



Tutorial 2g.04

Construction of a bolted tunnel

Ref: CESAR-TUT(2g-04)-v2021.0.1-EN

1. PREVIEW

Tunnelling consists in a series of construction processes through variable soil conditions. For the tunnel engineer it is important to model accurately this construction process as it can affect dramatically the project environment (settlements at the surface for example) or the tunnel structure (stress in the lining or shotcrete for example).

CESAR-LCPC provides a complete set of tools for this kind of analysis.

The aim of this tutorial is to determine the stress field in a concrete tunnel lining at service time. Based on the works of the Tartaiguille tunnel (Charmetton, 2001), it has been simplified to fit tutorial objectives. Nevertheless, it will guide the user through the various aspects of modelling the tunnel construction.

General assumptions:

- Excavation is uniform and permanent along horizontal axis Oz. The tunnel section is symmetrical in regard of y- axis. Therefore, only one half-section of the tunnel in the plane Oxy will represent the problem in plane strain analysis conditions;
- Non-linear behaviour of the soil;
- The lining is modelled with volume elements. The concrete is assumed to behave linear elastically.

Geometry definition:

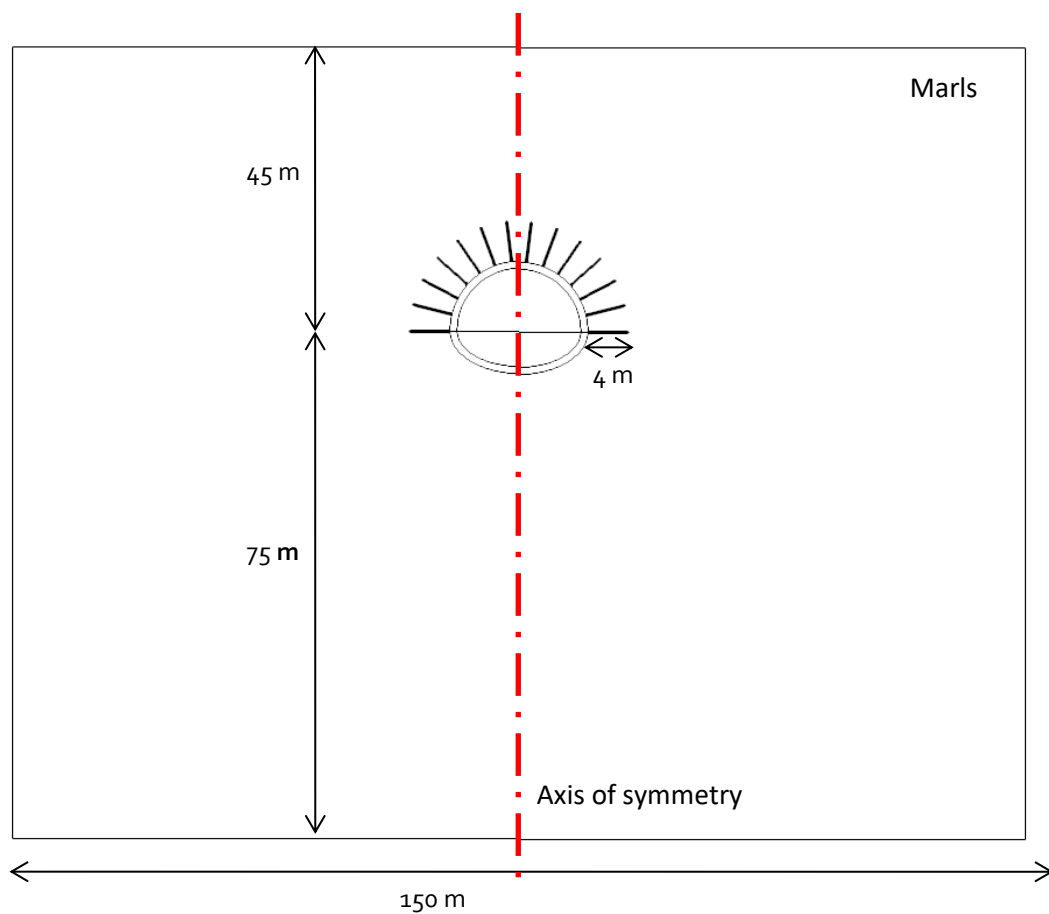


Figure 1: Geometrical description of the project

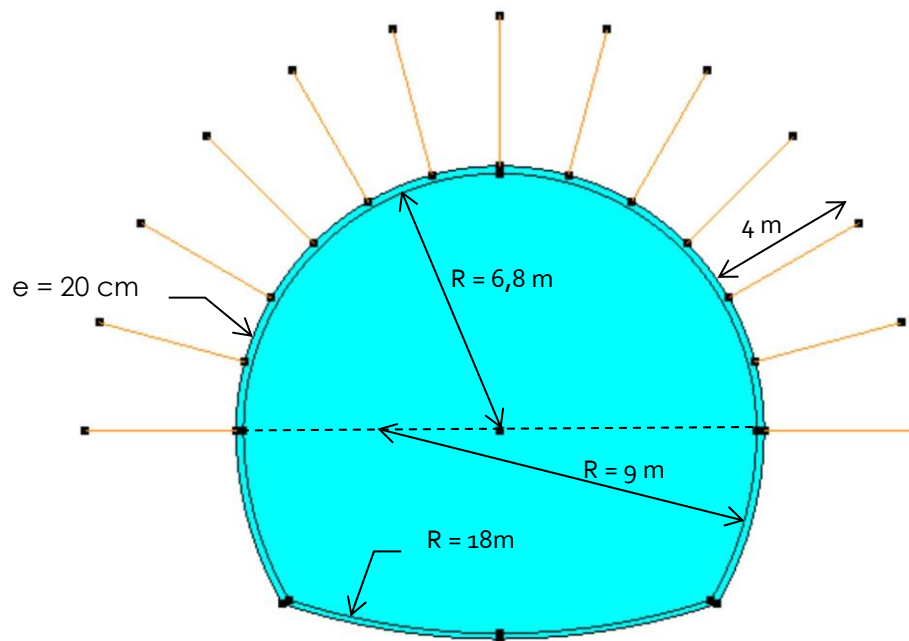


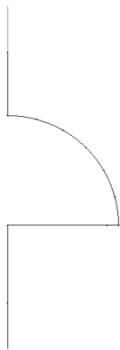
Figure 2: Description of the tunnel section

Definition of the stages:

Stage #1: Initial stress field generated under self-weight of the soil layers.

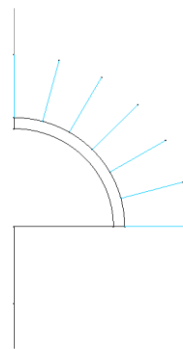
Stage #2:

Excavation of the top heading.



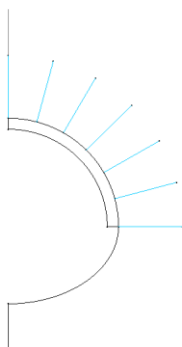
Stage #3:

Installation of the bolts and the lining in the upper section. Progression of the tunnel face resulting in total relaxation of soil.



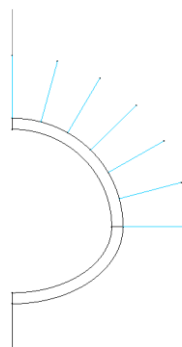
Stage #4:

Excavation of the bench.



Stage #5:

Installation of the lining in the lower section. Total loss of confinement.

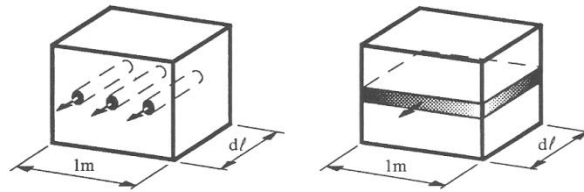


Material Properties:

The soil is assumed homogenous with Marls properties.

	γ_h (kN/m ³)	E (MPa)	ν	c' (kPa)	φ' (°)	ψ' (°)
Marls	23	1 500	0,3	300	20	0
Shotcrete	25	7 000	0,2	-	-	-

The bolts are bars with round section ($E_s = 21$ GPa; $\varnothing_s = 2,5$ cm). As we are modelling in 2-dimensions with the hypothesis of plane strain deformations, the Young's modulus and the section of the bolts have to be consistent in order to fulfil equivalence of traction-compression behaviour in 3D.



Considering that a crown of bolts is bored every 2m of the excavation length, we have following equation:

$$E_{eq} \times S_{eq} = \frac{E_s \times S_s}{2}$$

Thus, considering the same E, we have: $S_{eq} = \frac{S_s}{2}$

The values for the model are listed in the following table.

	E_{eq} (MPa)	S_{eq} (m ²)
Bolt (equivalent)	210 000	$2,4 \cdot 10^{-4}$

Bodies definition and status:

It is important to mark the different bodies (with a name) and to determine their status through the construction stages. This helps for the input of properties and management of the operations.



