of the tunnel select **Surface contraction** > **Distribution** > **Axial increment** 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*.

Note:

- 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.

5.4.3 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 = -4.75m) and should increase with -20 kN/m^2/m depth. To define the grout pressure:

1. Right-click the outer surface and select **Create** > **Create surface load** from the appearing menu to create a surface load around the entire tunnel.
2. In the **Selection explorer** of the tunnel, from the drop-down menu for **Surface load** > **Distribution** select **Perpendicular, vertical increment** .
3. 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 65 (on page 92))

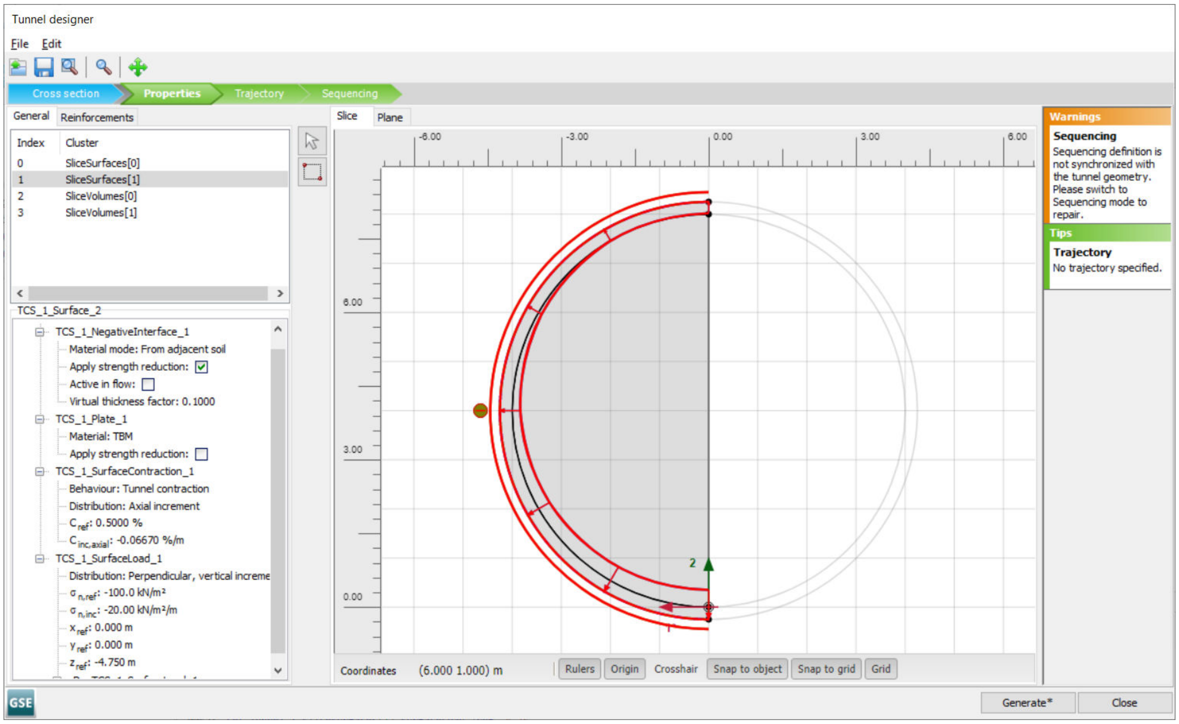

Figure 65: Slice tabsheet in the Tunnel designer

5.4.4 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.

1. Select the **Plane** tab located above the displayed tunnel cross section.
2. Multi-click both the surfaces, right-click and select **Create** > **Create surface load** from the appearing menu to create a surface load around the entire tunnel.
3. In the Selection explorer box go to **Selection** > **Surface Load** and from the **Distribution** option select **Perpendicular, vertical increment** from the drop-down menu.
4. 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 66 (on page 93)).

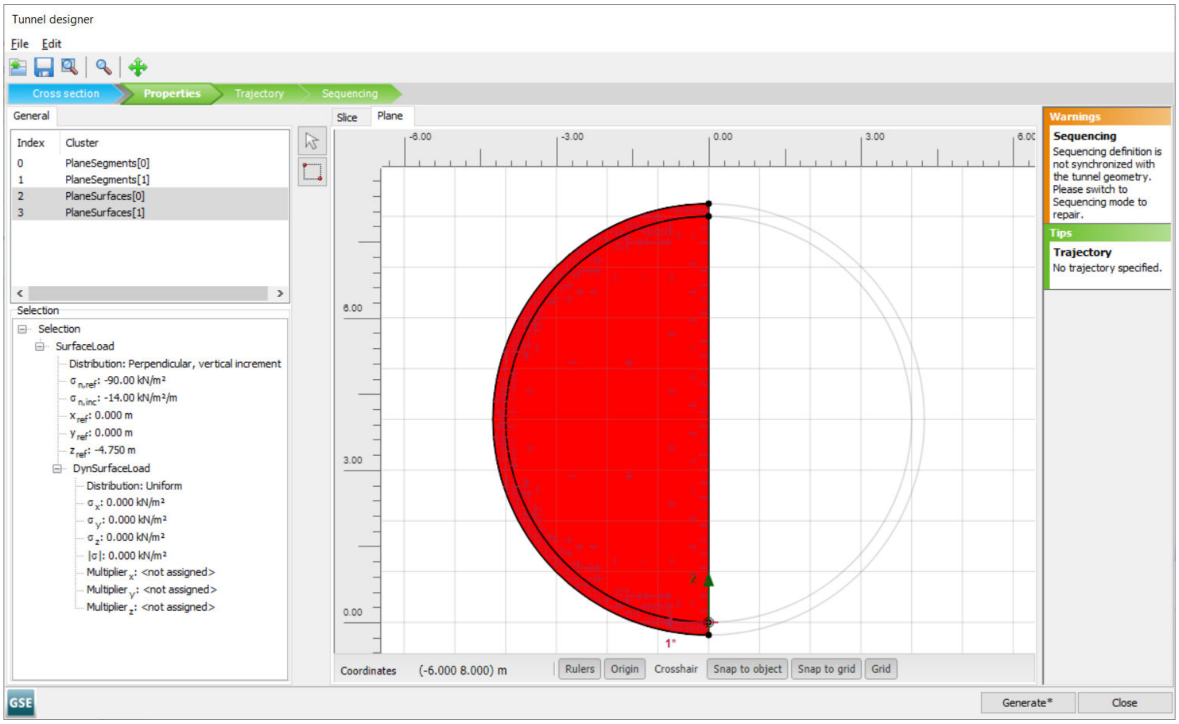

*Figure 66: Plane tabsheet in the **Tunnel designer***

5.4.5 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.

5.4.6 Trajectory

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

1. Click the **Trajectory** tab to proceed to the corresponding tabsheet.
2. In the **Segments** tab, click on the **Add segment** on the left toolbar.
3. In the properties box set the length to 25.
4. Add the next segment and set the length to 16.5.
5. To create the slices, proceed to the **Slices** tab.
6. Click on the second created segment. In the properties box, as the **Slicing method** select *Length* and set the **Slice length** as 1.5m (Figure 67 (on page 94)).

