6 SETTLEMENTS DUE TO TUNNEL CONSTRUCTION

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.

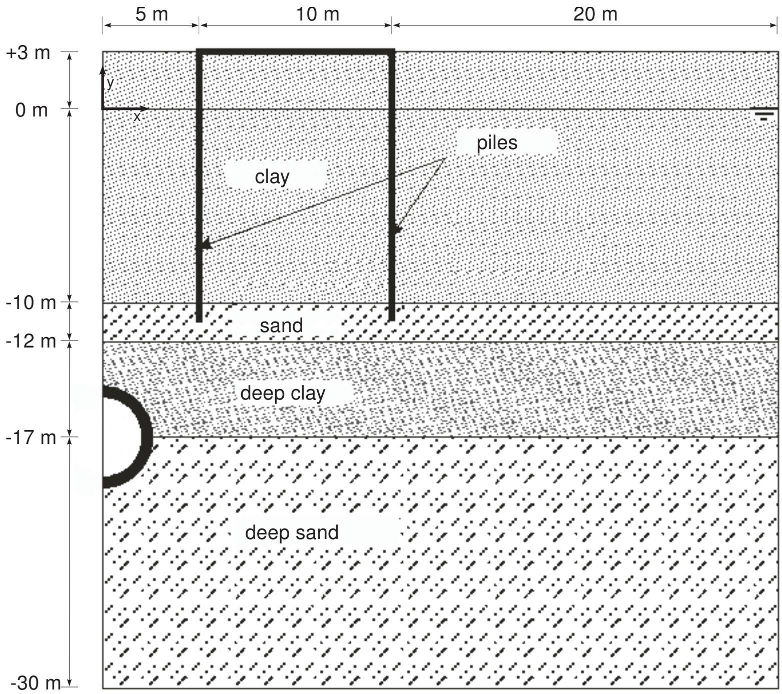

Figure 6.1 Geometry of the tunnel project with an indication of the soil layers

Objectives:

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

6.1 INPUT

The tunnel considered in this tutorial has a diameter of 5.0 m and is located at an average depth of 20 m. To create the geometry model, follow these steps:

General settings

- Start the Input program and select *Start a new project* from the *Quick select* dialog box.
- In the *Project* tabsheet of the *Project properties* window, enter an appropriate title.
- In the *Model* tabsheet keep the default options for *Model* (*Plane strain*), and *Elements* (*15-Noded*).
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 35.0 m, y_{min} = -30.0 m and y_{max} = 3.0 m.
- Keep the default values for units and constants and press *OK* to close the *Project properties* window.

Definition of 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 2.0 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 5.0 m thick deep loamy clay layer.

To define the soil stratigraphy:

- Create a borehole at x = 0. The *Modify soil layers* window pops up.
- Create the soil stratigraphy as shown in Figure 6.2.

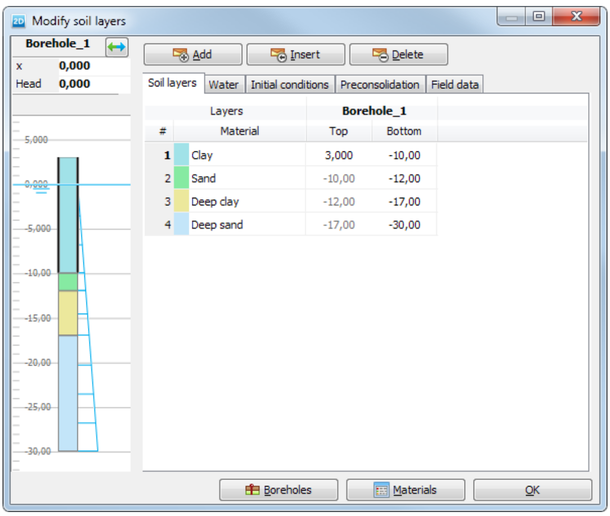

Figure 6.2 The soil stratigraphy in the *Modify soil layers* window

- Keep the *Head* in the borehole at 0.0 m.
- Open the *Material sets* window.
- Create data sets under the *Soil and interfaces* set type according to the information

given in Table 6.1.