Figure 3: List of bodies

Bodies	Stage #1	Stage #2	Stage #3	Stage #4	Stage #5
1	Marls	→	→	→	→
2	Marls	Inactive	→	→	→
3	Marls	Inactive	Concrete	→	→
4	Marls	→	→	Inactive	→
5	Marls	→	→	Inactive	Concrete
6	Inactive	→	Bolts	→	→

Boundary conditions:

As there is a vertical symmetry axis, the horizontal displacements are fixed along vertical limits, at $x=0$ and $x=75\text{m}$.



At the bottom of the model vertical displacements are fixed.

Meshing:

The mesh is refined close to the tunnel, where high strain gradients will occur, and coarsened away from it.

2. 2D MODELLING

2.1. General settings




1. Run CLEO2D.
2. Set the units in the menu  **Units**.
 - In the tree, select the leaf **General/Length** and set the unit **m** in the bottom left combo box.
 - In the tree, select the leaf **Mechanic/Force** and set the unit **kN** in the bottom left combo box.
 - In the tree, select the leaf **Mechanic/Displacement** and set the unit **mm**.
 - In the tree, select the leaf **Mechanical/Surface** and set the unit **m²**. Activate scientific notation (0.000E+000).
 - Click on **Apply** to close.
3. In  **Working plane**, set the visible grid to 5 m (dX = dY = 5m).



Use "Save as default" to set this system of units as your user environment.

2.2. Drawing of the geometry

CESAR is equipped with a tool for the design of tunnel sections.

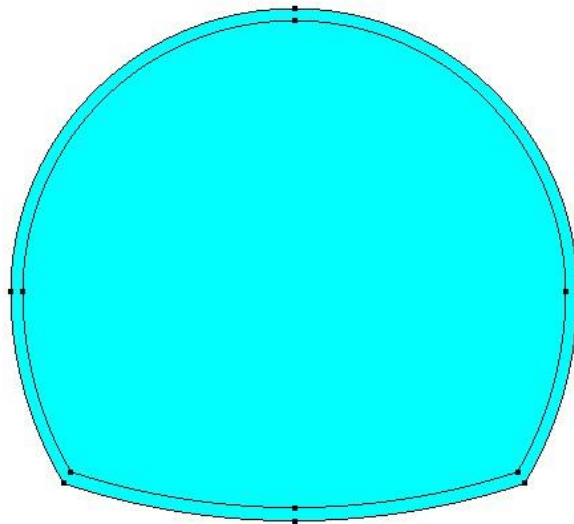
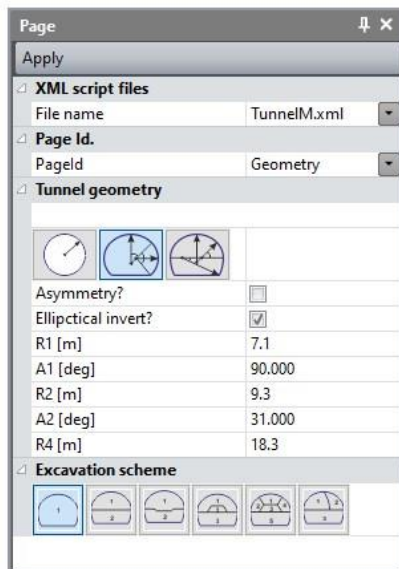
1. Activate the tool.  **Script**.
2. Select *TunnelM.xml* as **XML script file**.
3. As **tunnel geometry**, select  to define the tunnel section with 3 radius and 2 angles. Then, select the **excavation scheme**  (full section).

- Asymmetry?	No
- Elliptical invert?	Yes
- R1 [m]	6,8
- A1 [°]	90
- R2 [m]	9
- A2 [°]	30
- R4 [m]	18




Apply
4. Repeat the previous operations with modified radius and angles values, so that we create the lining thickness.

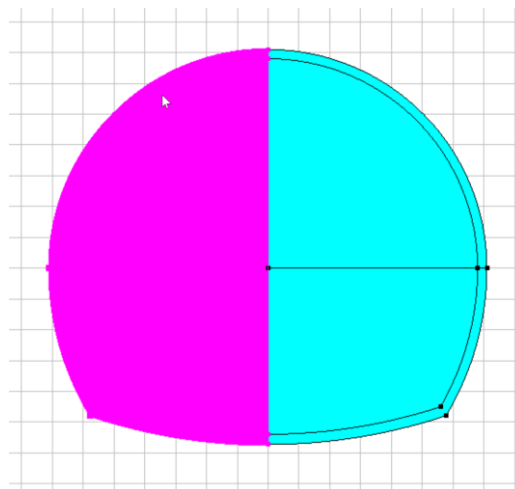
- Asymmetry?	No
- Elliptical invert?	Yes
- R1 [m]	6,8
- A1 [°]	90
- R2 [m]	9
- A2 [°]	30
- R4 [m]	18

Apply





Considering symmetries of the problem, we are modelling only one half of it in order to spare nodes and associated calculation times.

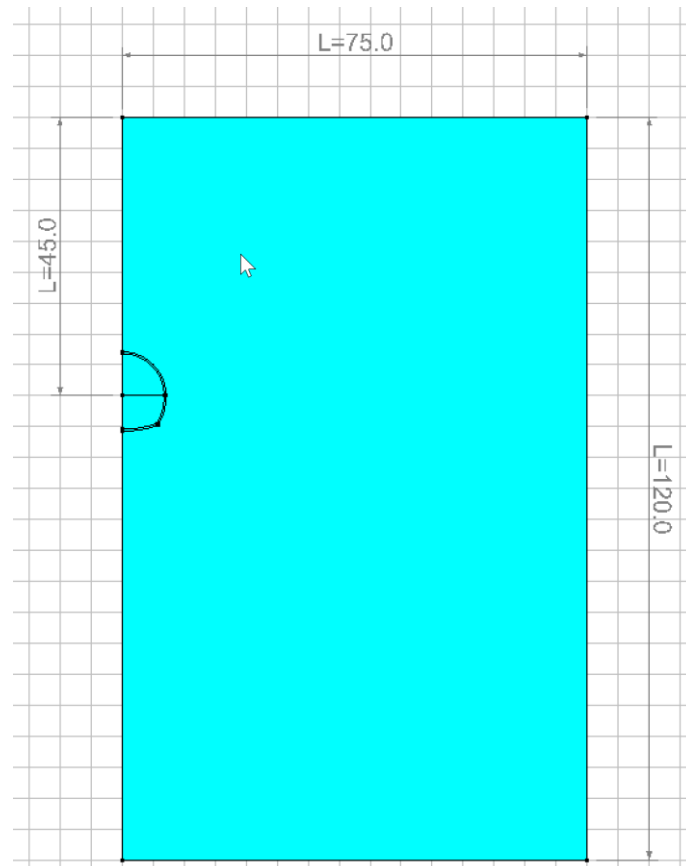
Using the **Lines**  tool, we add vertical segments from bottom to top and a horizontal segment at the half of it. Then, using selection tools  and tool  **Delete**, we can delete a half of the previous geometry (see figure below).



Drawing of the soil mass:

We can now complete the model with the soil mass.


Use the **Points**  and/or the **Lines**  feature to draw the geometry with help of the grid. We set the top of soil mass at $y = 45$ m, the bottom at $y = -75$ m and the lateral extension is at $x = 75$ m.

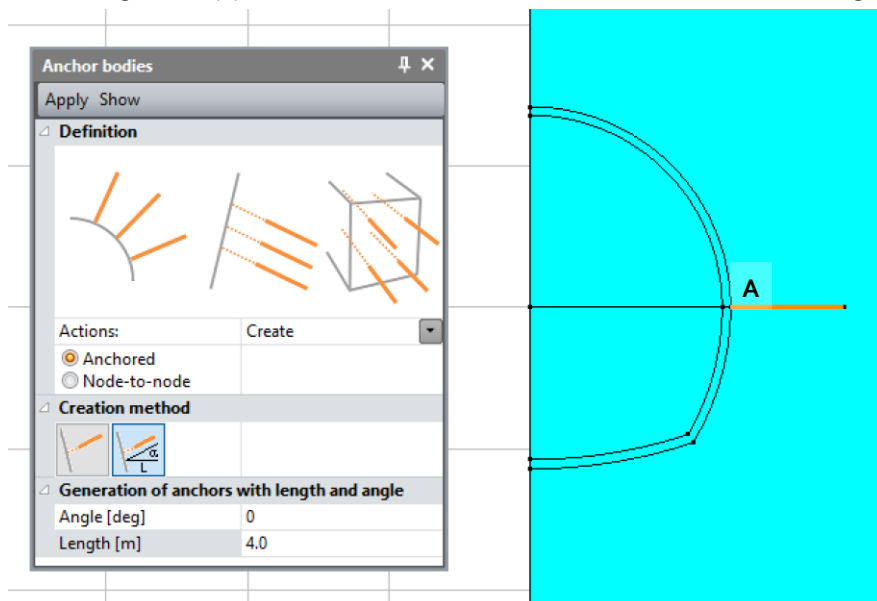



Soil mass after deletion of the half tunnel section

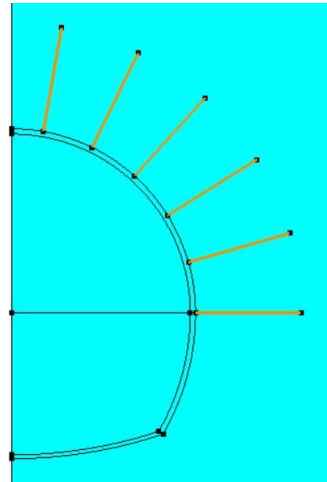
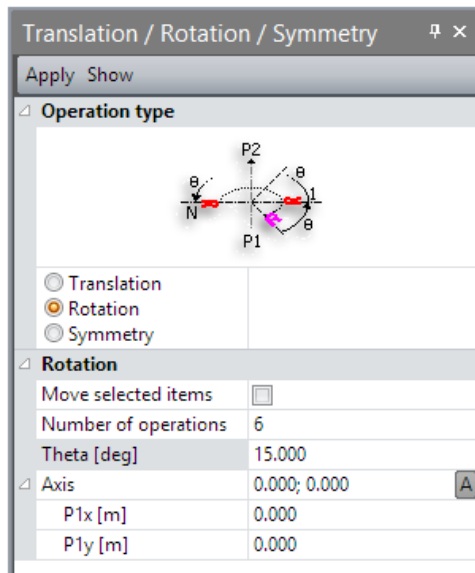
Drawing of the bolts:

We can now introduce the bolts as anchored 1D-bodies. We will start with the generation of one anchored 1D-body then copy it to create the crown.

1. Activate the tool.  **Anchor bodies.**
2. Select "Method 2" as generation method.
3. Set the length, 4m, and the angle, 0° , values of the anchored 1D-body to be created. We chose here to generate first the horizontal bolt.
4. Click on the geometry point A where the anchored element is connected. See figure below.




5. Activate the tool.  **Operations on bodies.**
6. Select *Rotation* as operation type and set the parameters of the rotation:
 - Move selected items = Inactivate
 - Number of operations = 5
 - Theta = 16°
 - Center = (0: 0). User may use the "A" function to define the center of rotation.
7. Select the anchored 1D-body already defined and **Apply**.



Bodies edition:

This step is optional but it eases the identification and selection of the group of bodies.

1. On the **Selection Toolbar**, activate **Select bodies**.
2. Activate  **Body properties**.
3. Right-click on the body corresponding to the Soil layer. Enter **Soil** as name. Specify a colour. Click on **Apply**.
4. Right-click on the body corresponding to the upper lining. Enter *Upper lining* as name. Specify a colour. Click on **Apply**.
5. Right-click on the body corresponding to the lower lining. Enter *Lower lining* as name. Specify a colour. Click on **Apply**.
6. Right-click on the body corresponding to the upper section. Enter *Top heading* as name. Specify a colour. Click on **Apply**.
7. Right-click on the body corresponding to the lower section. Enter *Bench* as name. Specify a colour. Click on **Apply**.



We can also merge the bolts. In GEOMETRY tab, use  **Merge bodies**.




8. Select the 1D-bodies corresponding to the bolts. Enter *Bolts* as a name. Specify a colour. Click on **Apply**.

2.3. Mesh

2.3.1 Density definition



Define dense divisions in the area of high strains, i.e. nearby the tunnel excavation. Use a progressive density definition to generate a progressive evolution from dense mesh to coarse mesh on the boundary edges.

1. Go to the tab **MESH** on the project flow bar to start the definition of divisions along lines.
2. Select the contour of the tunnel sections and bolts. Click on  **Fixed length density** to impose a length for the divisions of this segment. Enter **0,5 m** in the dialog box. Click on **Apply**.
3. Click on  **Variable density** to divide the segment with a variation of lengths. Tick **First division** and **Last division** to define the method.
 - Enter **0,4 m** as **First division** and **2 m** as **Last division**. Click on segment A.
 - Enter **2 m** as **First division** and **10 m** as **Last division**. Click on segment B.
 - Enter **0,4 m** as **First division** and **10 m** as **Last division**. Click on segment C.
4. Click on  **Fixed length density** to impose a length for the divisions of the last 2 segments on the basement and vertical limit of the model. Enter **10 m** in the dialog box. Select the external edges. Click on **Apply**.



The position of the click defines where the initial division is.

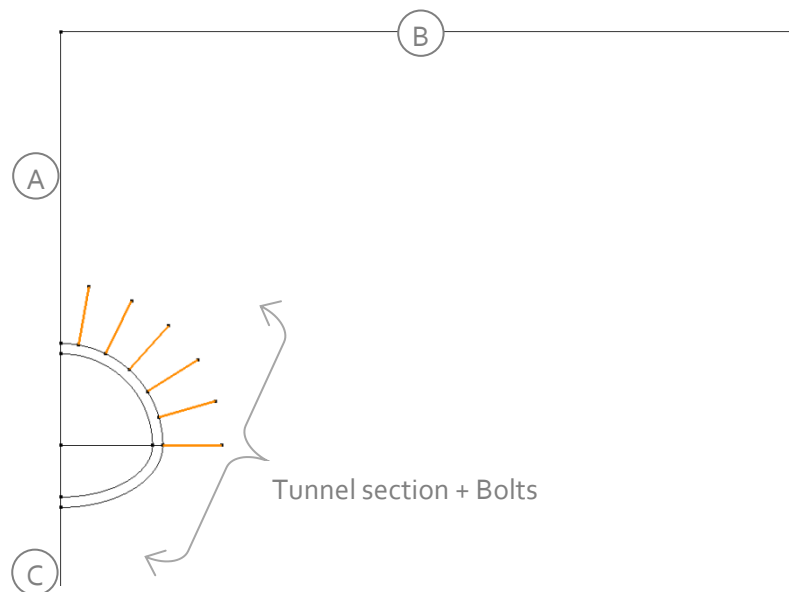


Figure 4: Numbering of the segments



CESAR-LCPC proposes a post-processing tool for integration of the stresses inside surface elements; results are common structural efforts (M, N, V). The stress integration gives better results when these elements are quadrangle. Thus, user will generate appropriate geometry (quadrangle surface) to generate this type of mesh.



2.3.2 Meshing

3 levels for the surface meshing procedure are proposed, giving the possibility to adjust the meshing process from coarser to denser. Default level is "quadratic", user may adjust it.

1. Go in **Preferences > Program settings**,
2. In **Mesh creating function**, modify the algorithm to cubic (linear = coarse, cubic = dense).

1D-bodies meshing:

We start with the mesh of the 1D-bodies. As they are anchored elements, they need to be meshed before the soil mass.

1. In the selection toolbar, deactivate selection of surface bodies and activate 1D-bodies.
2. Select the 1D-bodies corresponding to the bolts.
3. Click on  **1D-body meshing**.
 - Chose **Quadratic interpolation** as interpolation type.
 - Click on **Apply**.
4. If necessary, orientate the 1D bodies in the same direction using  **1D-body orientation**.



Depending of the initial settings, the user may want to modify the size of the arrows, orientation of the 1D bodies. This is set in the menu **Preferences > Properties display**:

- Set the vector scale to 0.02 ;
- Set the arrow scale to 0.003.

Surface bodies meshing:

1. Select all the surface bodies.
2. Click on the **Surface meshing** tool . Chose **Quadratic** as interpolation type. Chose **Triangle** as element shape.
3. Click on **Apply** to generate the mesh.

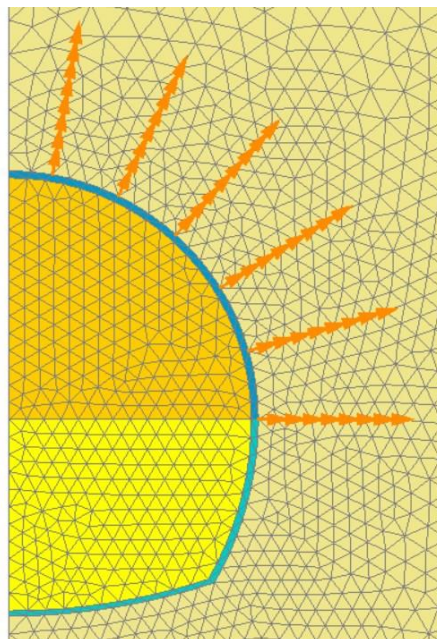


Figure 5: Zoom on the mesh of the tunnel section and bolts

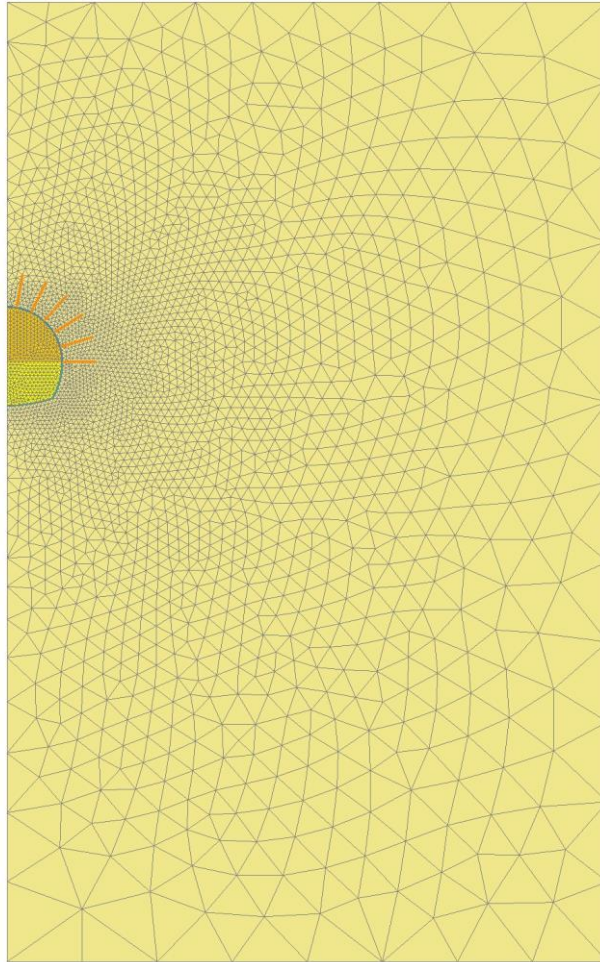


Figure 6: View of the complete mesh

3. CALCULATION SETTINGS

3.1. Edition of stage #1

The staged construction process requires the definition of an initial stress field before applying loads. For this project, we follow the geostatic stresses procedure where the initial stress field is initialized with the "Ko procedure" (see reference manual).