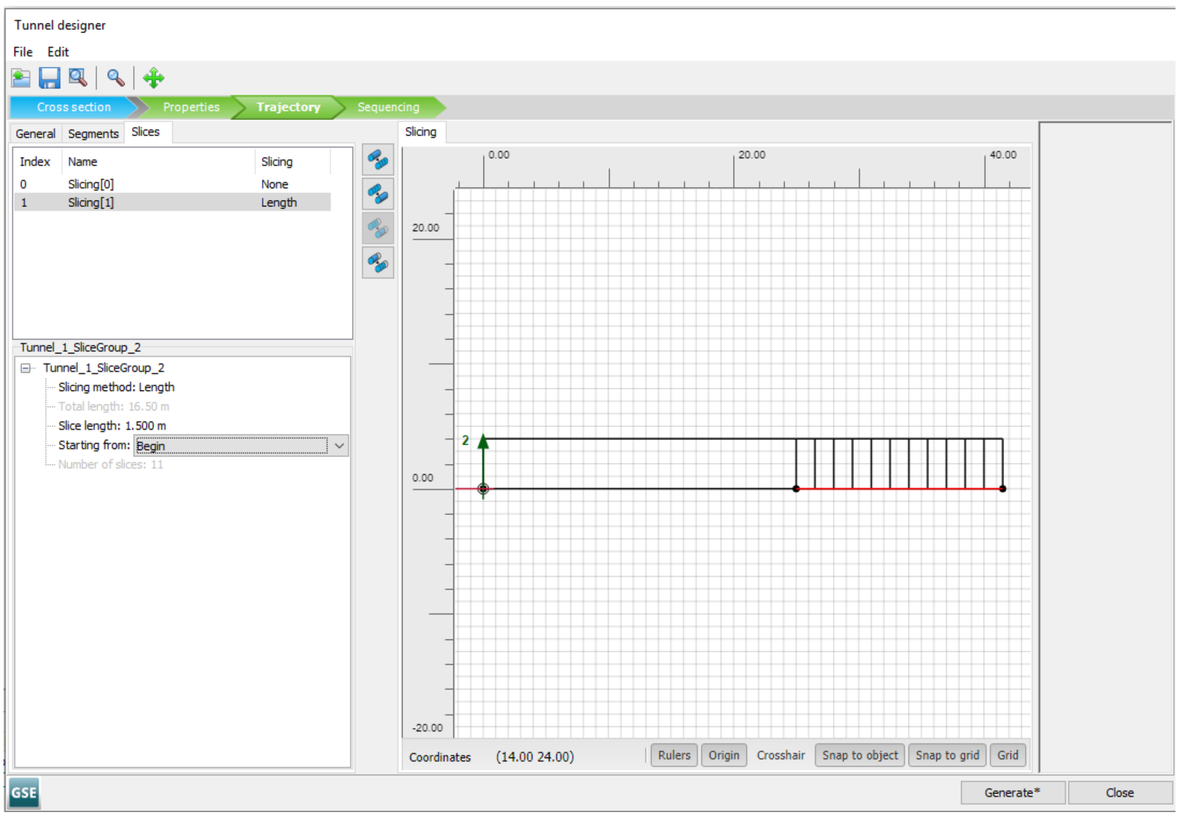

*Figure 67: Trajectory tabsheet in the **Tunnel designer***

5.4.7 Sequencing

In order to simplify the definition of the phases in the **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.Then at the back of the TBM , the pressure due to the tail void will be modelled.Then it is also required to model a driving force applied by hydraulic jacks on the TBM . And finally a new lining ring will be installed.

1. Click the **Sequencing** tab to proceed to the corresponding tabsheet.
2. In the **Sequencing** tabsheet, the **Excavation method** is set as *TBM*.

   a. ***<Step_1_1, face excavation>***

   - Select the **Slice** tab (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 as well, select the outer surface. In the **Selection explorer**, activate the negative interface, the plate and the surface contraction (Figure 68 (on page 95)).
   - Set C_{ref} = 0% for the surface contraction (since this is on the front of the excavation).
   - Go to the **Plane Front** tab and select all the surfaces. Activate the surface load corresponding to the face pressure (Figure 69 (on page 96)).

