5

Settlements due to tunnel construction [GSE]

In this tutorial the construction of a shield tunnel in medium soft soil and the influence on a pile foundation 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 re-distributions and deformations in the soil as a result of the tunnel construction process. 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 tutorial shows an example of such an analysis.

Objectives

- Modelling of the tunnel boring process
- Modelling undrained behaviour using the **Undrained (B)** option

Geometry

The tunnel considered in this tutorial has a diameter of 5 m and is located at an average depth of 20 m.
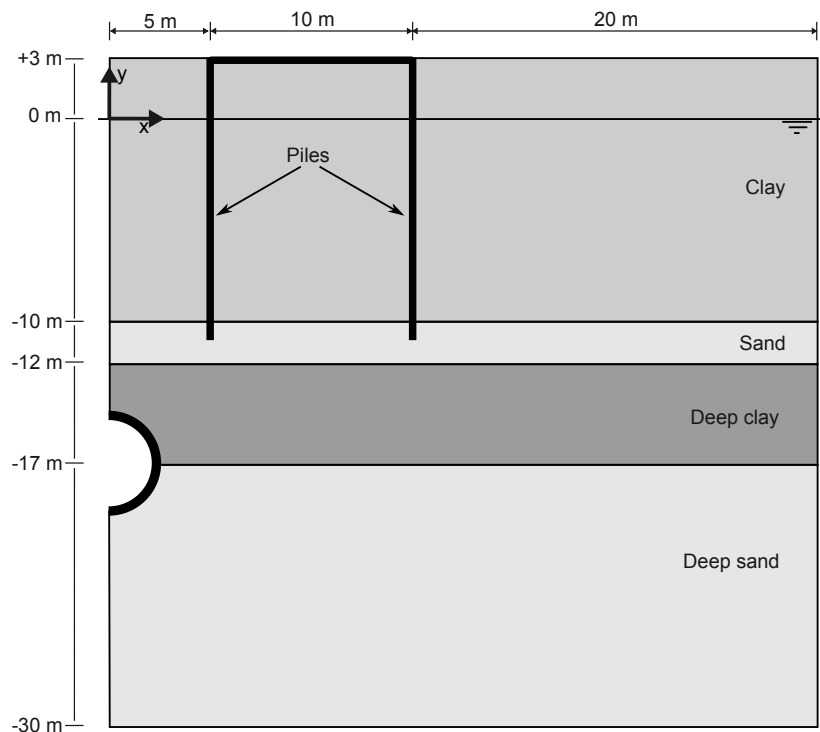

Figure 43: Geometry of the tunnel project with an indication of the soil layers

5.1 Create new project

To create the 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 keep the default options for **Model** (**Plane strain**), and **Elements** (**15-Noded**).
4. Set the model **Contour** to x_{min} = 0 m, x_{max} = 35 m, y_{min} = -30 m and y_{max} = 3 m.
5. Keep the default values for units and constants and press **OK** to close the **Project properties** window.

5.2 Define the soil stratigraphy

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 m thick fine sand layer. This layer is used as a foundation layer for old wooden piles on which traditional brickwork houses were built. The pile foundation of such a building is modelled next to the tunnel. Displacements of these piles may cause damage to the building, which is highly undesirable. Below the sand layer there is a 5 m thick deep loamy clay layer.

