6

Excavation of an NATM tunnel [GSE]

6.1 Introduction

This tutorial illustrates the use of PLAXIS 2D for the analysis of the construction of a NATM tunnel. The NATM is a technique in which ground exposed by excavation is stabilised with shotcrete to form a temporary lining.

Objectives

- Modelling the construction of an NATM tunnel using the **Deconfinement** method.
- Using **Gravity loading** to generate initial stresses.

6.2 Geometry

The geometry of the tunnel is shown in Figure 6–1 (p. 109) .

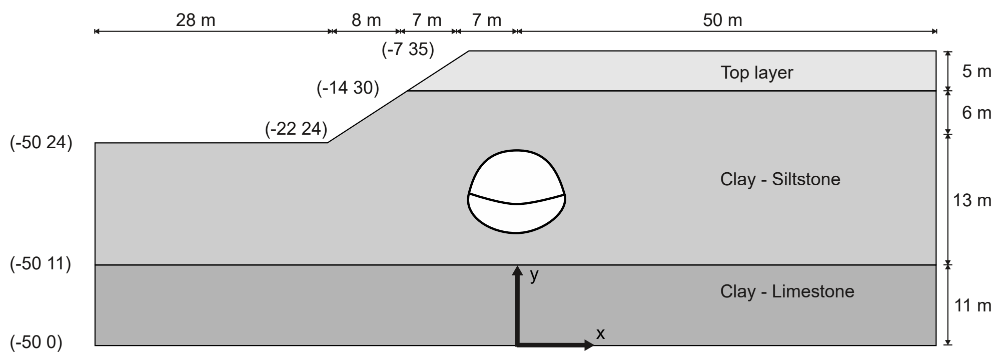

Figure 6–1: Geometry of the project

6.3 Create a new project

To create a new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet make sure that **Model** is set to **Plane strain** and that **Elements** is set to **15-Noded**.
4. Define the limits for the soil contour as x_{min} = -50 m, x_{max} = 50 m, y_{min} = 0 m and y_{max} = 35 m.

6.4 Define the soil stratigraphy

The basic stratigraphy will be created using the **Borehole** feature. In the model 11 m of the Clay-limestone layer is considered. The bottom of this layer is considered as reference in y direction (y_{min} = 0).

To define the soil stratigraphy:

1. Click the **Create borehole** button and create the first borehole at x = -22 m.
2. In the **Modify soil layers** window create three soil layers.
   a. **Layer number 1**: both **Top** and **Bottom** lie in 24, which means that layer 1 has a depth equal to zero in Borehole_1.
   b. **Layer number 2**: lies from **Top** = 24 to **Bottom** = 11.
   c. **Layer number 3**: lies from **Top** = 11 to **Bottom** = 0.
3. At the bottom of the **Modify soil layers** window click on the **Boreholes** button.
4. In the appearing menu select the **Add** option.

   The **Add borehole** window pops up.

5. Specify the location of the second borehole (x = -14).
6. Note that the soil layers are available for Borehole_2.
   a. **Layer number 1**: both **Top** and **Bottom** lie in 30, which means that layer number 1 has a depth equal to zero in Borehole_2. Notice how depth of layer 2 in Borehole_2 is higher than layer 1 in borehole_1.
   b. **Layer number 2**: lies from **Top** = 30 to **Bottom** = 11.
   c. **Layer number 1**: lies from **Top** = 11 to **Bottom** = 0.
7. Create a new borehole (Borehole_3) at x = -7.
8. In Borehole_3:
   a. **Layer number 1**: lies from **Top** = 35 to **Bottom** = 30, which means that layer 1 has a non-zero thickness in of Borehole_3.
   b. **Layer number 2**: lies from **Top** = 30 to **Bottom** = 11.
   c. **Layer number 3**: lies from **Top** = 11 to **Bottom** = 0.
9. In all the boreholes the water level is located at y = 0 m.
10. Specify the soil layer distribution as shown in Figure 6–2 (p. 110).