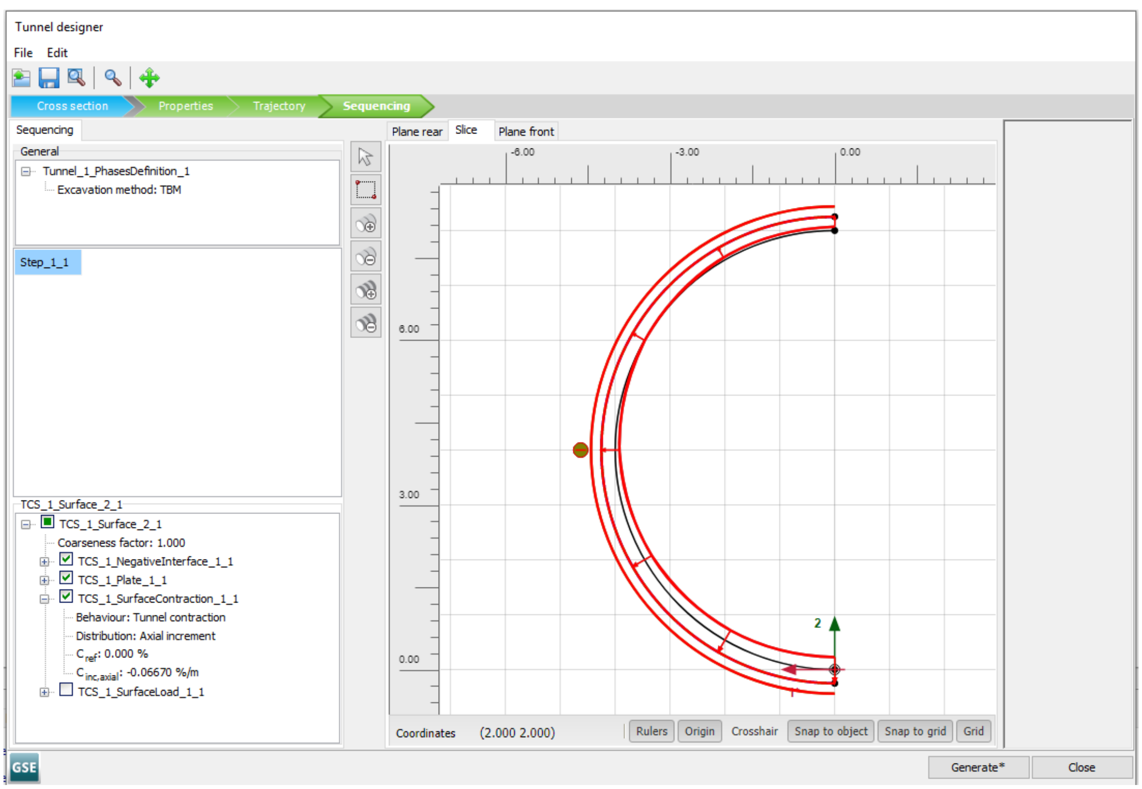

*Figure 68: Slice tabsheet in the **Tunnel designer** for Step_1_1*

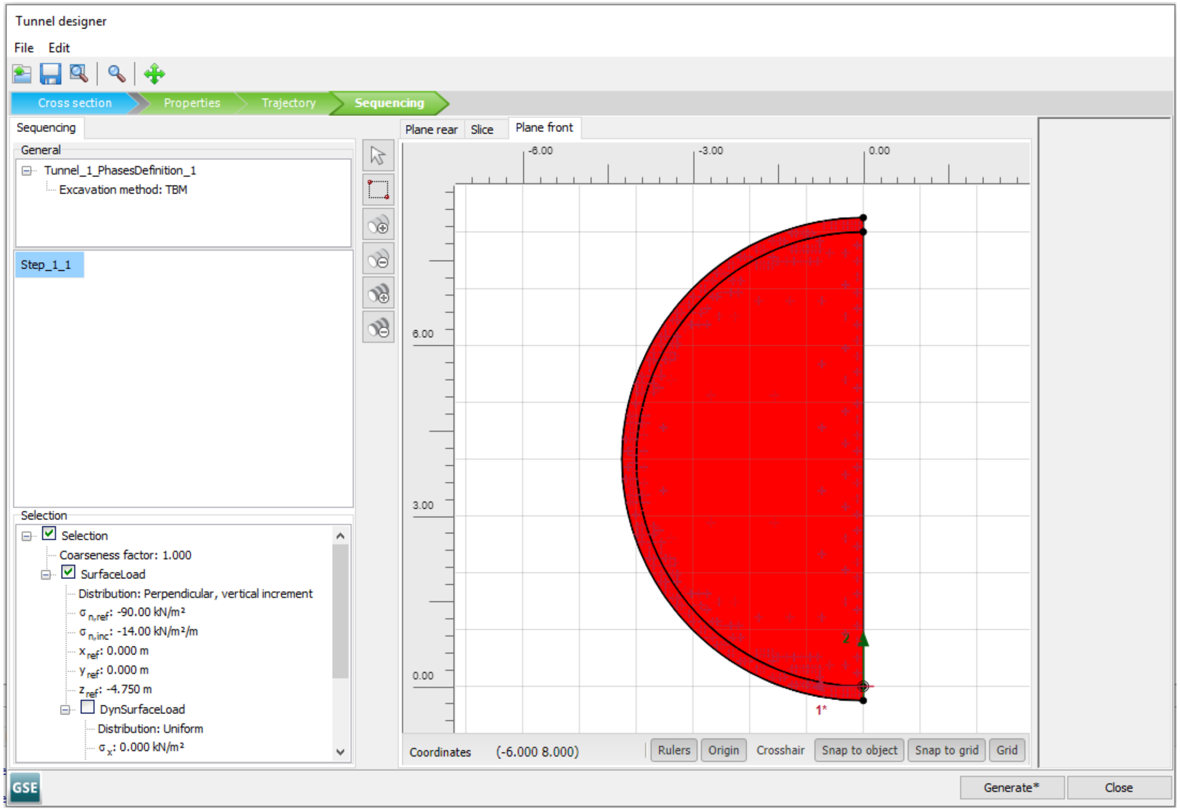

Figure 69: Plane Front tab in the **Tunnel designer** for Step_1_1

b. ***<Step_1_2, TBM with conicity>***

- Click on the **Add Step** to add a new step. The difference with the front of the TBM is only the face pressure.
- Go to the **Plane Front** tab and select all the surfaces. In the **Selection explorer**, the surface load corresponding to the face pressure is deactivated by default.
- Go to the **Slice** tab, select the outer surface and set C_{ref} = 0.1%, see Figure 70 (on page 97).

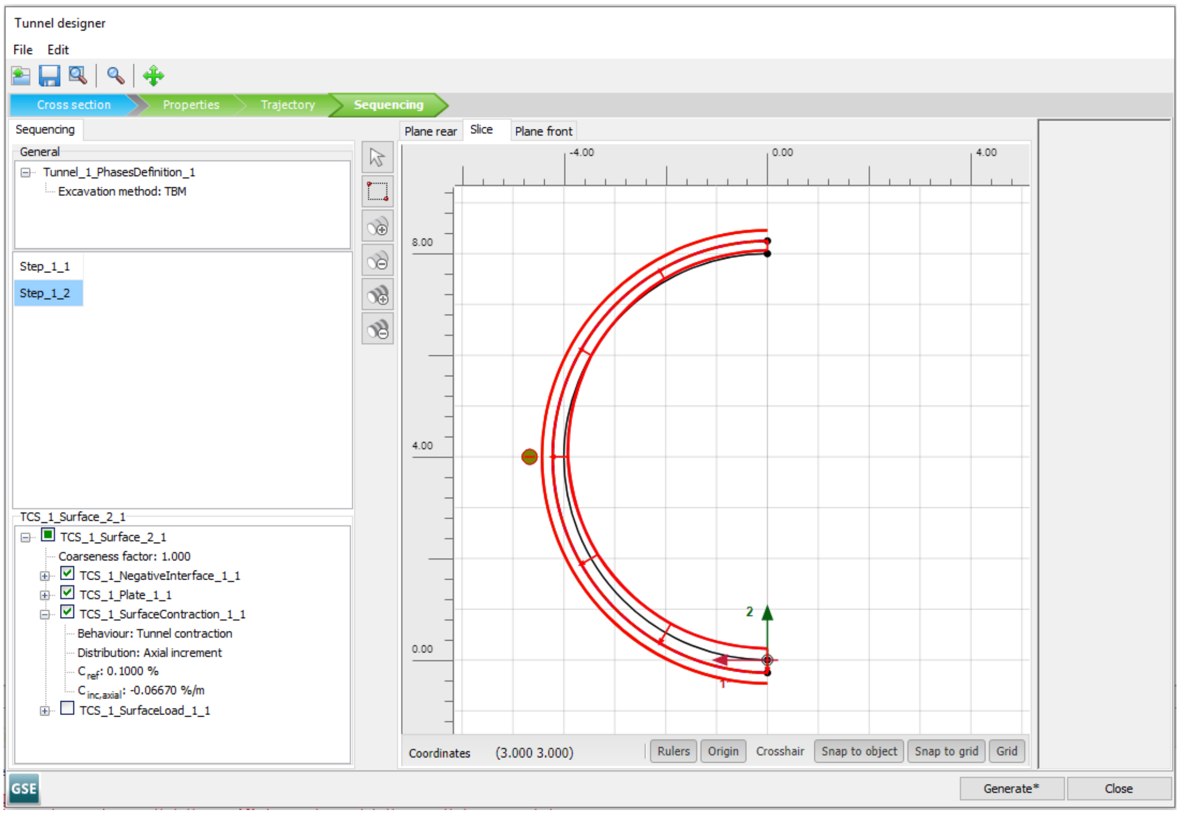

Figure 70: Slice tabsheet in the ***Tunnel designer*** *for Step_1_2*

c. *<Step_1_3 to Step_1_5, TBM with conicity>*

- Click on the **Add Step** button three times to add three new steps. These steps are necessary to define the remaining cone part of the TBM shield (Figure 71 (on page 98)).
- For each step go to the **Slice** tab, select the outer surface and set the following values for the surface contraction in the **Selection explorer**:

  Step_1_3: C_{ref}= 0.2%

  Step_1_4: C_{ref}= 0.3%

  Step_1_5: C_{ref}= 0.4%, see Figure 71 (on page 98)

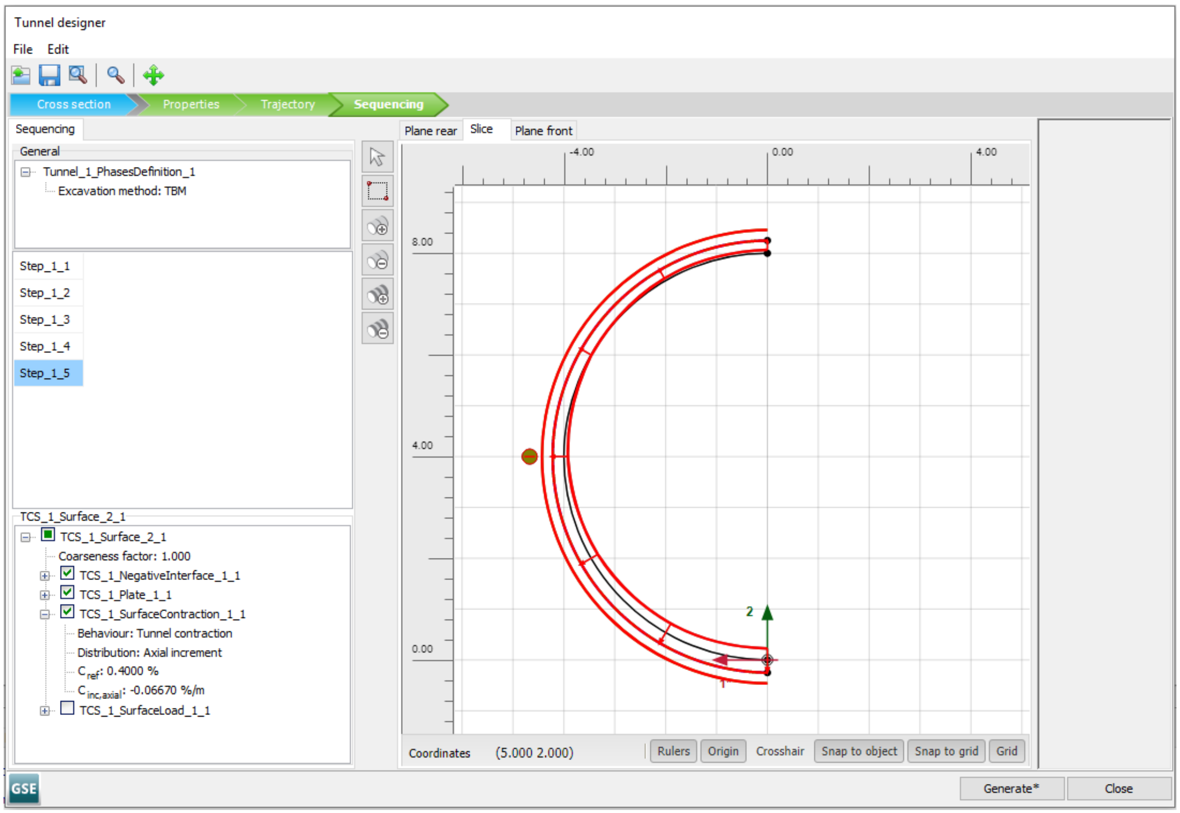

*Figure 71: Slice tab in the **Tunnel designer** from Step_1_3 to Step_1_5*

d. ***<Step_1_6, tail of the shield>***

- The last slice of the shield has a constant diameter. From the **Slice** tab select the outer surface and select the surface contraction.
- In the **Selection explorer** select from the **Surface contraction** > **Distribution** the *Uniform* option with C_{ref} = *0.5%* (Figure 72 (on page 99)).

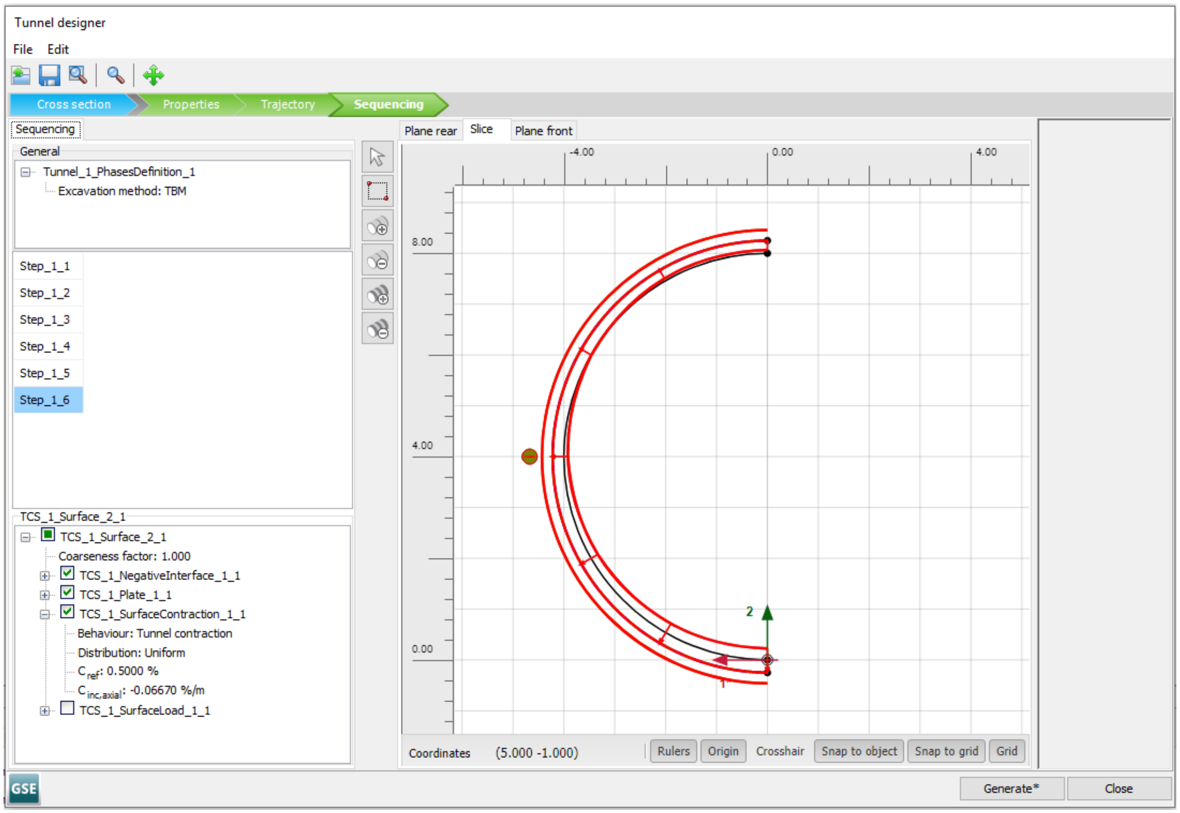

Figure 72: Slice tabsheet in the ***Tunnel designer*** *for Step_1_6*

e. ***<Step_1_7, grouting and jack thrusting>***

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.