Table 6.1 Material properties of soil in the tunnel project

Parameter	Name	Clay	Sand	Deep clay	Deep sand	Unit
General						
Material model	Model	Mohr-Coulomb	Hardening soil	Mohr-Coulomb	HS small	-
Drainage type	Type	Undrained (B)	Drained	Undrained (B)	Drained	-
Soil unit weight above p.l.	γ_{unsat}	15	16.5	16	17	kN/m^3
Soil unit weight below p.l.	γ_{sat}	18	20	18.5	21	kN/m^3
Parameters						
Young's modulus at reference level	E'	$3.4 \cdot 10^3$	-	$9.0 \cdot 10^3$	-	kN/m^2
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	-	$2.5 \cdot 10^4$	-	$4.2 \cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	-	$2.5 \cdot 10^4$	-	$4.2 \cdot 10^4$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	-	$7.5 \cdot 10^4$	-	$1.26 \cdot 10^5$	kN/m^2
Power for stress-level dependency of stiffness	m	-	0.5	-	0.5	-
Cohesion	c'_{ref}	-	0	-	0	kN/m^2
Undrained shear strength at reference level	$s_{u,ref}$	5	-	40	-	kN/m^2
Friction angle	φ'	-	31	-	35	°
Dilatancy angle	ψ	-	1.0	-	5	°
Shear strain at which $G_s = 0.722 G_0$	$\gamma_{0.7}$	-	-	-	$1.3 \cdot 10^{-4}$	-
Shear modulus at very small strains	G_0^{ref}	-	-	-	$1.1 \cdot 10^5$	kN/m^2
Poisson's ratio	ν'	0.33	0.3	0.33	0.3	-
Young's modulus inc.	E'_{inc}	400	-	600	-	kN/m^3
Reference level	y_{ref}	3.0	-	-12	-	m
Undrained shear strength inc.	$s_{u,inc}$	2	-	3	-	kN/m^2
Reference level	y_{ref}	3.0	-	-12	-	m
Groundwater						
Horizontal permeability	k_x	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
Vertical permeability	k_y	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
Interfaces						
Interface strength type	Type	Rigid	Rigid	Manual	Manual	-
Interface strength	R_{inter}	1.0	1.0	0.7	0.7	-
Initial						
K_0 determination	–	Manual	Automatic	Manual	Automatic	-
Lateral earth pressure coefficient	$K_{0,x}$	0.60	0.485	0.60	0.4264	-
Over-consolidation ratio	OCR	-	1.0	-	1.0	-
Pre-overburden ratio	POP	-	0.0	-	0.0	-

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 6.1 and assign them to the corresponding clusters in the geometry model.

6.1.1 DEFINITION OF STRUCTURAL ELEMENTS

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:

- In the *Structures* mode click the *Create tunnel* button in the side toolbar and click at (0.0 -17.0) in the drawing area. The *Tunnel designer* window pops up displaying the *General* tabsheet of the *Cross section* mode.
- Select the *Circular* option in the *Shape type* drop-down menu.
- Select the *Define right half* option in the *Whole or half tunnel* drop-down menu.
- In the *Offset to begin point* group set *Axis 2* to -2.5. No change is required for the orientation axes.
- 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.
- In the *Segment* box set *Radius* to 2.5 m. The generated segment is shown in Figure 6.3.