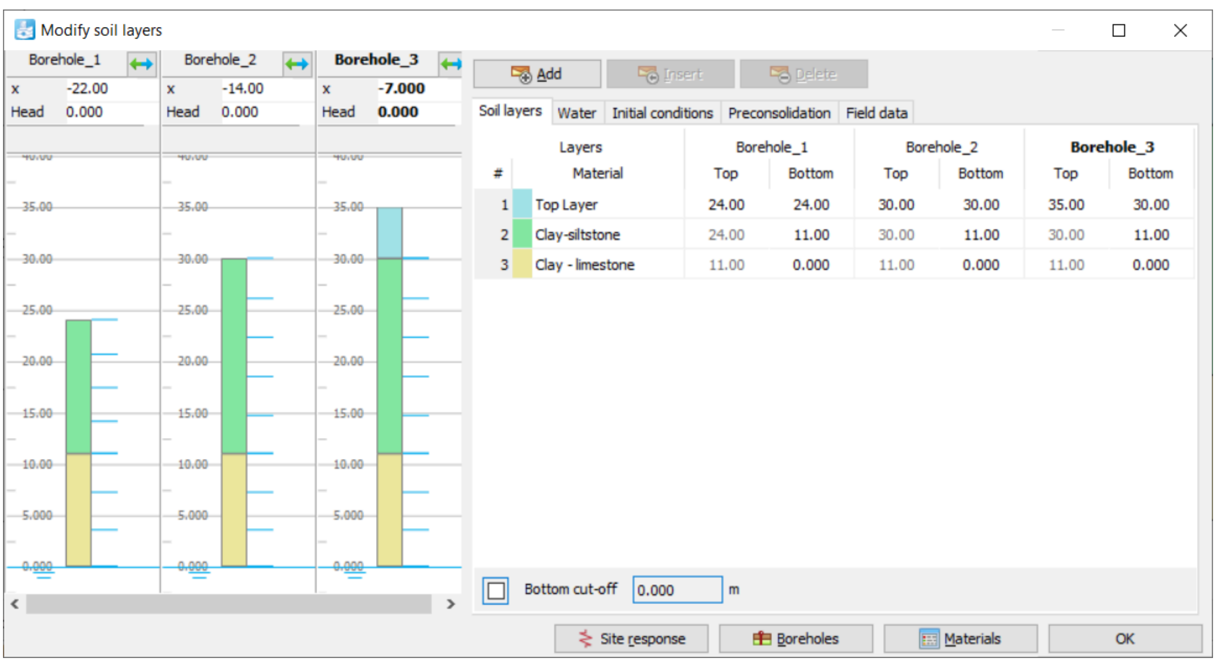

Figure 6–2: Soil layer distribution

6.5 | Create and assign material data sets

Three material sets need to be created for the soil layers.

Note that the layering of the model left from the first borehole is based on Borehole_1 and the layering right from the last borehole is based on Borehole_3. Hence, no borehole is needed at x = -50 m or x = 50 m.

The layers have the following properties as shown in Table 6–1 (p. 111):

Table 6–1: Material properties of the soil layer

Property	Name	Top layer	Unit
General			
Soil model	-	Hardening soil	-
Drainage type	-	Drained	-
Unsaturated unit weight	γ_{unsat}	20	kN/m^3
Saturated unit weight	γ_{sat}	22	kN/m^3
Mechanical			
Secant stiffness in standard drained triaxial test	$E_{50}{}^{ref}$	$40 \cdot 10^3$	kN/m^2
Tangent stiffness for primary oedometer loading	$E_{oed}{}^{ref}$	$40 \cdot 10^3$	kN/m^2
Unloading / reloading stiffness	$E_{ur}{}^{ref}$	$120 \cdot 10^3$	kN/m^2
Poisson's ratio	ν_{ur}	0.2	-
Power for stress-level dependency of stiffness	m	0.5	-
Cohesion	c'_{ref}	10	kN/m^2
Friction angle	φ'	30	°
Interfaces			
Strength determination	-	Rigid	-
Interface reduction factor	R_{inter}	1.0	-

Table 6–2: Material properties of the soft rock layers

Parameter	Name	Clay-silt stone	Clay-limestone	Unit
General				
Soil model	-	Hoek-Brown	Hoek-Brown	-
Type of material behaviour	-	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	25	24	kN/m^3
Saturated unit weight	γ_{sat}	25	24	kN/m^3
Mechanical				
Young's modulus	E_{rm}	$1.0 \cdot 10^6$	$2.5 \cdot 10^6$	kN/m^2
Poisson's ratio	ν (nu)	0.25	0.25	-
Uniaxial compressive strength	$\lvert\sigma_{ci}\rvert$	$25 \cdot 10^3$	$50 \cdot 10^3$	kN/m^2

Mechanical				
Material constant for the intact rock	m_i	4	10	-
Geological Strength Index	*GSI*	40	55	-
Disturbance factor	*D*	0.2	0.0	-
Dilatancy parameter	ψ_{max}	30	35	°
Dilatancy parameter	σ_ψ	400	1000	kN/m^2
Interfaces				
Strength determination	-	Manual	Rigid	-
Interface reduction factor	R_{inter}	0.5	1.0	-

1. Create soil material data sets according to Table 6–1 (p. 111) and assign them to the corresponding layer Figure 6–2 (p. 110). Then assign the values for the soft rock layers as per Table 6–2 (p. 111), find the analysis for various strength parameters from the emerging window as shown in Figure 6–3 (p. 112).
2. Close the **Modify soil layers** window and proceed to the **Structures mode** to define the structural elements.