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
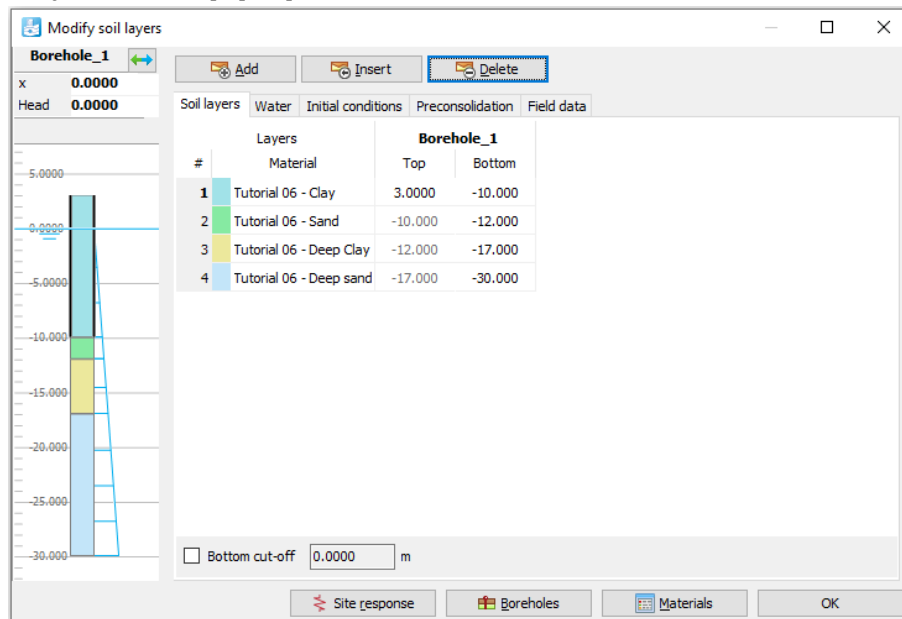   The **Modify soil layers** window pops up.

*Figure 44: The soil stratigraphy in the **Modify soil layers** window*

2. Create the soil stratigraphy as shown in the following figure:
3. Keep the **Head** in the borehole to 0 m.

5.2.1 Create and assign material data sets

Four data sets need to be created for the clay and sand layers.

For the upper clay layer the stiffness and shear strength increase with depth. Therefore values for E'$_{inc}$ and s$_{u,inc}$ are entered in the **Advanced** subtree. The values of E'$_{ref}$ and s$_{u,ref}$ become the reference values at the reference level y$_{ref}$. Below y$_{ref}$ the actual values of E' and s$_u$ increase with depth according to:

$$E^{'}(y) = E_{ref}{'} + E_{inc}{'}(y_{ref} - y)$$

$$s_u(y) = s_{u,ref} + s_{u,inc}(y_{ref} - y)$$

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 13 (on page 67) and assign them to the corresponding clusters in the geometry model.

The layers have the following properties:

Table 7: Material properties of the clay layers

Parameter	Name	Clay	Deep clay	Unit
General				
Material model	-	Mohr-Coulomb	Mohr-Coulomb	-
Type of material behaviour	-	Undrained (B)	Undrained (B)	-
Soil unit weight above phreatic level	γ_{unsat}	15	16	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	18	18.5	kN/m^3
Parameters				
Young's modulus at reference level	E'	$3.4 \cdot 10^3$	$9.0 \cdot 10^3$	kN/m^2
Poisson's ratio	v'	0.33	0.33	-
Undrained shear strength at reference level	$s_{u,ref}$	5	40	kN/m^2
Young's modulus increment	$E_{inc}{'}$	400	600	kN/m^3
Reference level	y_{ref}	3.0	-12	m

Parameter	Name	Clay	Deep clay	Unit
Undrained shear strength inc.	$s_{u,inc}$	2	3	kN/m^2
Reference level	y_{ref}	3.0	-12	m
Groundwater				
Data set	-	Standard	Standard	-
Soil type	-	Coarse	Coarse	-
Use defaults	-	None	None	-
Permeability in horizontal direction	k_x	$0.1 \cdot 10^{-3}$	0.01	m/day
Permeability in vertical direction	k_y	$0.1 \cdot 10^{-3}$	0.01	m/day
Interfaces				
Interface strength	-	Rigid	Manual	-
Strength reduction factor	R_{inter}	1.0	0.7	-
Initial				
K$_0$ determination	-	Manual	Manual	-
Lateral earth pressure coefficient	$K_{0,x}$	0.6	0.7	-