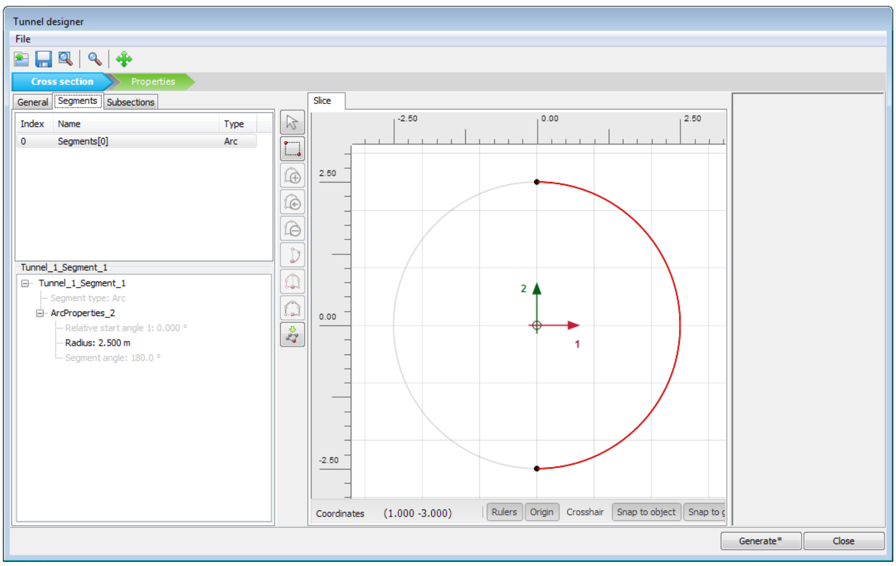

Figure 6.3 The geometry of the tunnel segment

- Click the *Properties* tab to proceed to the corresponding tabsheet.
- Right-click the segment in the display area and select the *Create plate* option in the appearing menu.
- Create a new material dataset. Specify the material parameters for lining according to Table 6.2.

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

Table 6.2 Material properties of the plates

Parameter	Name	Lining	Building	Unit
Material type	*Type*	Elastic; Isotropic	Elastic; Isotropic	-
Normal stiffness	*EA*	$1.4 \cdot 10^7$	$1 \cdot 10^{10}$	kN/m
Flexural rigidity	*EI*	$1.43 \cdot 10^5$	$1 \cdot 10^{10}$	kNm^2/m
Weight	*w*	8.4	25	kN/m/m
Poisson's ratio	ν	0.15	0.0	-

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.

- Right-click the segment in the display area and select the *Create negative interface* option in the appearing menu.
- 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 Figure 6.4.

Hint: 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.

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

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

Draw a plate from (5.0 3.0) to (15.0 3.0), representing the building.

Create a material set for the building according to Table 6.2 and assign it to the plate.

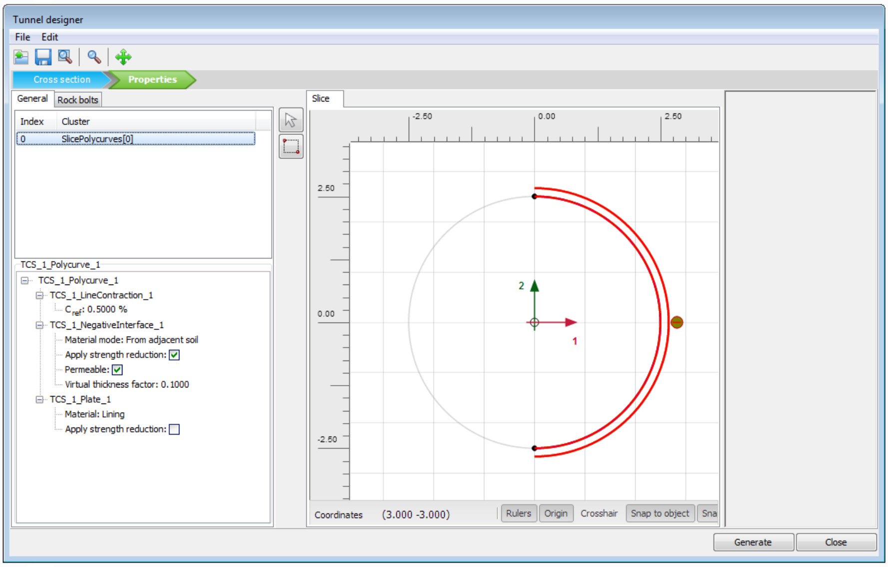

Figure 6.4 Tunnel model in the *Properties* mode

Draw two piles (embedded beam rows) from (5.0 3.0) to (5.0 -11.0) and from (15.0 3.0) to (15.0 -11.0).

Create a material set for the foundation piles according to Table 6.3 and assign it to the foundation piles.