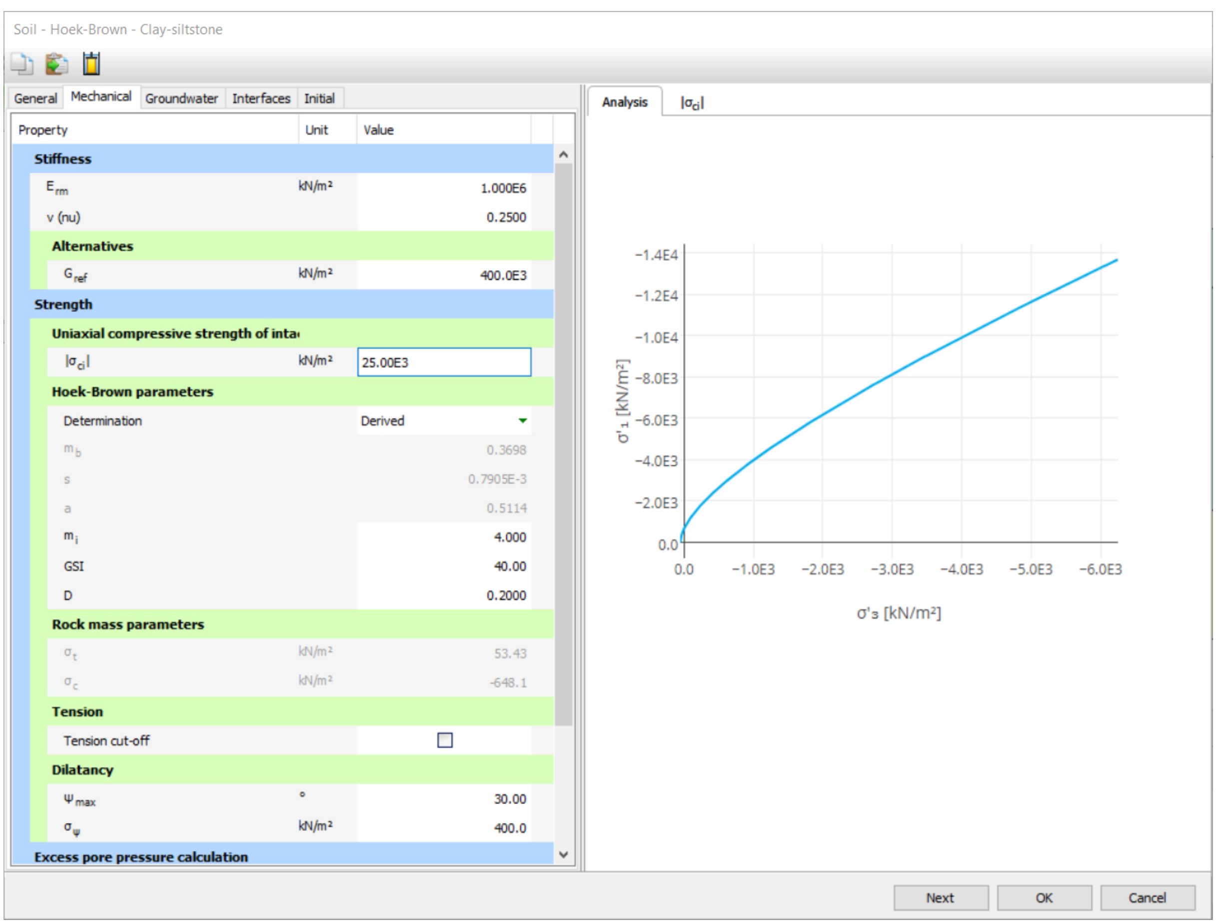

Figure 6–3: Mechanical Parameter

6.6 Define the tunnel

1. In the **Structures mode** click on the **Create tunnel** button in the side toolbar and click on (0 16) in the drawing area to specify the location of the tunnel.