Model definition:

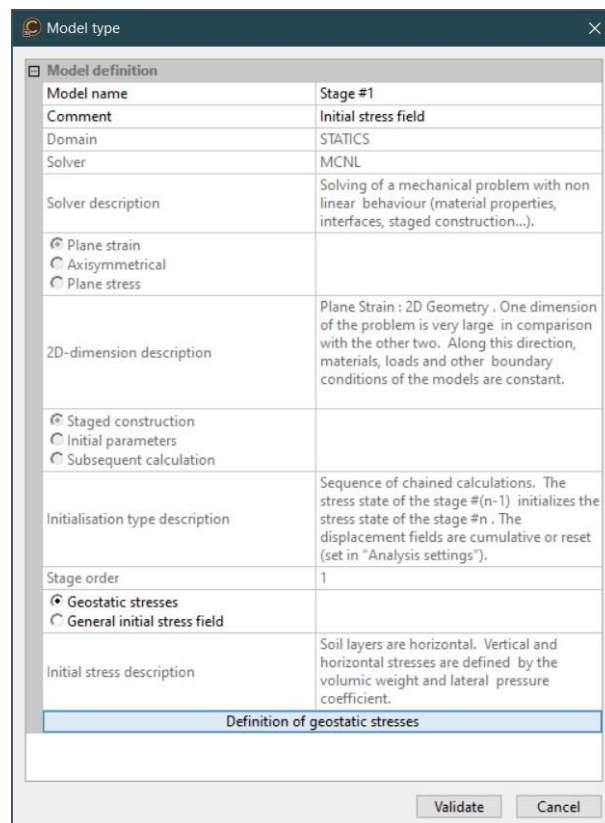
1. On the right side of the working window, the "Tree view" window displays the list of physical domains. Right click on **STATICS**. Click on **Add a model**. A new toolbox is open for definition of the Model.
2. Enter **Stage #1** as "Model name".
3. Select **MCNL** as "Solver".
4. Tick **Plane strain** as model configuration, with **Staged construction**.
5. Tick **Geostatic stresses** as initialization type. Click on **Definition of geostatic stresses**.

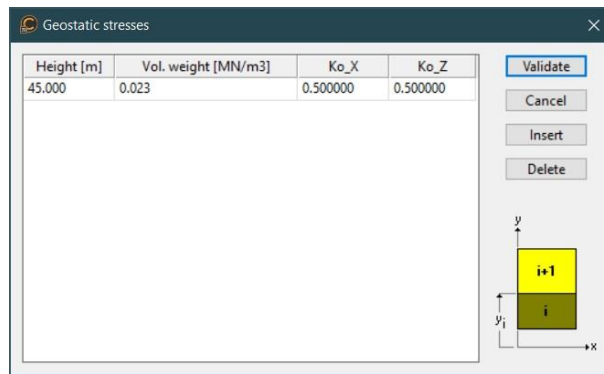
- Click on **Insert** to define 1 new layer.
- Enter the following values:

Height (m)	Unit weight (MN/m ³)	Ko_x (-)	Ko_z (-)
45	0.023	0.5	0.5

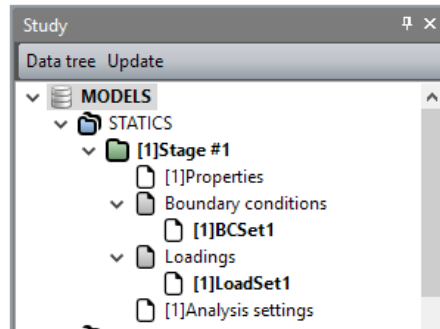
- Click on **Validate**.

6. Click on **Validate**.







The date tree is now as illustrated below.

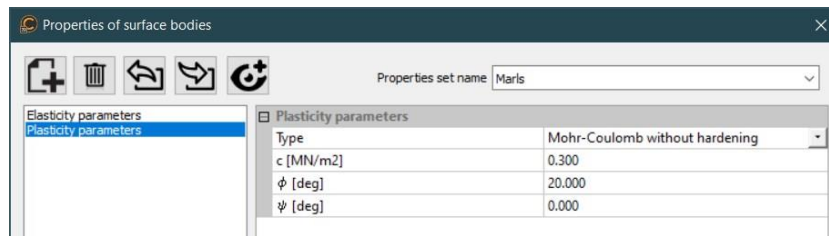


Material properties for the solid bodies:


We initially define the material library of the study in the step **PROPERTIES**.

1. Click on  **Properties for volume bodies**.
2. Give a name for the properties set name (*Concrete* for example).
 - In **Elasticity parameters**, chose "Isotropic linear elasticity" and define ρ , E and ν .
 - Click on **Validate**.
3. Click on  to create another properties set.
 - Modify the name for the properties set (*Marls* for example).
 - In **Elasticity parameters**, chose "Isotropic linear elasticity" and set values for ρ , E and ν .
 - In **Plasticity parameters**, chose "Mohr-Coulomb without hardening" and set values for c , ϕ and ψ .
 - Click on **Validate**
4. **Close**.

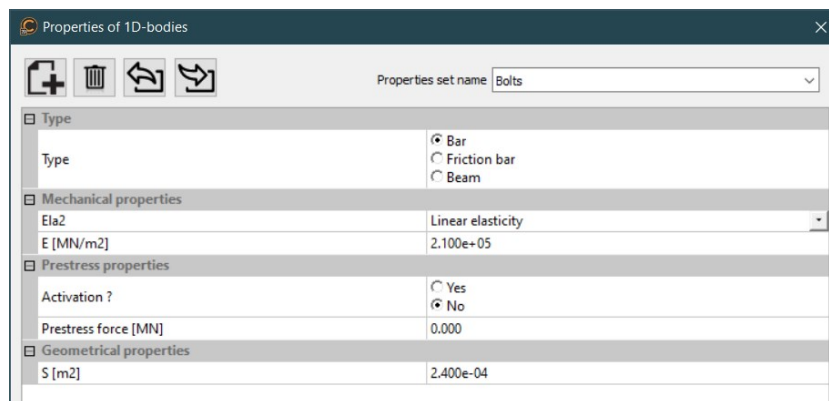
	ρ (kg/m ³)	E (MN/m ²)	ν	c (MN/m ²)	ϕ (°)	ψ (°)
<i>Concrete</i>	2500	7 000	0,2	-	-	-
<i>Marls</i>	2300	1 500	0,3	0.02	30	10



Material properties for the bolts:




1. Click on  **Properties for 1D bodies**.
 - Give a name for the properties set name (*Bolts* for example).
 - As **Type**, chose "Bar" and define the rigidity *E*, and the section *S*.
 - Click on **Validate**.
2. **Close**.

Name of the body	Type	E (MN/m ²)	S (m ²)
<i>Bolts</i>	Bar	210 000	2,4.10 ⁻⁴




Assignment of data sets:

As data sets are created, we affect them to the bodies of the model.

1. Click on  **Assign properties** tool.
2. On the left side, a new toolbox is displayed. Click on  **Properties for 2D bodies**.
 - Select all the surface bodies on the model window and the set of parameter *Marls* in the list.
 - **Apply**.
3. On the left side, click on  **Properties for 1D bodies**.
 - Select all the 1D bodies (anchored bolts) of the model and the set of parameter *Bolts* in the list.
 - **Apply**.

Active/inactive bodies

In stage #1, all groups have Soil properties. Bolts are not activated.

1. Select the *Bolts* bodies.
2. Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**. The 1D bodies are now marked with the neutral colour.