Table 8: Material properties of the sand layers

Parameter	Name	Sand	Deep sand	Unit
General				
Material model	-	HS small	HS small	-
Type of material behaviour	-	Drained	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	16.5	17	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20	21	kN/m^3
Parameters				
Secant stiffness in standard drained triaxial test	$E_{50}{}^{ref}$	$25 \cdot 10^3$	$42 \cdot 10^3$	kN/m^2
Tangent stiffness for primary oedometer loading	$E_{oed}{}^{ref}$	$25 \cdot 10^3$	$42 \cdot 10^3$	kN/m^2
Unloading / reloading stiffness	$E_{ur}{}^{ref}$	$75 \cdot 10^3$	$126 \cdot 10^3$	kN/m^2

Parameter	Name	Sand	Deep sand	Unit
Power for stress-level dependency of stiffness	m	0.5	0.5	-
Cohesion	c_{ref}'	0	0	kN/m^2
Friction angle	φ'	31	35	°
Dilatancy angle	ψ	1	5	°
Shear strain at which $G_s = 0.722\ G_0$	$\gamma_{0.7}$	$0.2 \cdot 10^{-3}$	$0.13 \cdot 10^{-3}$	-
Shear modulus at very small strains	G_0^{ref}	$80 \cdot 10^3$	$110 \cdot 10^3$	kN/m^2
Poisson's ratio	v_{ur}'	0.2	0.2	-
Groundwater				
Data set	-	Standard	Standard	-
Soil type	-	Coarse	Coarse	-
Use defaults	-	None	None	-
Permeability in horizontal direction	k_x	1.0	0.5	m/day
Permeability in vertical direction	k_y	1.0	0.5	m/day
Interfaces				
Interface strength	-	Rigid	Manual	-
Strength reduction factor	R_{inter}	1.0	0.7	-
Initial				
K_0 determination	-	Automatic	Automatic	-
Over-consolidation ratio	*OCR*	1.0	1.0	-
Pre-overburden pressure	*POP*	0.0	0.0	-

To create the material sets, follow these steps:

1. Click the **Materials** button in the **Modify soil layers** window.
2. Create data sets under the **Soil and interfaces** set type according to the information given in Table 13 (on page 67).

5.3 Define the structural elements

The tunnel and the building are defined as structural elements.

5.3.1 Define the tunnel

The tunnel considered here is the right half of a circular tunnel. After generating the basic geometry, follow these steps to design the circular tunnel:

1. In the **Structures** mode click the **Create tunnel** button in the side toolbar and click at (0 -17) in the drawing area.
The **Tunnel designer** window pops up displaying the **General** tabsheet of the **Cross section** mode.