   The **Tunnel designer** window pops up.
2. The default shape option (**Free**) will be used. The default values of the rest of the parameters defining the location of the tunnel in the model are valid as well.
3. Click on the **Segments** tab.
4. Click the **Add section** button in the side toolbar. In the tunnel Selection Explorer.

   a. Set the **Segment type** to **Arc**.
   b. Set **Radius** to 10.4 m
   c. Set the **Segment angle** to 22°.
5. The default values of the remaining parameters are valid.
6. Click the **Add section** button to add a new arc segment.

   a. Set **Radius** to 2.4 m.
   b. Set the **Segment angle** to 47°.
   c. The default values of the remaining parameters are valid.
7. Click the **Add section** button to add a new arc segment.

   a. Set **Radius** to 5.8 m.
   b. Set the **Segment angle** to 50°.
   c. The default values of the remaining parameters are valid.
8. Click the **Extend to symmetry axis** option to complete the right half of the tunnel.

   A new arc segment is automatically added closing the half of the tunnel.
9. Click the **Symmetric close** button to complete the tunnel. Four new arc segment are automatically added closing the tunnel.
10. Click on the **Subsections** tab.
11. Click the **Add** button to add a new subsection.

    Note:
    - The current subsection will be used to separate the top heading (upper excavation cluster) from the invert (lower excavation cluster).
    - While creating the new subsection some warnings and/or errors will be displayed. For this tutorial, ignore them and continue with the instructions.

    With the created subsection selected, in the **Selection Explorer** do the following:

a. Set the **Offset 2** as 3 m.
b. From the **Segment type** drop-down menu select **the Arc** option.
c. Set **Radius** to 11 m.
d. **Segment angle** to 360°.

12 Click the **Select multiple objects** button and select all the geometric entities in the slice.

13 Click the **Intersect** button .

14 Delete the part of the subsection outside of the slice by selecting it in the display area and clicking the **Delete** button in the side toolbar.

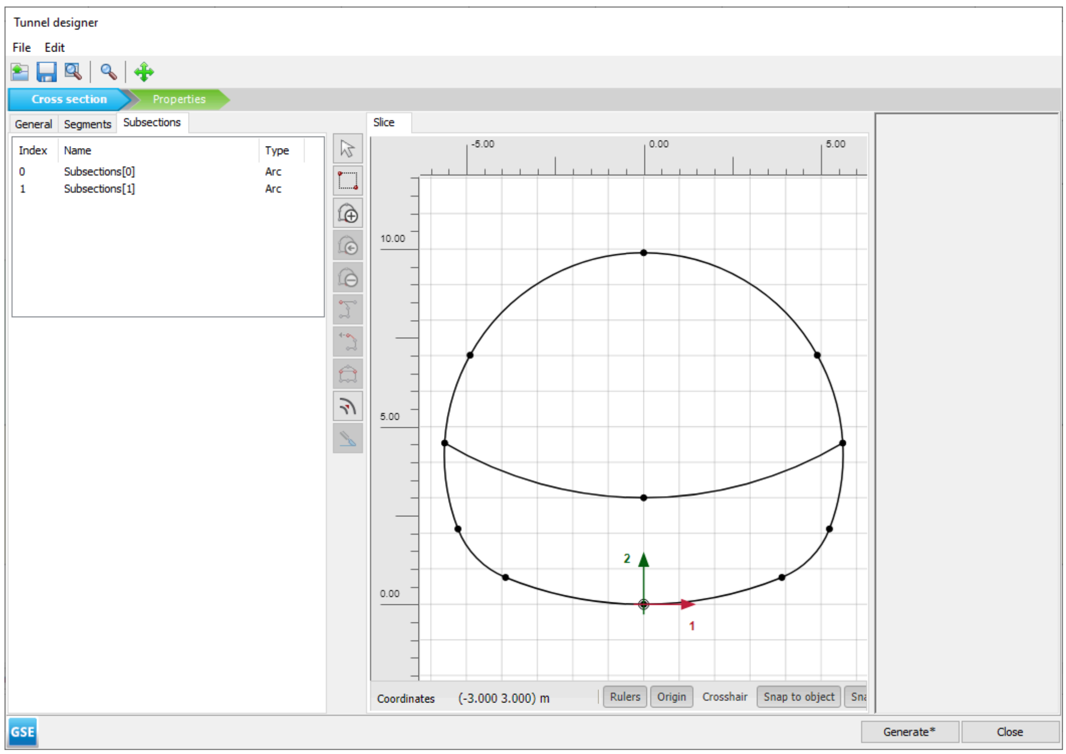

Figure 6–4: Segments in the tunnel cross section

15 Proceed to the **Properties** tabsheet.

16 Multi-select the polycurves in the display area, right click and in the appearing menu select **Create** > **Create Plate**.