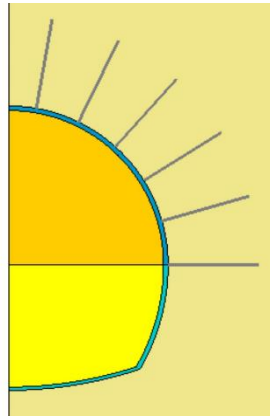



Figure 7: Status of the model when bodies are set inactive at stage #1

Boundary conditions:

1. Activate the **BOUNDARY CONDITIONS** tab.
2. On the toolbar, activate  to define side and bottom supports.
3. **Apply**. Supports are automatically affected to the limits of the mesh.




Default name of the boundary condition, BCSet1, set can be edited using the [F2] key. For explicit identification, we rename it to *Standard fixities*.

Loading set:

No load is defined here as we are defining the initial stress field as geostatic stresses and we want to visualize it separately from other tunnel construction stages.

Calculation parameters:

1. In **ANALYSIS**, activate **Analysis settings** .
2. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 1
 - Max number of iterations per increment: 500
 - Tolerance: 0,01
 - Method of resolution: 1- initial stresses
 - Solver type: Pardiso
3. Click on **Validate**.

3.2. Edition of stage #2

In this stage, we excavate the upper section. To unbalance the model, excavation forces on the section limits are applied.

When creating a new analysis phase, it is handy to use parameters of the previous one by copy of the model. Subsequently, we make modifications where necessary:

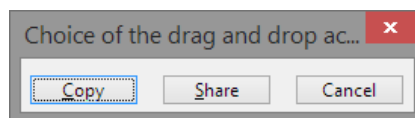
- In properties as bodies activity is changed,
- In loads in order to activate the excavation forces.

Model definition:

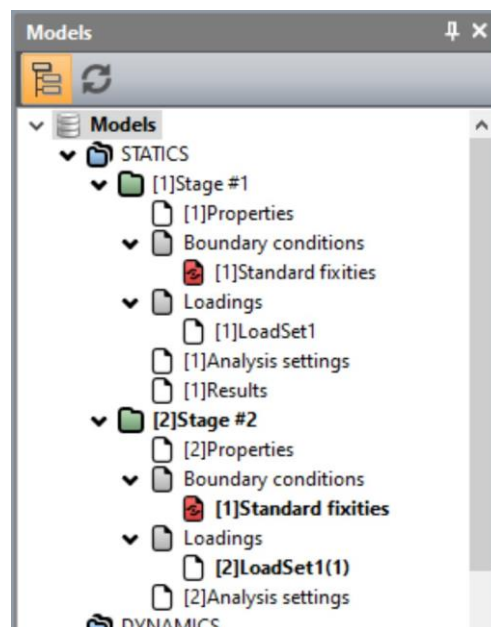
1. In the "Tree view", right click on **STATICS**. Click on **Add a model**. A new toolbox is open for definition of the Model.
2. Enter *Stage #2* as "Model name".
3. Select **Staged analysis** as initialization type.
4. Click on **Validate**.

We can now copy the definitions sets of the model from previous model.

1. Select the *Properties* set of the model *Stage #1*.
2. By drag and drop, place it on the new *Properties* set of model *Stage #2*.
3. A dialog box is proposing to copy or share these parameters. We chose here to **Copy**.
4. Repeat the operations 1 to 3 for the *Boundary conditions* set. Chose **Share**.




The date tree is now as illustrated below.



Active/inactive bodies

The excavation of the upper section leads to set at inactive state the groups **Upper section** and **Upper lining**.

1. Select the surface bodies *Upper section* and *Upper lining* and the 1D bodies.
2. In the step **PROPERTIES**, click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**. The elements are now marked with the neutral colour.

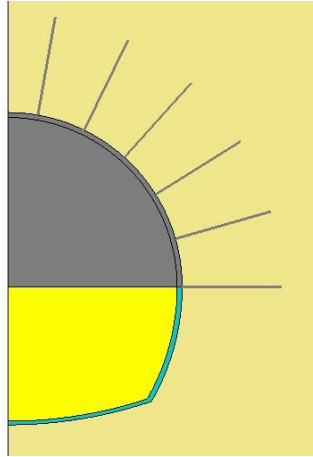



Figure 8: Status of the model when bodies are set inactive at stage #2

Boundary conditions:

There is no change in the boundary conditions.

Loading set:

The tunnel face is not far enough to consider that the soil is completely relaxed. Only 40% of the excavation forces are applied ($\lambda = 0,4$)

1. Activate the **LOADS** tab.
2. In the model tree, select load set *LoadSet1 (1)* Rename it to *Excavation forces* using the [F2] key.
3. On the toolbar, activate  **Excavation forces**.
 - Tick "Active"
 - Set the value of λ to 0.4,
 - As stress origin, select *Stage #1* as stress field origin.
4. Click on **Apply**. The excavation forces are applied.

Analysis settings:

There is no change in the analysis settings.

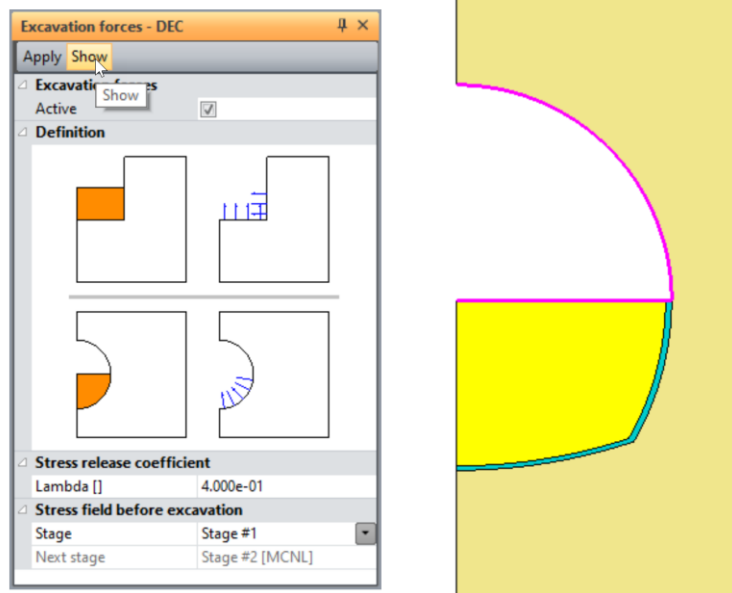


Figure 9: Display of the excavation forces

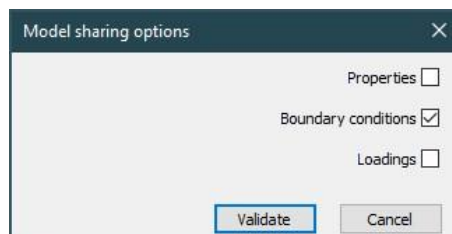
3.3. Edition of stage #3

In this stage, we activate the upper lining and the bolts. The tunnels face progress far enough to consider that the soil is completely relaxed.


Model definition:

As most of the parameters of *Stage #3* are common with *Stage #2*, we make a copy of them and modify where necessary.

1. In the STUDY tree, right click on the model *Stage #2*. Click on **Copy of the model**. A new toolbox is open for definition of the Model.
 - Enter *Stage #3* as "Model name".
 - Select "Staged analysis" as initialization type.
 - Click on **Validate**.
2. A toolbox is displayed. User can share sets of properties, boundary conditions or loads, or just copy them. We chose here to share boundary conditions.



Active/inactive elements

1. Select the bodies *Upper lining* and *Bolts*.
2. In the step **PROPERTIES**, click on  **Activate/deactivate bodies**.
3. In the grid, tick "Active".
4. **Apply**. The elements are now marked with their original colour.

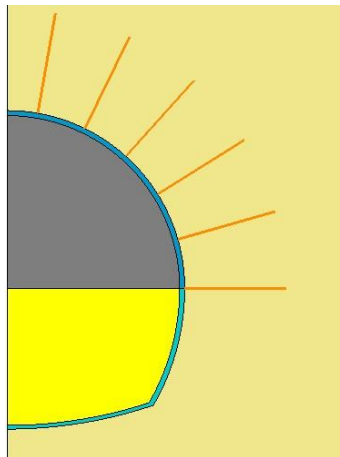




Figure 10: Status of the model when bodies are set active at stage #3

Assignment of material properties:


1. Click on  **Assign properties** tool.
2. On the left side, a new window is displayed. Click on  **Properties for 2D elements**.
3. Select the surface body of the *Upper lining* on the model window and the set of parameter "Concrete" in the list.
4. **Apply**.

Boundary conditions:


There is no change in the boundary conditions. As we activate the upper lining, the existing fixities are adapted on its limit.

Loading set:

The excavation forces are modified as the tunnel face has progressed. The soil stresses are totally released on the limits of the excavation: we apply a lambda coefficient of 0,6 so that 100% of the excavation forces.

1. Activate the **LOADS** tab.
2. In the model tree, select load set *LoadSet1 (1)* Rename it to *Excavation forces* using the [F2] key.
3. On the toolbar, activate  **Excavation forces**.
 - Tick "Active"
 - Set the value of lambda to 0,6,
 - As stress origin, select *Stage #1* as stress field origin.
4. Click on **Apply**. The excavation forces are applied.

The gravity forces are applied to the lining.