Table 6.3 Material properties of the piles

Parameter	Name	Foundation piles	Unit
Stiffness	E	$1.0 \cdot 10^7$	kN/m^2
Unit weight	γ	24.0	kN/m^3
Diameter	D	0.25	m
Pile spacing	$L_{spacing}$	3.0	m
Skin resistance	$T_{skin,start,max}$	1.0	kN/m
	$T_{skin,end,max}$	100.0	kN/m
Base resistance	F_{max}	100.0	kN
Interface stiffness factors	-	Default	-

Hint: 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.

6.2 MESH GENERATION

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.

- Proceed to the *Mesh* mode.

Create the mesh. Use the default option for the *Element distribution* parameter (*Medium*).

View the mesh.

- In the Output program click on the *Fixities* option in the *Geometry* menu to display them in the model. The generated mesh is shown in (Figure 6.5).
- Click on the *Close* tab to close the Output program.

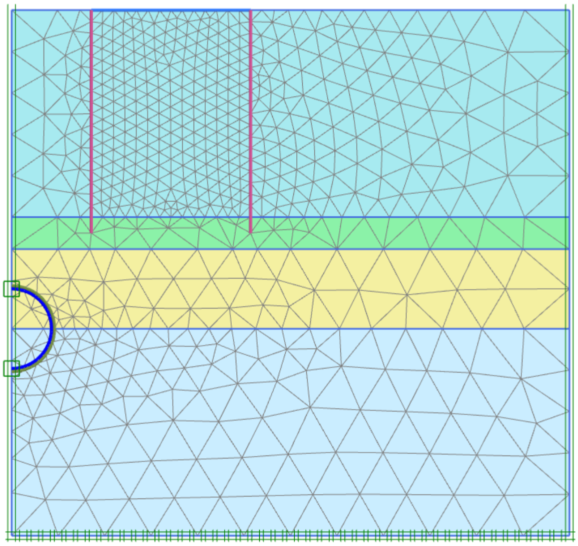

Figure 6.5 The generated mesh

6.3 CALCULATIONS

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

- Click on the *Staged construction* tab to proceed with the definition of the calculation phases.
- 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.

Phase 1: Building

The first calculation phase is used to activate the building.

In the *Phases explorer* click the *Add phase* button to introduce a new phase.

- In the *Phases* window rename the phase as 'Building'.
- 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.

- In the drawing area activate the plate and the foundation piles.

Simulation of 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. Deactivating the soil inside the tunnel only affects the soil stiffness and strength and the effective stresses. Without additional input the water pressures remain. To remove the water pressure inside the tunnel the *Water conditions* of the two soil clusters in the tunnels must be set to *Dry* in the *Object explorers*. The calculation phases are *Plastic* analyses, *Staged construction*. To create the input, follow these steps.

Phase 2: Tunnel

Add a new calculation phase.

- In the *Phases* window select the *Reset displacements to zero* option in the *Deformation control parameters* subtree.
- 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*.
- Activate the tunnel lining and the negative interfaces. Note that contraction is not active in this phase.

Phase 3: Contraction

In addition to the installation of the tunnel lining, the excavation of the soil and the dewatering of the tunnel, the volume loss is simulated by applying a contraction to the tunnel lining. This contraction will be defined in a staged construction calculation phase:

Add a new calculation phase.

- Multi-select the plates. In the *Selection explorer* activate the contraction.

Hint: For a more realistic model, different properties should be defined for the lining in this phase and in the final one.

» 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.

Phase 4: 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.

Add a new calculation phase.

- In the *Phases* window do NOT select the *Reset displacements to zero* option in the *Deformation control parameters* subtree.

- In the *Staged construction* mode deactivate the tunnel lining (plates, negative interfaces and contraction).
- Multi-select the clusters inside the tunnel. In the *Selection explorer* activate *WaterConditions*.
- 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.

Phase 5: Final lining

Add a new calculation phase.

- In the *Phases* window, do NOT select the *Reset displacements to zero* option in the *Parameters* subtree.
- In the *Staged construction* set the clusters inside the tunnel to *Dry*.
- Activate the tunnel lining (plates) and the negative interfaces in the tunnel.

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).

Calculate the project.

Save the project after the calculation has finished.

6.4 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 6.6).