17 The various segments in the tunnel cross section can be seen in Figure 6–4 (p. 114).

18 Press Ctrl + M to open the **Material sets** window. Create a new material dataset for the created plates according to Table 6–3 (p. 114).

Table 6–3: Material properties of the plates

Parameter	Name	Lining	Unit
General			
Material type	-	Elastic	-
Unit Weight	w	5	kN/m/m

Parameter	Name	Lining	Unit
General			
Prevent punching	-	No	-
Mechanical			
Isotropic	-	True	-
Axial stiffness	EA_1	$6.0 \cdot 10^6$	kN/m
Bending stiffness	EI	$20 \cdot 10^3$	kNm^2/m
Poisson's ratio	v	0.15	-

19. Multi-select the created plates and in the **Selection explorer**, assign the material **Lining** to the selected plates.
20. Create negative interfaces to the lines defining the shape of the tunnel (not the excavation levels or subsection).

The final tunnel view in the **Tunnel designer** window is shown in Figure 6–5 (p. 115):

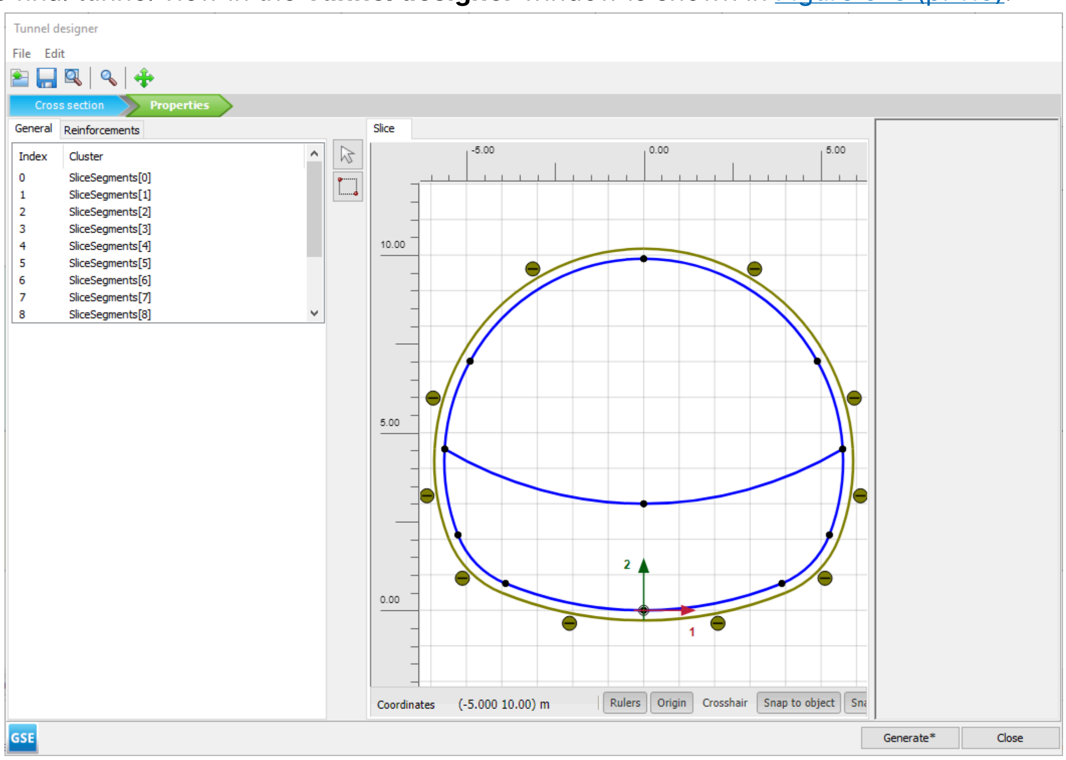

Figure 6–5: Final tunnel

21. Click on **Generate** to update the tunnel in the model and click **Close**.

6.7 Generate the mesh

The default global coarseness parameter (**Medium**) can be accepted in this case.

1. Proceed to the **Mesh mode**.

2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 6–6 (p. 116).