- Select the **Slice** tabsand select the 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 73 (on page 100)).
- Select the **Plane rear** tab 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 kN/m^2 (Figure 74 (on page 101)).

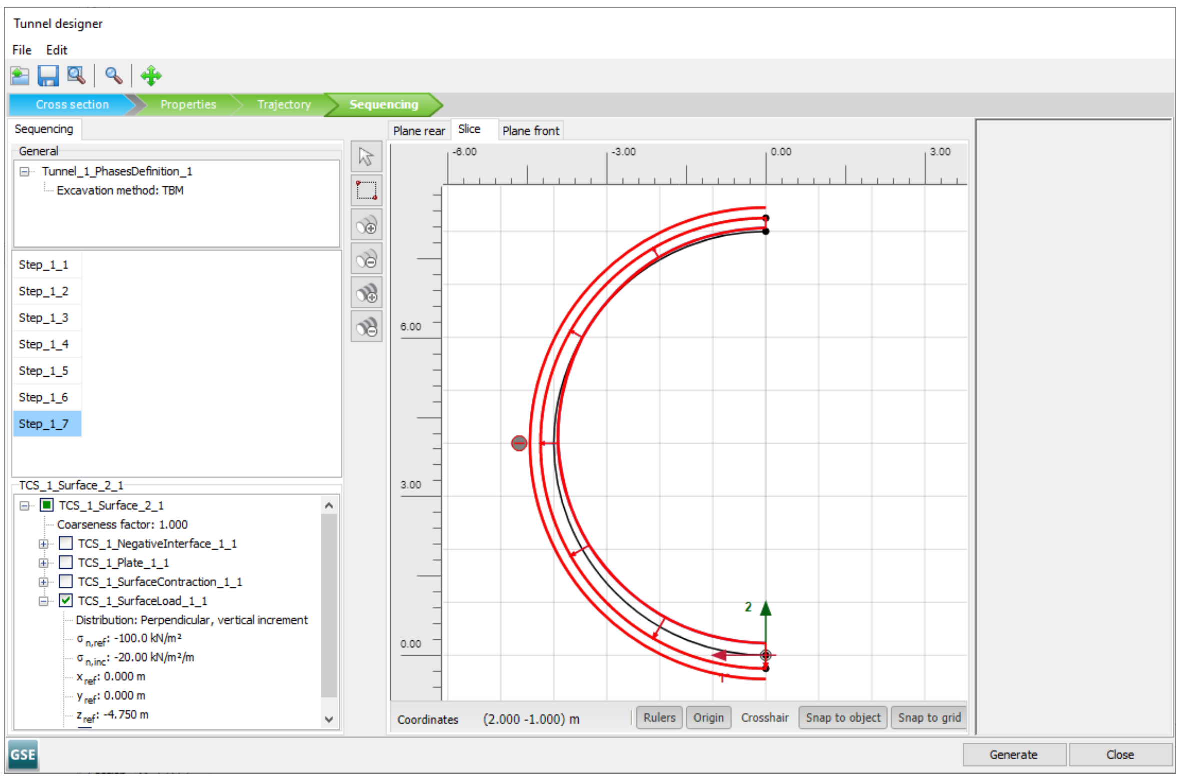

*Figure 73: Slice tabsheet in the **Tunnel designer** Step_1_7*

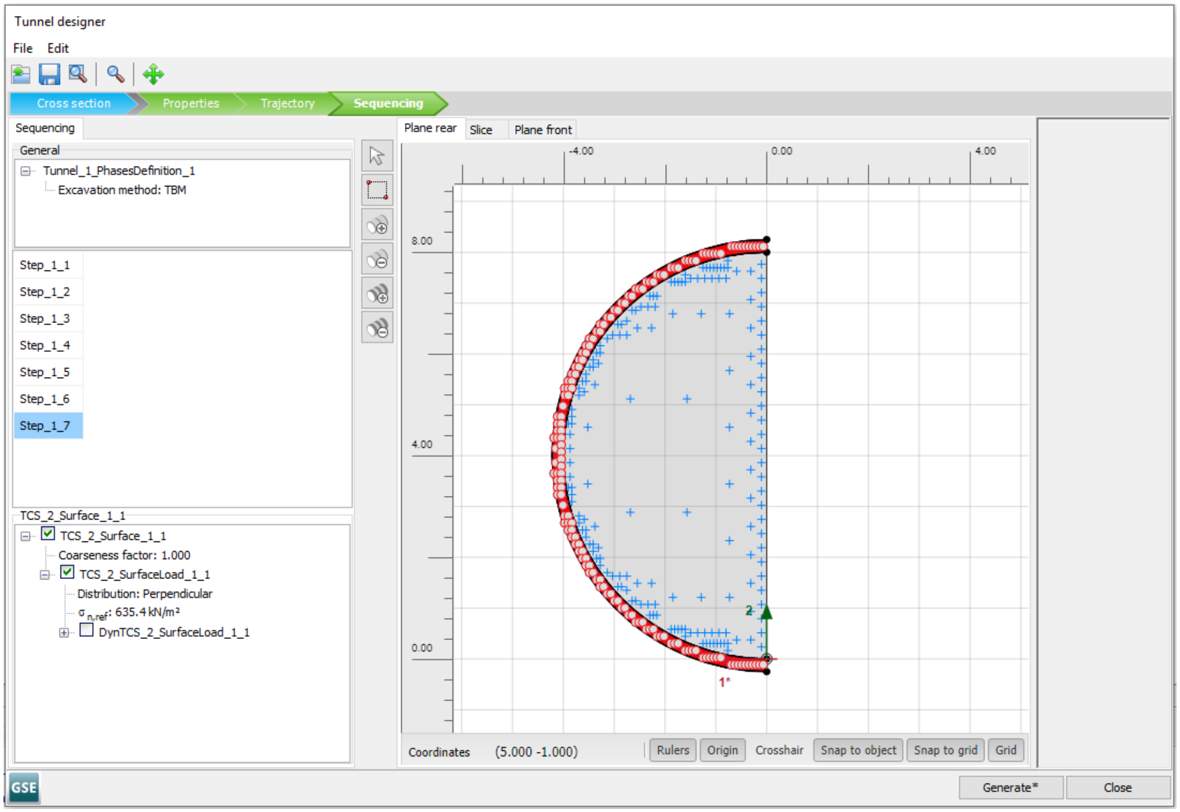

*Figure 74: Plane Rear tabsheet in the **Tunnel designer** Step_1_7*

f. ***<Step_1_8, final lining>***

- Select the **Slice** tabsheet and select the outer surface.
- In the **Selection explorer**, deactivate the surface load corresponding to 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 75 (on page 102)).
- Select the **Plane rear** tabsheet and select the outer surface.
- In the **Selection explorer**, deactivate the surface load corresponding to the thrusting jacks (Figure 76 (on page 103)).