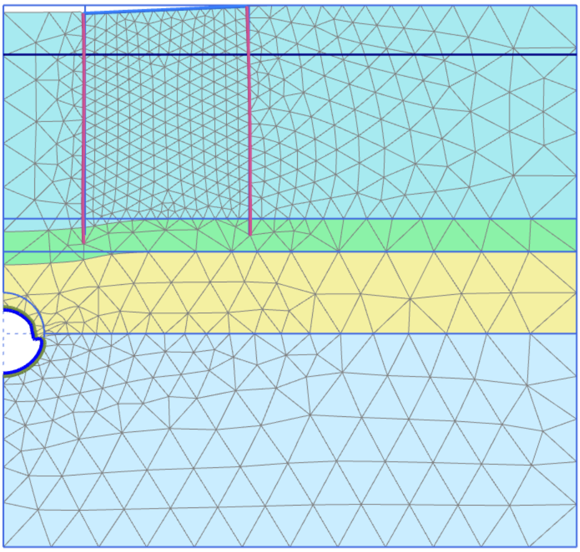

Figure 6.6 Deformed mesh after construction of the tunnel (Phase 5; scaled up 20.0 times)

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 6.7).

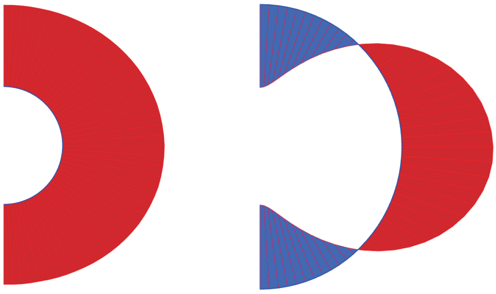

Figure 6.7 Axial forces and Bending moments in the lining after the second phase

The plot of effective stresses, Figure 6.8, 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.

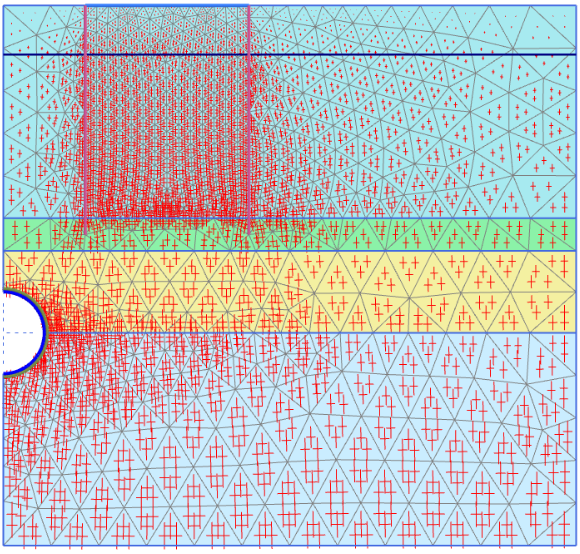

Figure 6.8 Effective principal stresses after the construction of the tunnel

To display the tilt of the structure:

 Click the *Distance measurement* in the side toolbar.

- Click the node located at the left corner of the structure (5.0 3.0).
- Click the node located at the right corner of the structure (15.0 3.0). The *Distance measurement information* window is displayed where the resulting tilt of the structure is given (Figure 6.9).

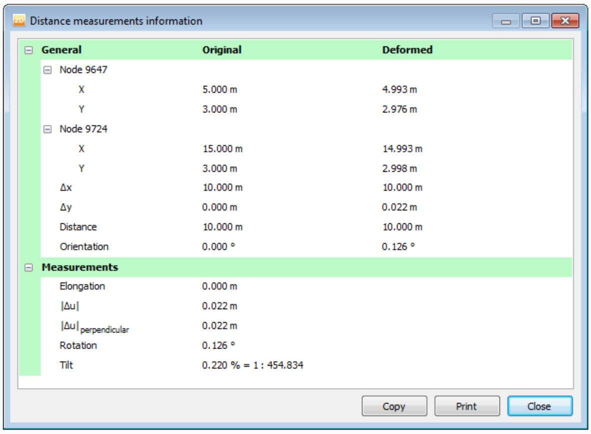

Figure 6.9 *Distance measurement information* window