2. Select the **Circular** option in the **Shape type** drop-down menu.

3. Select the **Define right half** option in the **Whole or half tunnel** drop-down menu.

4. In the **Offset to begin point** group set **Axis 2** to -2.5. No change is required for the orientation axes.

5. Click the **Segments** tab 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.

6. In the **Segment** box set **Radius** to 2.5 m.
The generated segment is shown:
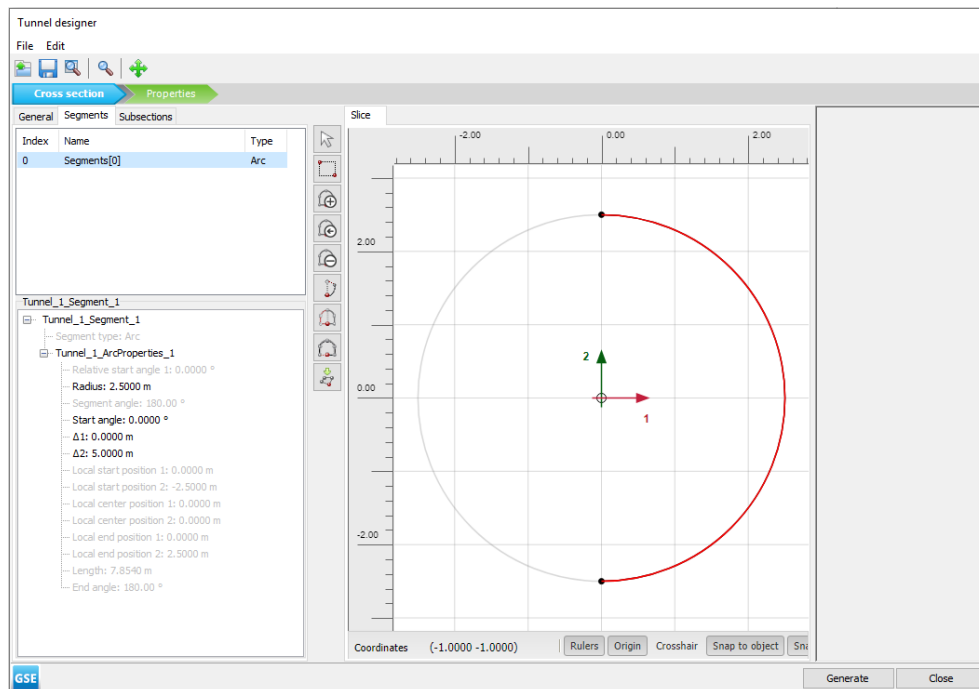
Figure 45: The geometry of the tunnel segment

> **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 the general drawing area).
> - The tunnel shape is composed of arcs with different radii (for example NATM tunnels).

7. Click the **Properties** tab to proceed to the corresponding tabsheet.
8. Right-click the segment in the display area and select the menu **Create** > **Create plate** option in the appearing menu.
9. In Tunnel Designer go to the Material property in the **Selection explorer** and click the plus button ⊞ to create a new material dataset. Specify the material parameters for the TBM according to the following table:

Table 9: Material properties of the plates

Parameter	Name	TBM	Lining	Building	Unit
Material behaviour	-	Elastic	Elastic	Elastic	-
Isotropic	-	Yes	Yes	Yes	-
Axial stiffness	EA_1	$63 \cdot 10^6$	$14 \cdot 10^6$	$1 \cdot 10^{10}$	kN/m
Bending stiffness	EI	$472.5 \cdot 10^3$	$143 \cdot 10^3$	$1 \cdot 10^{10}$	kNm^2/m
Weight	w	17.7	8.4	25	kN/m/m
Poisson's ratio	ν	0	0.15	0	-
Prevent punching	-	No	No	No	-

> **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.

10. Right-click the segment in the display area and select the **Create negative interface** option in the appearing menu.
11. Right-click the segment in the display area and select the **Create line contraction** option in the appearing menu. In the polycurve box specify a value of 0.5% for C$_{ref}$. The tunnel model is shown in the following figure.
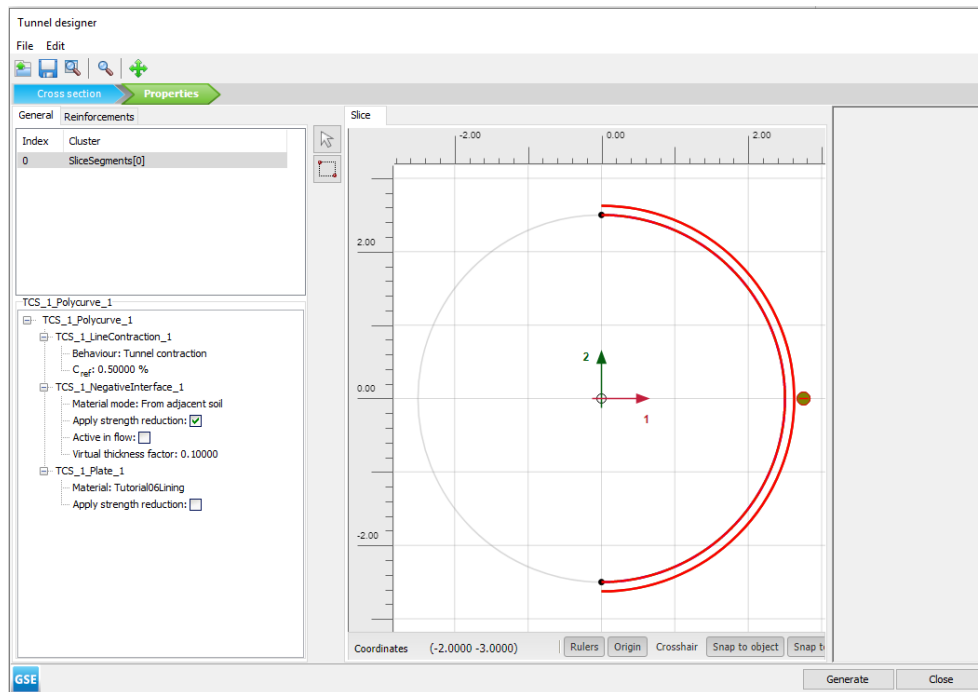
*Figure 46: Tunnel model in the **Properties** mode*