1. In the model tree, right click on *Loadings of Stage #3*. Click **Add loading set**.
2. Enter *Self-weight of upper lining* as a name.
3. Click **Validate**.
4. Go to the tab **LOADS**. On the toolbar, activate **Gravity forces**  .
 - Click on **Automatic selection**. This feature detects the groups activated in the actual stage, i.e. the *Upper lining* body.
 - **Apply**. On the bottom left of the working space, "Gravity forces" is displayed.

Calculation parameters:

No modifications from previous stage #3.

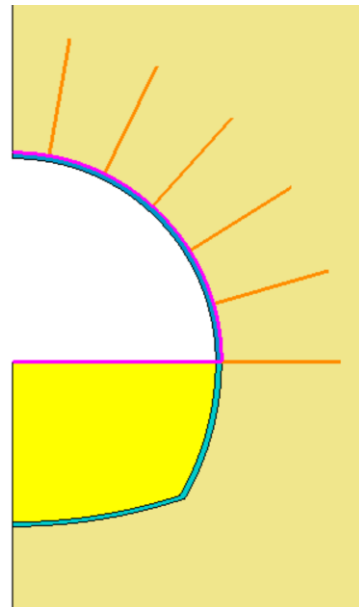
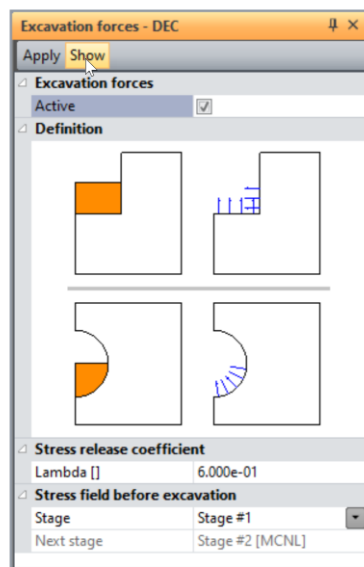


Figure 11: Display of the excavation forces at stage #3

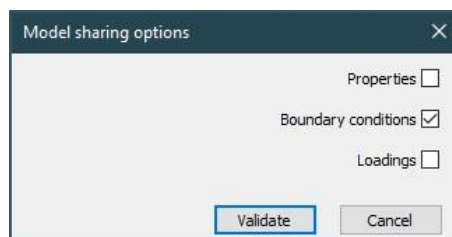
3.4. Edition of stage #4

In this stage, we excavate the lower section.

Model definition:


As most of the parameters of *Stage #4* are common with *Stage #3*, we make a copy of them and modify where necessary.

1. In the STUDY tree, right click on the model *Stage #3*. Click on **Copy of the model**. A new toolbox is open for definition of the Model.
2. Enter *Stage #4* as "Model name".
3. Select **Staged analysis** as initialization type.
4. Click on **Validate**.
5. A toolbox is displayed. User can share sets of properties, boundary conditions or loads, or just copy them. We chose here to share boundary conditions.



Active/inactive elements

The excavation of the lower section leads to set at inactive state the groups **Lower section** and **Lower lining**.

1. Select the surface bodies : *Bench* and *Lower lining*.
2. Click on  **Activate/deactivate bodies**.
3. In the grid, tick "Inactive".
4. **Apply**. The elements are now marked with the neutral colour.

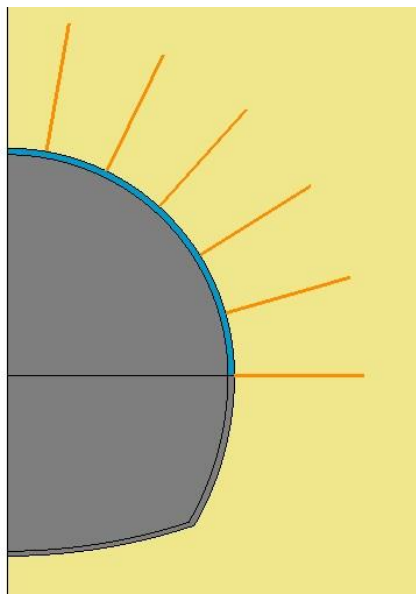



Figure 12: Status of the model when bodies are set inactive at stage #4

Boundary conditions:

There is no change in the boundary conditions.

Load case:

Here, we first generate the new loading set for the lower excavation and then delete the copied ones from *Stage #3*, useless in this *Stage #4*.

1. In the model tree, right click on *Loadings*. Click **Add loading set**.
2. Enter *Excavation of bench* as a name. **Validate**.
3. Activate the **LOADS** tab.
4. On the toolbar, activate  **Excavation forces**.
 - Tick "Active"
 - Set the value of lambda to 0.4,
 - As stress origin, select *Stage #3* as new stress field origin.
5. Click on **Apply**. The excavation forces are applied.
6. Delete the loading sets copied from previous *Stage#3* (*Excavation forces* and *Self-weight of upper lining*). Right click on them and click on **Delete loading set**.

Calculation parameters:

No modifications from previous stage #3.

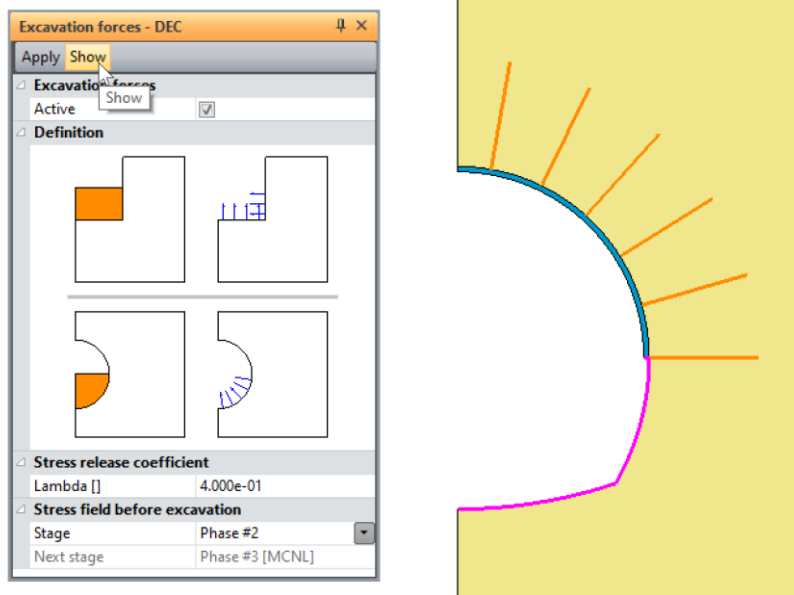


Figure 13: Display of the excavation forces at stage #4

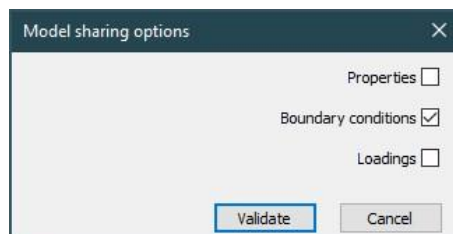
3.5. Edition of stage #5

In this stage, we activate the lower lining. The tunnels face progress far enough to consider that the soil is completely relaxed.


Model definition:

As most of the parameters of *Stage #5* are common with *Stage #4*, we make a copy of them and modify where necessary.



1. In the STUDY tree, right click on the model *Stage #4*. Click on **Copy of the model**. A new toolbox is open for definition of the Model.
2. Enter *Stage #5* as model name.
3. Select **Staged analysis** as initialization type.
4. Click on **Validate**.
5. A toolbox is displayed. User can share sets of properties, boundary conditions or loads, or just copy them. We chose here to share boundary conditions.

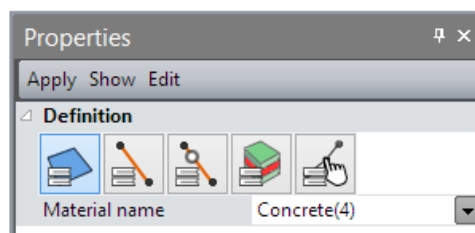


Active/inactive elements

1. Select the body *Lower lining*.
2. Click on  **Activate/deactivate bodies**.
3. In the grid, tick "Active".
4. **Apply**. The body is now marked with its original colour.

Assignment of material properties:

1. Click on  **Assign properties** tool.
2. On the left side, a new window is displayed. Click on  **Properties for 2D elements**.
3. Select the surface body of the *Lower lining* on the model window and the set of parameter "Concrete" in the list.
4. **Apply**.




Boundary conditions:


There is no change in the boundary conditions. As we activate the lower lining, the existing fixities are adapted on its limit.

Loading sets:

The gravity forces are applied to the lower lining.

1. In the model tree, right click on *Loadings* of *Stage #5*. Click **Add loading set**.
2. Enter *Self-weight of lower lining* as a name.
3. Click **Validate**.
4. Go to the tab **LOADS**. On the toolbar, activate **Gravity forces** .
 - Click on **Automatic selection**. This feature detects the groups activated in the actual stage, i.e. the lower lining.
 - **Apply**.

The excavation forces are modified as the tunnel face has progressed.

1. Activate the **LOADS** tab.
2. On the toolbar, activate  **Excavation forces**.
 - Tick "Active"
 - Set the value of lambda to 0.4,
 - As stress origin, select *Stage #3* as new stress field origin.
3. Click on **Apply**. The excavation forces are applied.