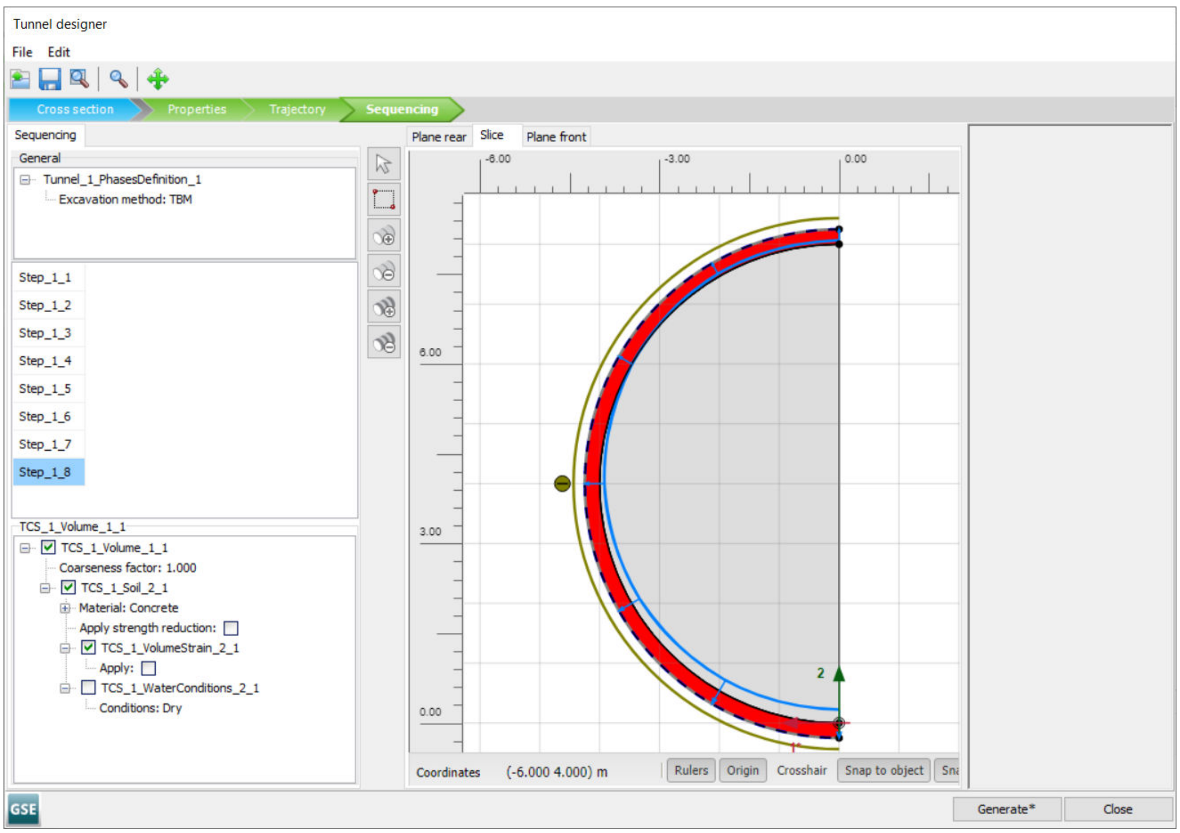

*Figure 75: Slice tabsheet in the **Tunnel designer** Step_1_8*

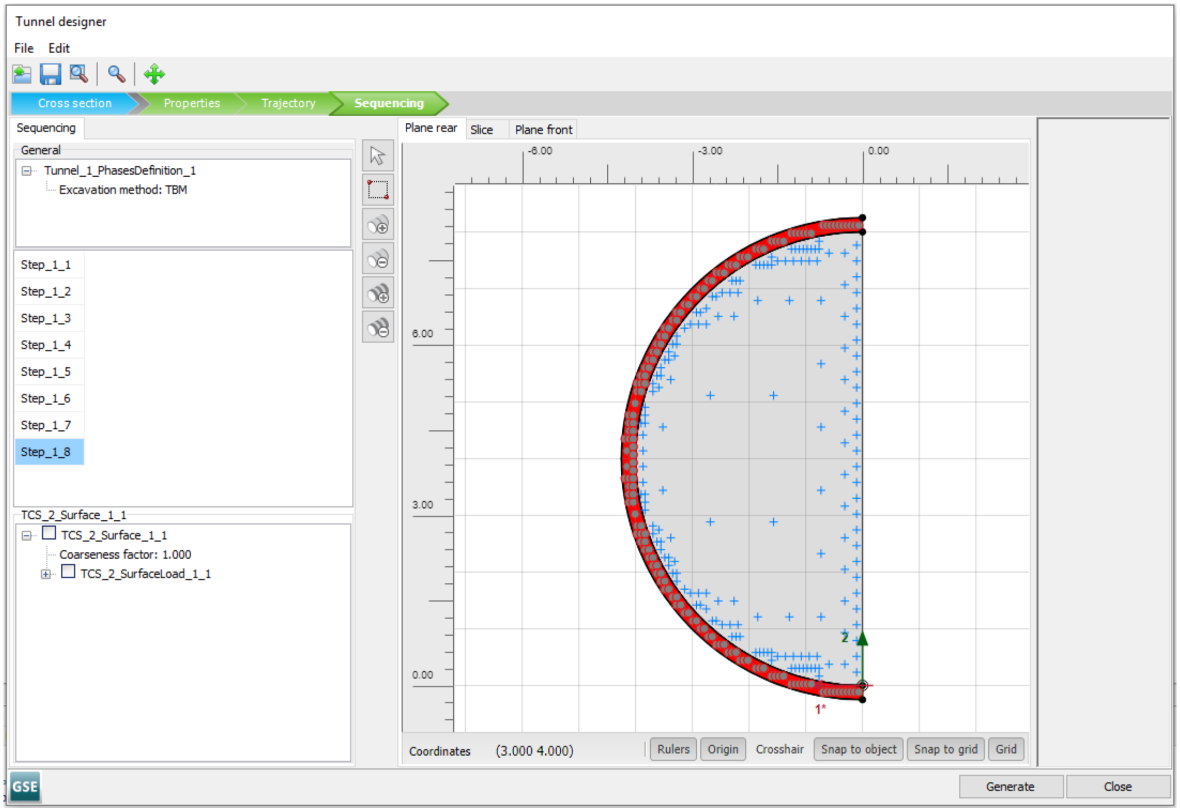

*Figure 76: Plane rear tabsheet in the **Tunnel designer** Step_1_8*

Note: For the steps *Step1_1* to *Step1_5* keep in mind that the contraction increment $C_{inc,axial}$ has to be -0.0667%.

3. Click on **Generate** to include the defined tunnel in the model.
4. To create the slices, proceed to the **Slices** tabsheet.
5. Close the **Tunnel designer** window.
   Then the model is created in the **Structures mode**. Click on the **Options** menu and then select **Show local axes on surfaces with structure** this is displayed on Figure 77 (on page 104).