> **Note:**
>
> - A C_{ref} value of 0.5% corresponds to a volume loss of 0.5% of the tunnel volume. The actual strain that is applied to the line is half the applied contraction. Hence, the resulting liner contraction is 0.25%.
> - The entered value of contraction is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

12. Click on **Generate** to include the defined tunnel in the model.

13. Close the **Tunnel designer** window.

5.3.2 Define building

The building itself will be represented by a stiff plate founded on piles.

1. From the side bar, select **Create line** > **Create plate** and draw a plate from (5 3) to (15 3), representing the building.

2. Create a material set for the building according to Table 16 (on page 73) and assign it to the plate. Do not forget to also create the material set for the tunnel lining.

3. From the side bar, select **Create line** > **Create embedded beam row** and draw two piles (embedded beam rows) from (5 3) to (5 -11) and from (15 3) to (15 -11).

4. Create a material set for the foundation piles according to the table below and assign it to the foundation piles.

Table 10: Material properties of piles

Parameter	Name	Foundation piles	Unit
Material type	-	Elastic	-
Stiffness	E	$10 \cdot 10^6$	kN/m^2
Unit weight	γ	7.0	kN/m^3
Beam type	-	Predefined	-
Predefined beam type	-	Massive circular beam	-
Diameter	-	0.25	m
Pile spacing	$L_{spacing}$	3.0	m
Axial skin resistance	Axial skin resistance	Linear	-
	$T_{skin, start, max}$	1.0	kN/m
	$T_{skin, end, max}$	100.0	kN/m
Lateral resistance	Lateral resistance	Unlimited	-
Base resistance	F_{max}	100.0	kN
Interface stiffness factors	Default values	Yes	-

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

5.4 Generate the mesh

The default global coarseness parameter (**Medium**) can be accepted in this case. Note that the structural elements (plate and embedded beams) are internally automatically refined by a factor of 0.25.

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.

Figure 47: The generated mesh

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

5.5 Define and perform the calculation

To simulate the construction of the tunnel it is clear that a staged construction calculation is needed.

5.5.1 Initial phase

1. Proceed to the **Staged construction** mode to proceed with the definition of the calculation phases.
2. The initial phase has already been introduced. Keep its calculation type as **K0 procedure**. The water pressures can be generated on the basis of a general phreatic level at a level of y = 0.0 m as already defined in the borehole. Make sure that the building, foundation piles and tunnel lining are deactivated.

5.5.2 Phase 1: Building

The first calculation phase is used to activate the building.

1. Click the **Add phase** button 🔲 to create a new phase.
2. In the **Phases** window rename the phase as `Building`.
3. In the **Deformation control parameters** subtree select the **Ignore undr. behaviour (A,B)** option. The default values of the remaining parameters are valid for this phase.
4. In the drawing area activate the plate and the foundation piles.