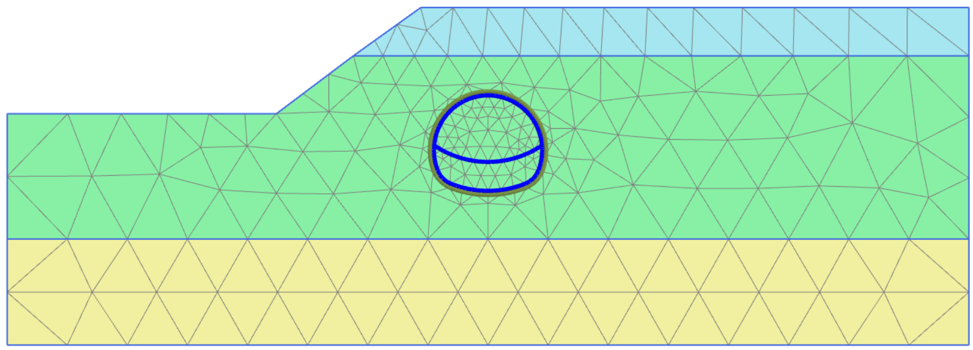

Figure 6–6: The generated mesh

4. Click the **Close** tab to close the Output program.

6.8 Define and perform the calculation

To simulate the construction of the tunnel a staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. The calculation phases are **Plastic** analyses, **Staged construction**. The three-dimensional arching effect is emulated by using the so-called β-method. The idea is that the initial stresses p_K acting around the location where the tunnel is to be constructed are divided into a part (1-β) p_K that is applied to the unsupported tunnel and a part **Deconfinement** method that is applied to the supported tunnel.

To apply this method in PLAXIS 2D, one can use the **Deconfinement** option, which is available for each deactivated soil cluster in the model explorer. **Deconfinement** is defined as the aforementioned factor (1-β). For example, if 60% of the initial stresses in a deactivated soil cluster should disappear in the current calculation phase (so the remaining 40% is to be considered later), it means that the **Deconfinement** (1-β) parameter of that inactive cluster should be set to 60%. The value of **Deconfinement** can be increased in subsequent calculation phases until it reaches 100%.

To define the calculation process follow these steps:

6.8.1 Initial phase

1. Click on the **Staged construction** tab to proceed with the definition of the calculation phases.
2. The initial phase has already been introduced. Note that the soil layers are not horizontal. It is not recommended in this case to use the **K0 procedure** to generate the initial effective stresses. Instead **Gravity loading** will be used. This option is available in the **General** subtree of the **Phases** window.
3. Water will not be considered in this example. The general phreatic level should remain at the model base.

4. Make sure that the tunnel is inactive.

6.8.2 Phase 1: First tunnel excavation (deconfinement)

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** mode deactivate the upper cluster in the tunnel. Do NOT activate the tunnel lining.
3. While the deactivated cluster is still selected, in the **Selection explorer** set **Deconfinement(1 - β)** to 60 %.

The model for Phase 1 is displayed in Figure 6–7 (p. 117).

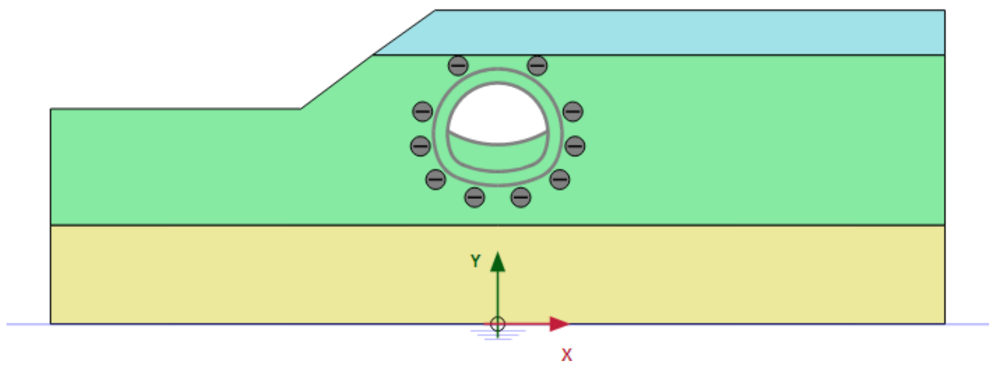

Figure 6–7: Configuration of Phase 1

6.8.3 Phase 2: First (temporary) lining

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** mode, **activate** the **lining** and **interfaces** of the part of the tunnel excavated in the previous phase.
3. Select the deactivated cluster. In the **Selection explorer** set **Deconfinement** to 100 % as shown in Figure 6–8 (p. 117).