Calculation parameters:

No modifications from previous stage #4.

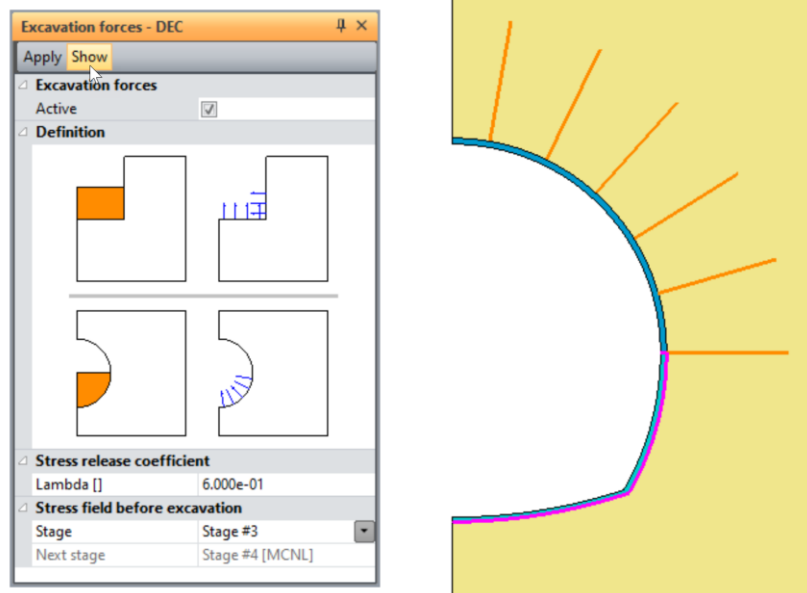




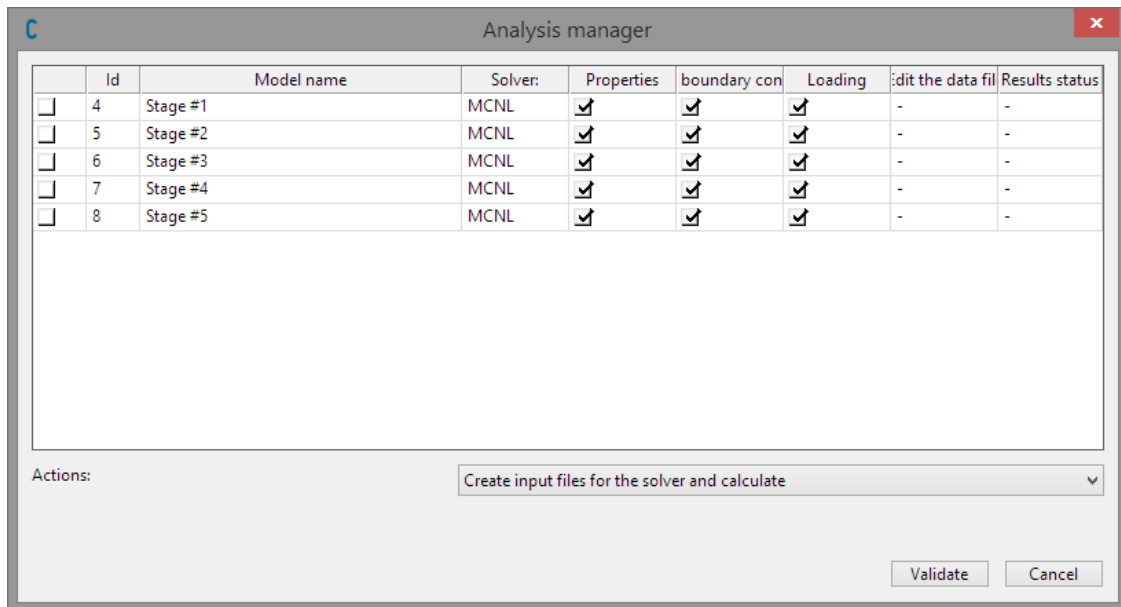
Figure 14: Display of the excavation forces at stage #5


4. SOLVE

We launch the calculations simultaneously. It is obviously possible to launch the calculations one by one.

1. Go to the **ANALYSIS** tab.
2. Click on  **Analysis manager**.
3. Select the 5 stages.
4. Select **Create input files for the solver and calculate**. Click on **Validate**.
5. The iteration process is displayed on the **Working window**. It ends with the message "End of analysis in EXEC mode".

 CESAR-LCPC detects if the models are ready for calculation. All steps should be validated with a tick mark.







 All the messages during the analysis will be shown in an **Output Window**. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The result is saved as a binary file (*.RSV4) in the temporary folder (.../TMP/), defined during setup. The detailed analysis information is also saved in a text file (*.LIST).

5. RESULTS



5.1. Displacements



The figures below show the total displacement at the end of the tunnel construction (Stage #5). This results may also be displayed at each stage with any other scalar (displacement, stress or strain).

1. Go to the tab **RESULTS**. Select the model named *Stage #5*.
2. Activate  **Result Types**
 - Select **Deformed** as Mesh,
 - Tick **Contour plots**, in the list select **Total displacement |u|**,
 - Click on **Apply**.
3. Activate  **Mesh settings**.
 - Select **Group borders**, as mesh border style,
 - **Apply**.
4. Activate  **Contour plots settings**.
 - Tick **Contouring**, as isovalues style,
 - Tick **Contour lines** and select **Grey** color,
 - Select **Standard** as **Color scheme**.
 - **Apply**.
5. Activate  **Legend**.
 - Select **Contour plot** as *Legend type*,
 - Activate *Legend border*,
 - Activate *Background colour*,
 - **Apply**.

We get Figure 15.

Figure 17 shows the settlement trough at the surface of the model after tunnel completion.

1. Activate the **CHARTS** tab.
2. Select the edges at the soil surface.
3. Click on  **Lines set**.
 - Give a name to this line set, **Surface** for example.
 - **Add**.
4. Activate  **Charts for a line set**.
 - Select **v, vertical displacement** as parameter,
 - Select **Surface** as line set,
 - Select the **Increment_1**,
 - **Apply**.

User can adjust the settings of the graph with tool . Data of the graphs can be exported in an external text file for use with an external grapher software (tool ).

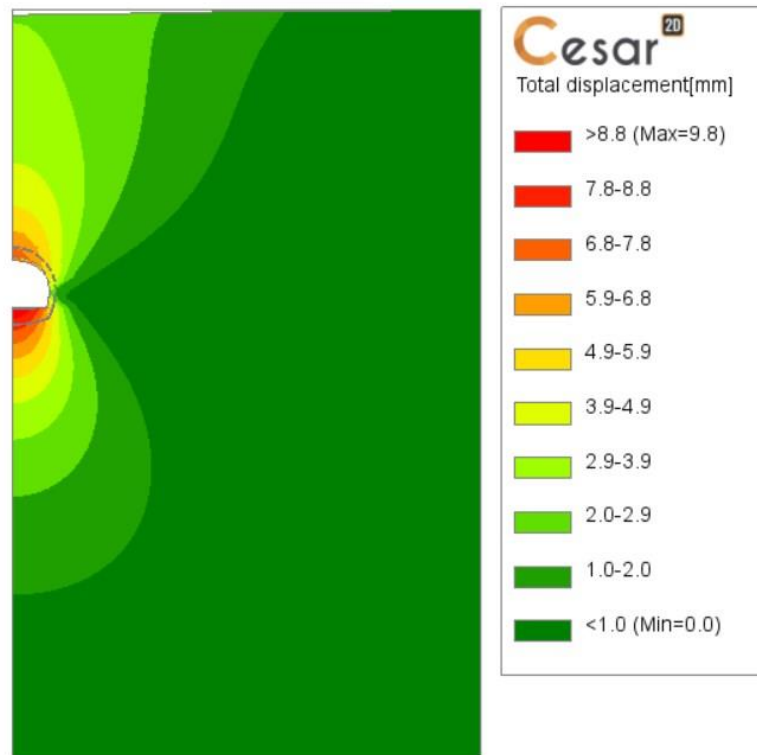


Figure 15: Display of the total displacements at stage #5

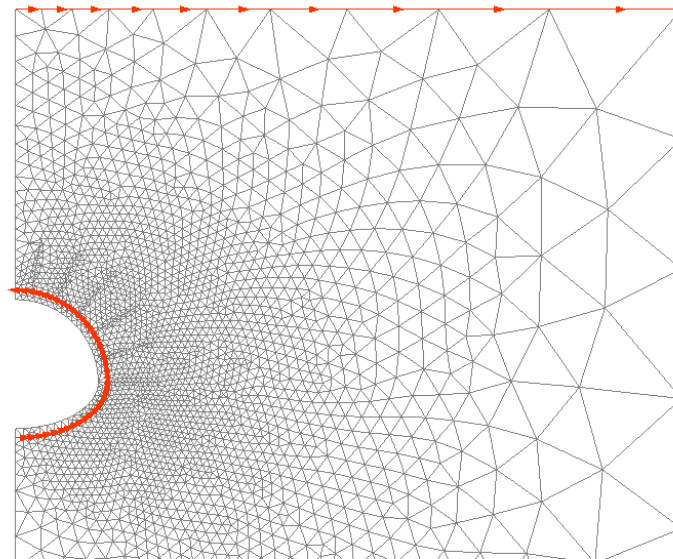


Figure 16: Line set generated at the surface of the soil and at the extrados of the tunnel.