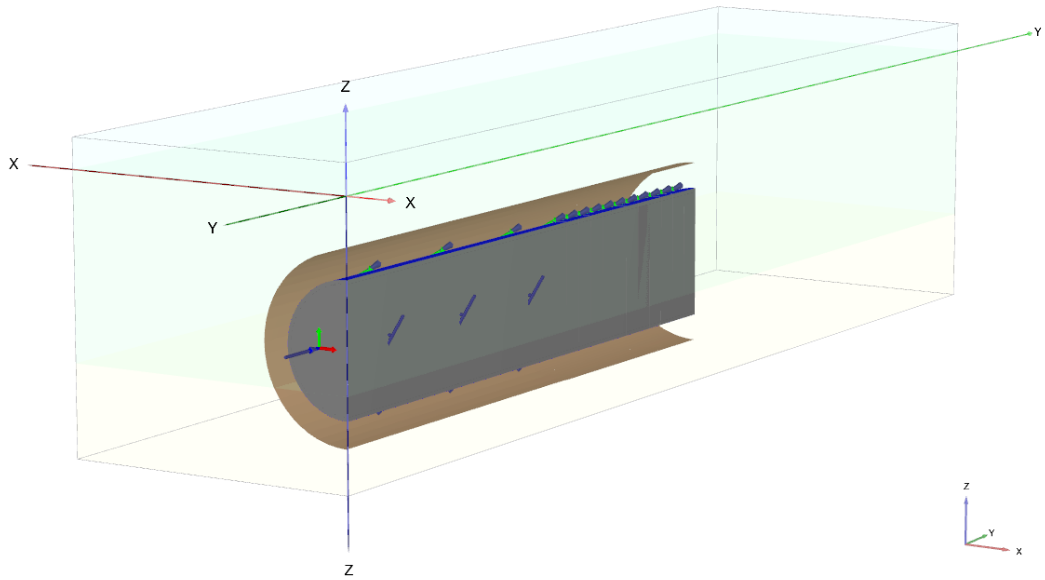

Figure 77: The created tunnels in Structures mode

5.5 Generate the mesh

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.

1. 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.
2. Click the **View mesh** button to inspect the generated mesh(Figure 78 (on page 105)).

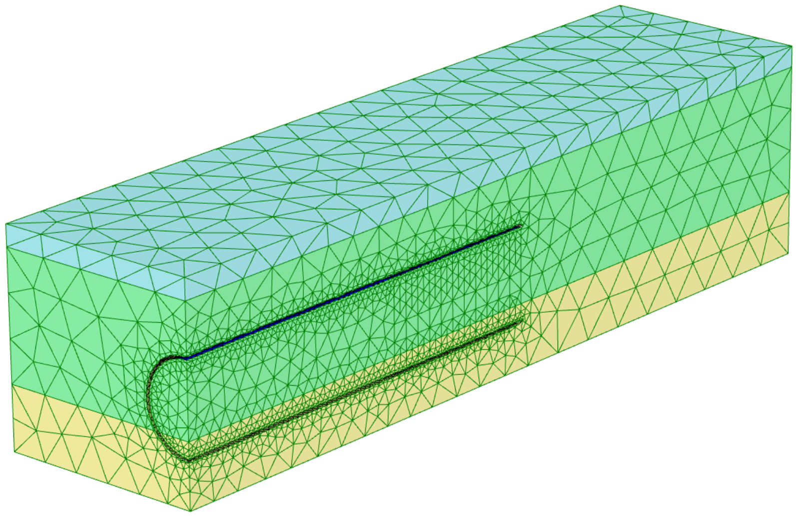

Figure 78: The generated mesh

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.

5.6 Define and perform the calculation

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.

5.6.1 Initial phase

The initial phase consists of the generation of the initial stresses using the K_0 *procedure*. The default settings for the initial phase are valid.

5.6.2 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 where grout is injected in the tail void. In the next 6 sections (26.5 m - 35.5 m) the TBM will be modelled.

1. Add the first calculation phase
2. 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.
3. Select the right view to reorientate the model in order to obtain a clearer view of the inside of the tunnel.
4. In the drawing area select the soil volumes corresponding to the lining in the first 25 m (Figure 79 (on page 106))

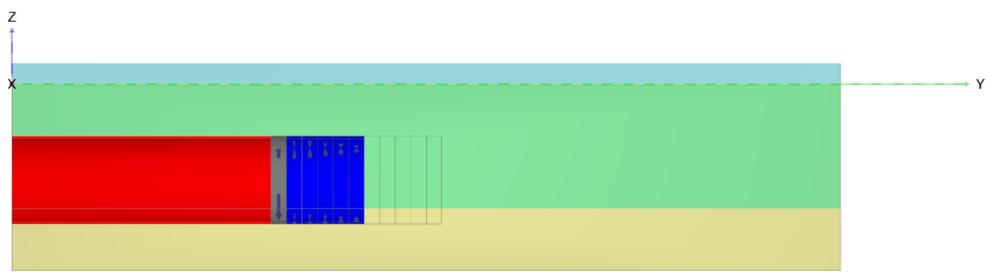

Figure 79: Selection of soil volumes (0 m - 25 m)

5. 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.

> **Note:** 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 of the REFERENCE MANUAL)

6. The interface is already activated. To activate the plate and the contraction in the first 25 m of the tunnel:
   a. 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 80 (on page 107))
   b. In the **Selection explorer** activate plate and surface contraction by checking the corresponding boxes.
   c. 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 81 (on page 107)). 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.

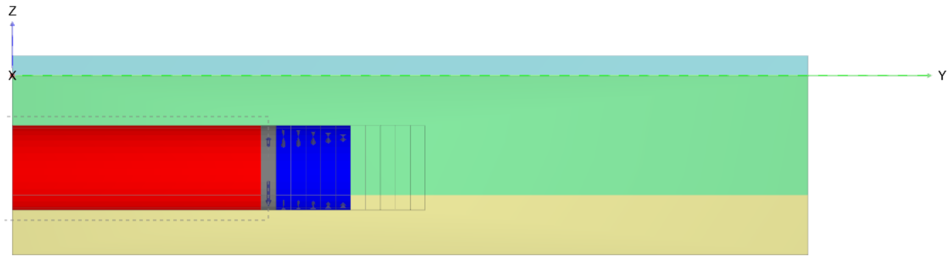

Figure 80: Selection of plate (0 m - 25 m)

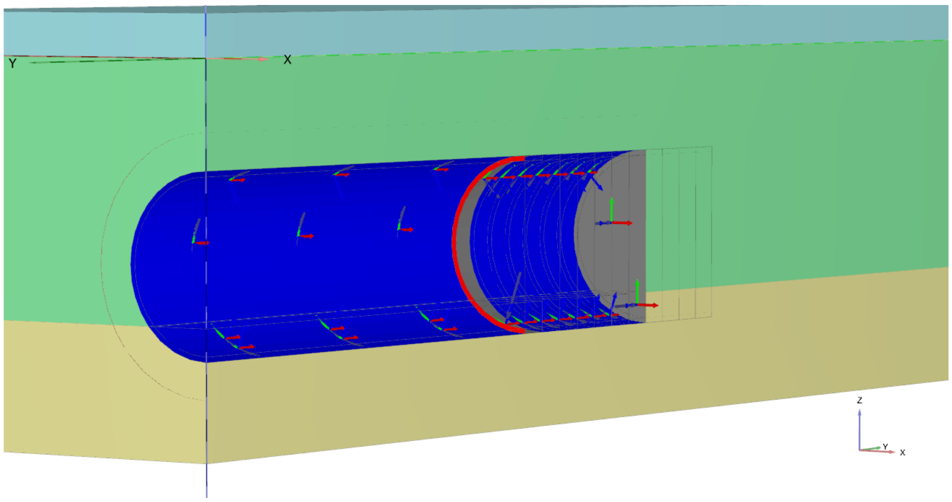

Figure 81: Selection of soil surfaces (25.0 m)

7. Click the **Preview** button to get a preview of everything that has been defined (Figure 82 (on page 107)). Make sure that both grout pressure and tunnel face pressure are applied and that both increase from top to bottom.