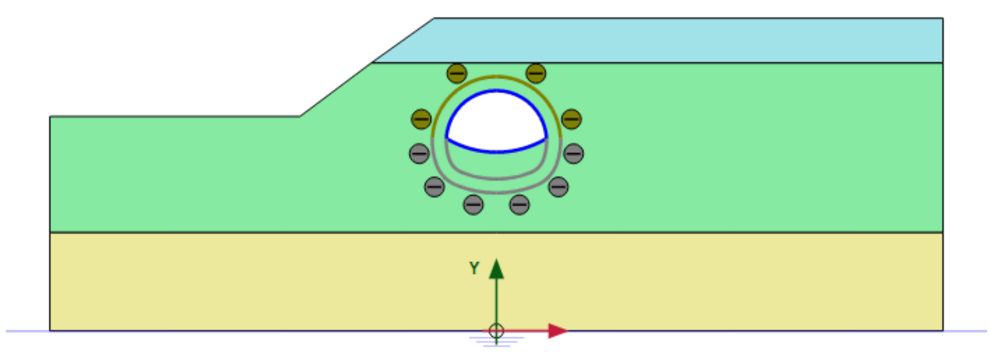

Figure 6–8: Configuration of Phase 2

6.8.4 Phase 3: Second tunnel excavation (deconfinement)

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction mode** deactivate the lower cluster (invert) and the temporary lining in the middle of the tunnel.
3. While the lower deactivated cluster is still selected, set in the **Selection explorer** **Deconfinement** to 60%.
4. The model for phase 3 can be seen in Figure 6–9 (p. 118).

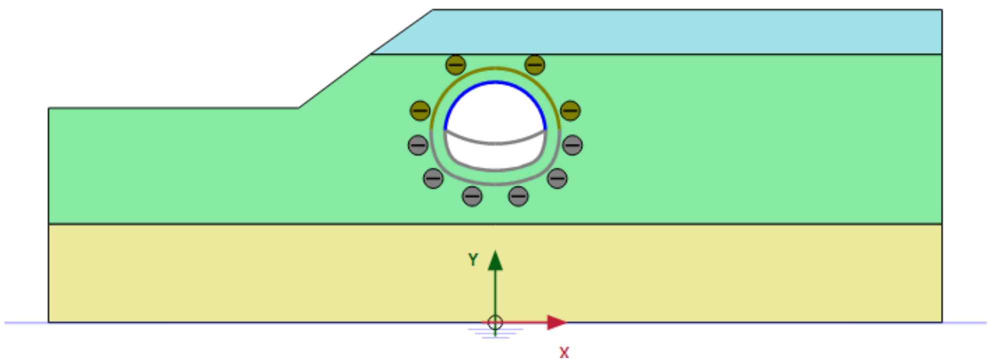

Figure 6–9: Configuration of Phase 3

6.8.5 Phase 4: Second (final) lining

1. Click the **Add phase** button to create a new phase.
2. Activate the remaining lining and interfaces.

   All the plates and interfaces around the full tunnel are active.
3. Select the lower deactivated cluster. In the **Selection explorer** set **Deconfinement** to 100 %.
4. The model for phase 4 can be seen in Figure 6–10 (p. 118).

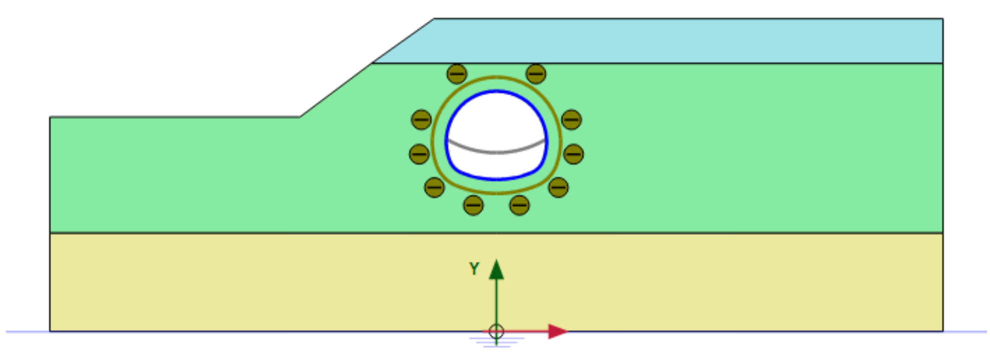

Figure 6–10: Configuration of Phase 4

6.8.6 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select a node at the slope crest point and the tunnel crest. These points might be of interest to evaluate the deformation during the construction phases.
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button.