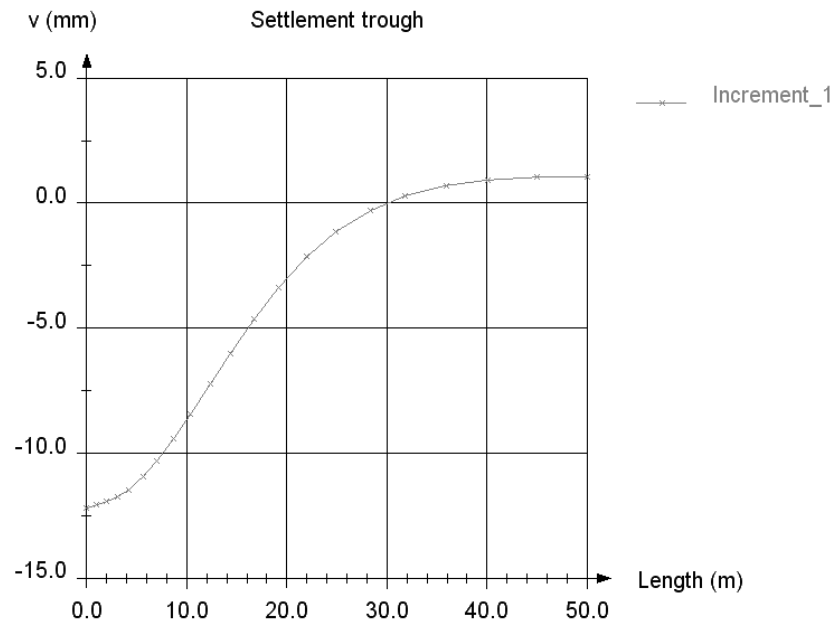




Figure 17: Settlement trough achieved at stage #5.

We can also isolate the tunnel section and get efforts in the bolts and in the lining.

1. Make a selection box with the selection tool .
2. Use  **Show selection only**

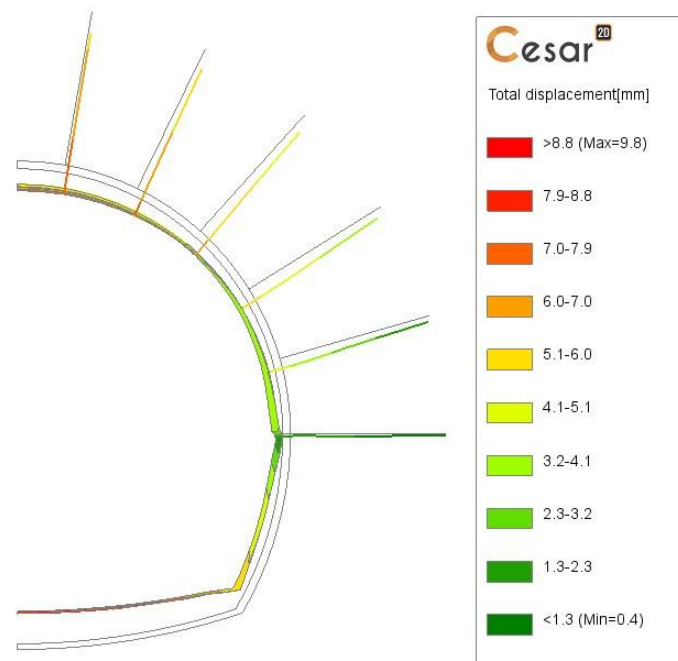



Figure 18: Display of the total displacements at stage #5 – Zoom on the tunnel and the bolts

5.2. Structural efforts

We get the effort in the bolts:

1. Activate  **Result Types**
 - Select **Undeformed** as Mesh,
 - Tick **1D-bodies results**, in the list select **Normal force**,
 - Click on **Apply**.

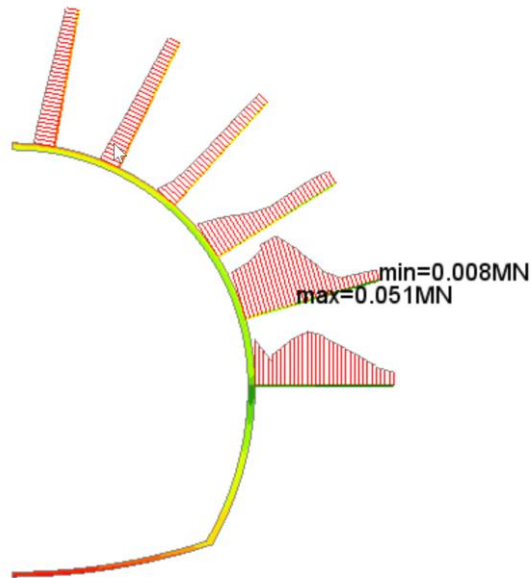




Figure 19: Display of the normal forces in the bolts at stage #5

In order to get the efforts in the lining, we will integrate the stresses for each section on the lining thickness. For this purpose, we first generate the integration line.

Figure 21 shows the settlement trough at the surface of the model after tunnel completion.

1. Activate the **CHARTS** tab.
2. Select the edges at the extrados of the tunnel section.
3. Click on  **.Lines set.**
 - Give a name to this line set, **Extrados** for example.
 - **Add.**
4. Activate  **Charts for a line set.**
 - Select **M, bending moment** as parameter,
 - Select **Extrados** as line set,
 - Select the **Increment_1**,
 - **Apply.**
5. Repeat previous operation to reach the bending moment distribution.

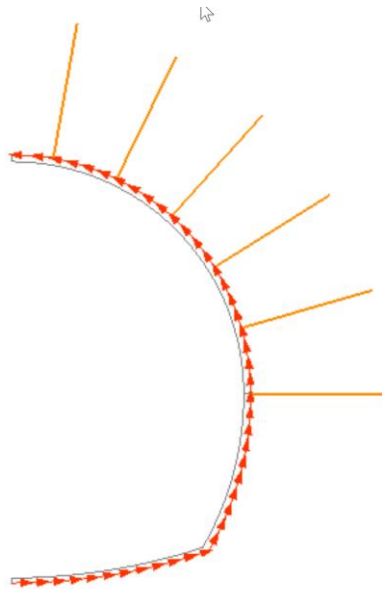


Figure 20: Line set generated at the extrados of the tunnel.

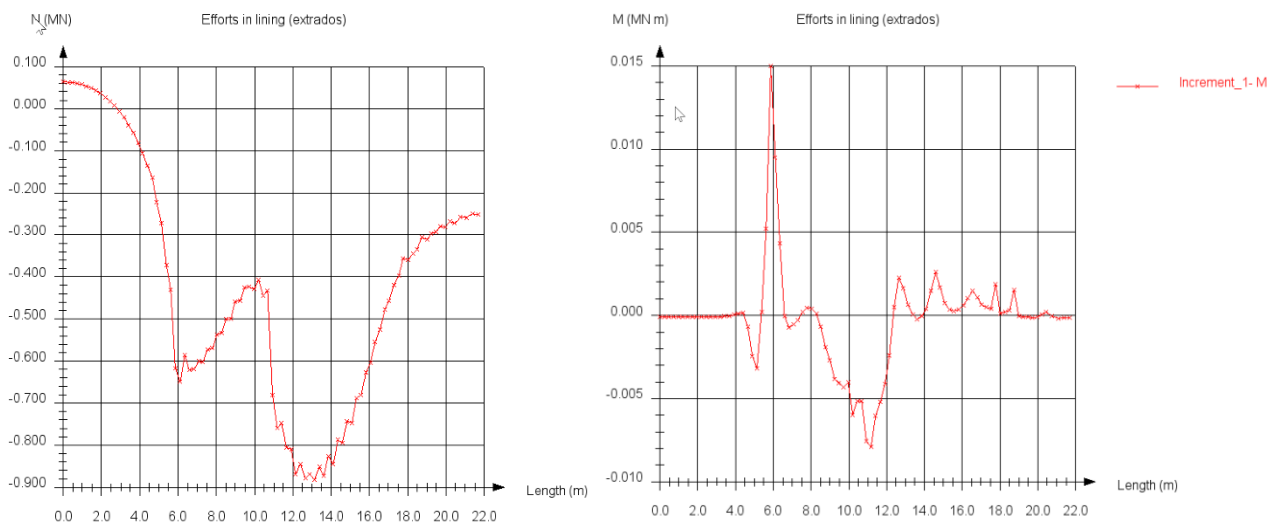


Figure 21: Normal forces in the lining, end of phase 5.



It is important to note here that the mesh type and density may have a large influence on the previous scalar values as they are the results of stresses integration on normal stresses in the lining thickness. It is highly recommended to use square shaped elements.

Edited by:



8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tél. : +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com

© itech - 2020