5.5.3 Phase 2: TBM

1. Click the **Add phase** button 🔲 to create a new phase.
2. In the **Phases** window select the **Reset displacements to zero** option in the **Deformation control parameters** subtree.
3. In **Staged construction** multi-select the clusters inside the tunnel. In the **Selection explorer** deactivate the two soil clusters and set the **Water conditions** to **Dry**.
4. Activate the circular plate that in this phase represents the Tunnel Boring Machine (and thus has the TBM material set) as well as the negative interfaces. Note that contraction is not active in this phase.

5.5.4 Phase 3: TBM conicity

1. Click the **Add phase** button 🔲 to create a new phase.
2. Multi-select the plates. In the **Selection explorer** activate the contraction.

Note:

- The contraction represents the effects of the cone shape of the TBM (cutter head has larger diameter than the tail).
- The contraction of the tunnel lining by itself does not introduce 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.

5.5.5 Phase 4: Tail void grouting

At the tail of the tunnel boring machine (TBM), grout is injected to fill up the gap between the TBM and the final tunnel lining. The grouting process is simulated by applying a pressure on the surrounding soil.

1. Click the **Add phase** button 🔲 to create a new phase.
2. In the **Staged construction** mode deactivate the TBM (plates, negative interfaces and contraction).
3. Multi-select the clusters inside the tunnel. In the **Selection explorer** activate **WaterConditions**.

4. Select the **User-defined** option in the **Condition** drop-down menu and set p_{ref} to -230 kN/m^2. The pressure distribution in the tunnel is constant.

5.5.6 Phase 5: Lining installation

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** set the clusters inside the tunnel to **Dry**.
3. Activate the plates and the negative interfaces of the tunnel.
4. Since the plates now represent the final lining of the tunnel, assign the *Lining* material set to the plate elements.

5.5.7 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar.
2. Select some characteristic points for load-displacement curves (for example the corner point at the ground surface above the tunnel and the corner points of the building).
3. Click the **Calculate** button to calculate the project.
4. After the calculation has finished, save the project by clicking the Save button.

5.6 Results

After the calculation, select the last calculation phase and click the **View calculation results** button. The **Output** program is started, showing the deformed meshes at the end of the calculation phases:
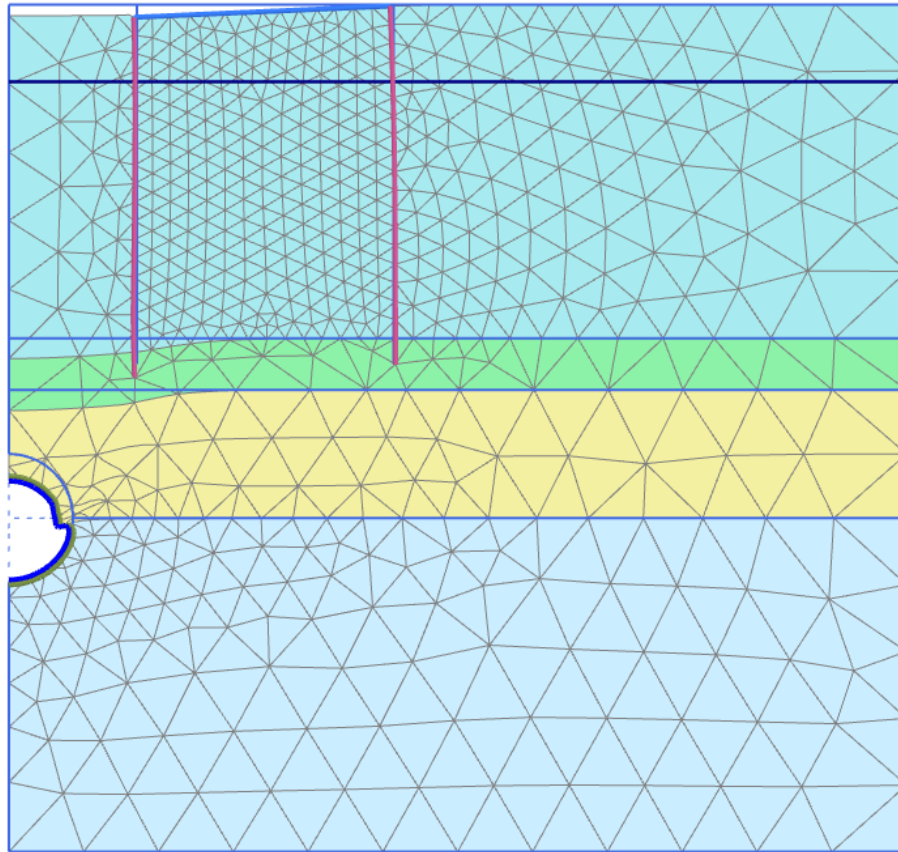
Figure 48: Deformed mesh after construction of the tunnel (Phase 5; scaled up 20 times)