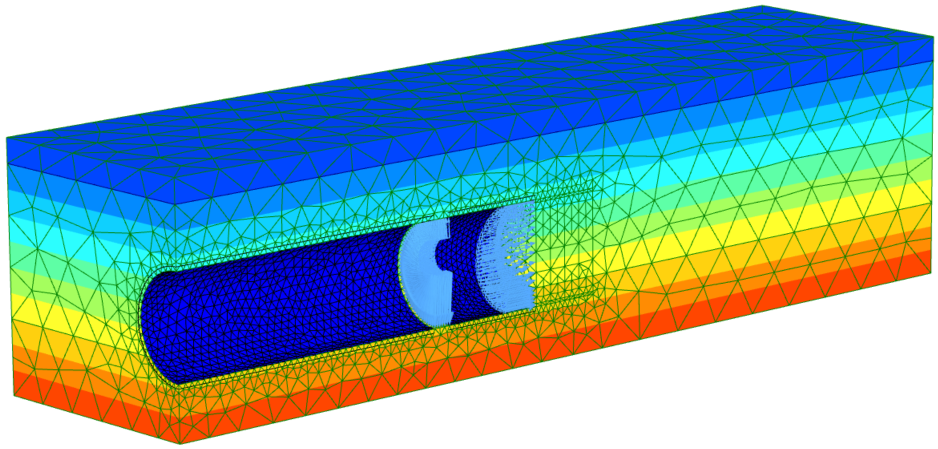

Figure 82: Preview of the Phase 1

5.6.3 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.

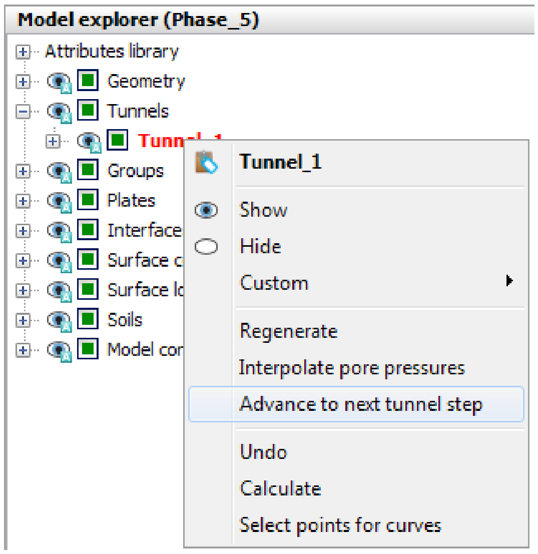

Figure 83: The Advance to next tunnel step option from Model explorer

1. Add a new phase
2. 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.

5.6.4 Phase 3: TBM advancement 2

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

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

5.6.5 Phase 4: TBM advancement 3

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

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

5.6.6 Phase 5: TBM advancement 4

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

1. Add a new phase
2. In the **Model explorer** expand *Tunnels* and right-click *Tunnel_1*. Then click on **Advance to next tunnel step**.
3. 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.

5.7 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:

1. Select the last calculation phase (Phase 5) in the *Phases explorer*
2. 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.
3. 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 84 (on page 110)).
4. 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 1.95m.
   - In the **View** menu select **View Point** > **Right view**.
   - In the **Mesh** menu make visible the **Cluster borders**.
   - In the **View** menu select the **Distribution plane** option.

   As a result, the window with the cross section is displayed (Figure 85 (on page 110)). The maximum settlement at ground level is about 1.9 cm.

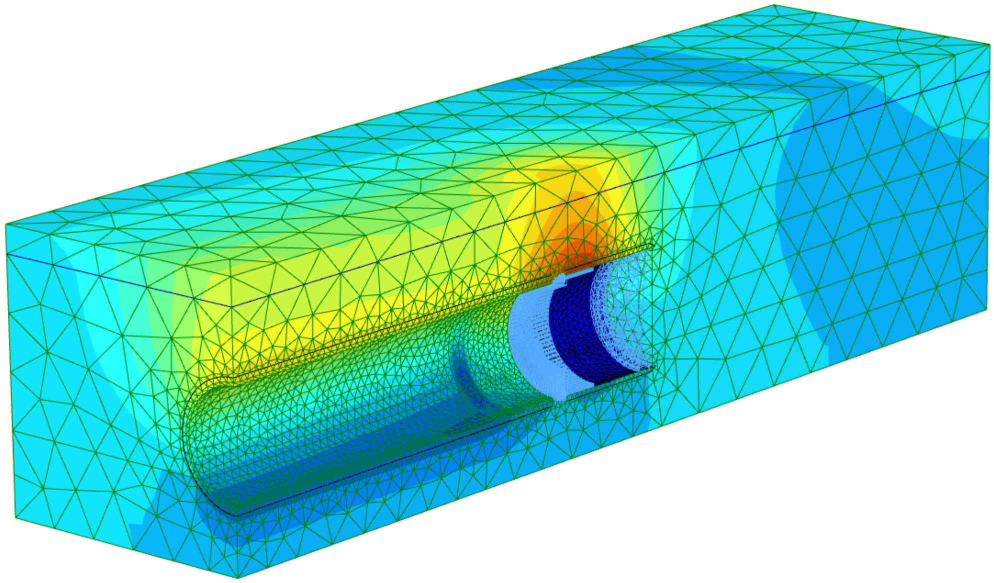

Figure 84: Total vertical displacements after the final phase $u_z \approx 3.1cm$

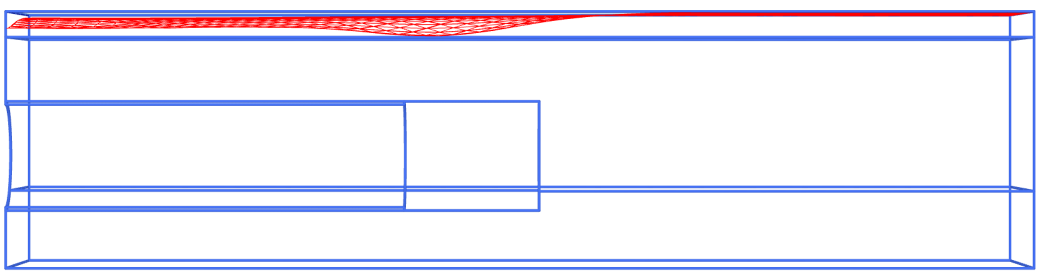

Figure 85: Settlement trough at ground level $u_z \approx 1.9cm$