6.9 Results

After the calculation, select the last calculation phase and click the **View calculation results** button. The **Output** program is started, showing the deformed mesh at the end of the calculation phases as shown in Figure 6–11 (p. 119):

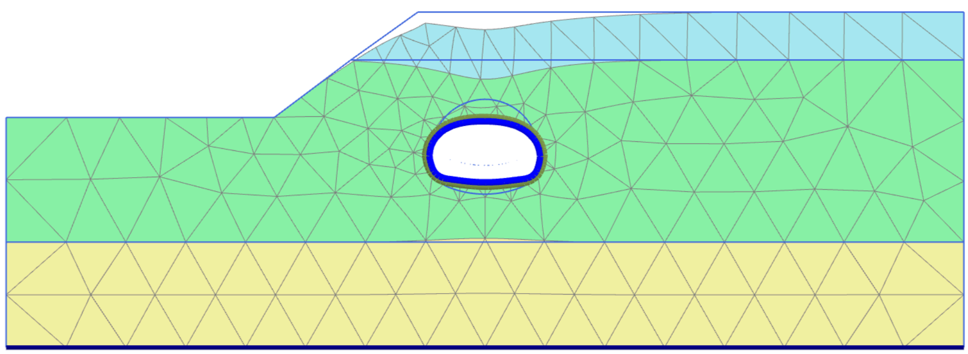

Figure 6–11: The deformed mesh at the end of the final calculation phase

To display the bending moments resulting in the tunnel:

1. To select the lining of all the tunnel sections, click the corresponding button in the side toolbar and drag the mouse to define a rectangle where all the tunnel sections are included. Select the **Plate** option in the appearing window as shown in Figure 6–12 (p. 119):

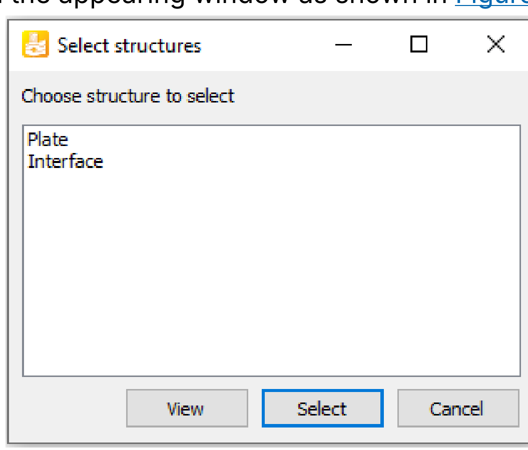

Figure 6–12: Select structures window

2. Click **View**.

   Note that the tunnel lining is displayed in the **Structures** view.

3. From the **Forces** menu select the **Bending moment M** option. The result, scaled by a factor of 0.5 is displayed in Figure 6–13 (p. 120).

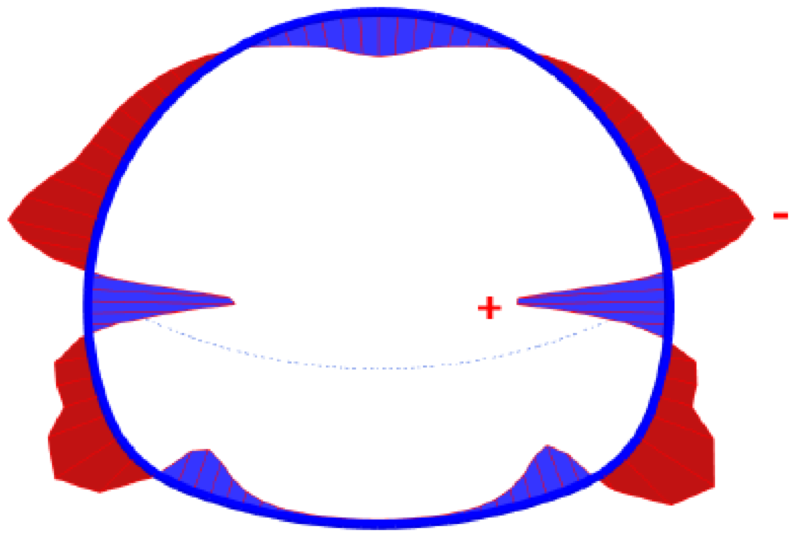

Figure 6–13: Resulting bending moments in the NATM tunnel