As a result of the second 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 force related options from the **Force** menu. The plots of the axial forces and bending moment are scaled by factors of $5 \cdot 10^{-3}$ and 0.2 respectively.
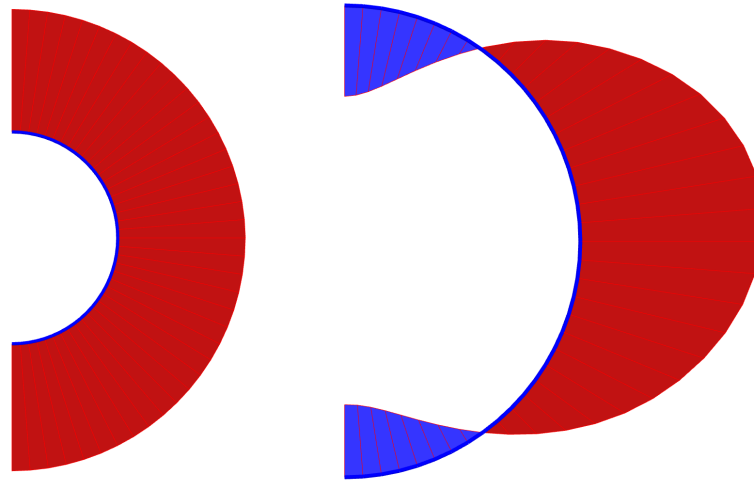
Figure 49: Axial forces and Bending moments in the lining after the second phase

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 in the final phase is lower than that after the second calculation phase.
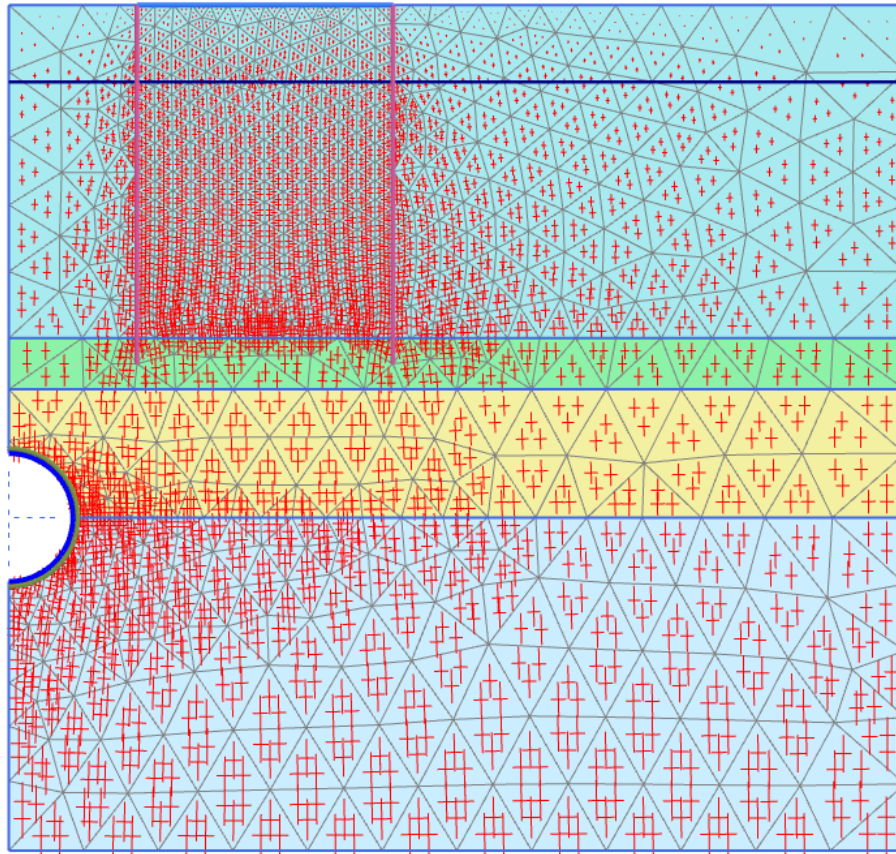
Figure 50: Effective principal stresses after the construction of the tunnel
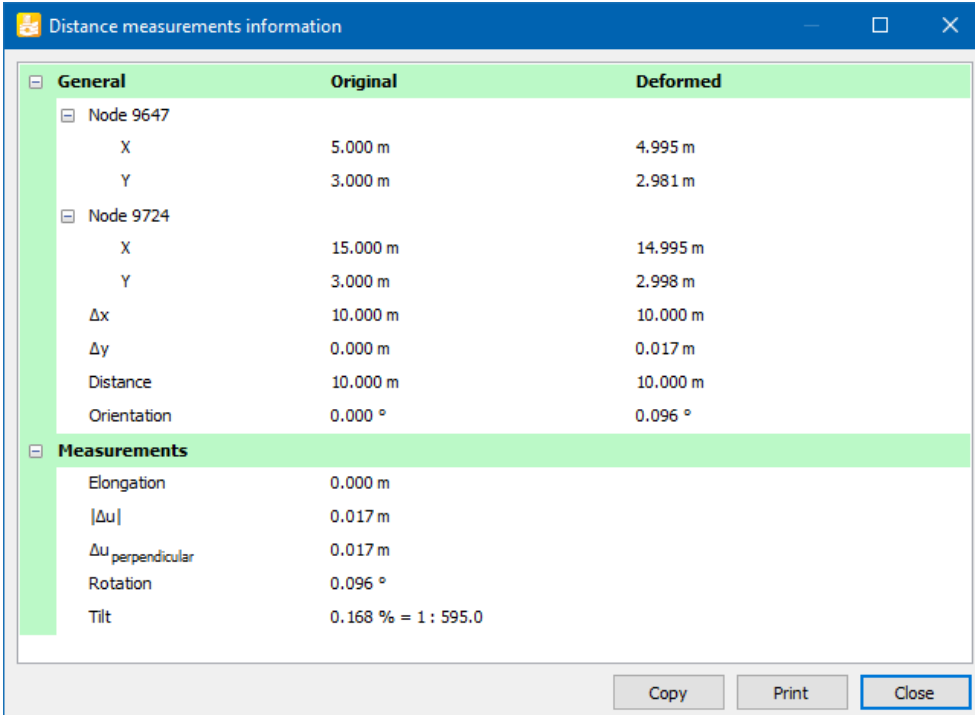
To display the tilt of the structure:

1. Click the **Distance measurement** button ▭ in the side toolbar.
2. Click the node located at the left corner of the structure (5 3).
3. Click the node located at the right corner of the structure (15 3).

The **Distance measurement information** window is displayed where the resulting tilt of the structure is given:

Figure 51: Distance measurement information window