Tutorial 3g.03

Interconnexion of tunnels

Ref: CESAR-TUT(3g.03)-v2021.0.1-EN

1. INTRODUCTION

Tunnelling may be analysed as a 2-dimension model (cf. Tutorial 3) but, in most case, the geometry of the tunnel project in 3-dimensionnal. CESAR-LCPC offers the features to generate accurate geometry and model the stages of construction.

In this tutorial, we will analyse the excavation of a main tunnel gallery, previously to an adjacent gallery.

We focus on the CAD tools; the user will learn how to generate intersections (volume/volume, volume/plane…). The staged construction is also detailed but restricted to 2 excavations. The model is simplified in terms of geometry, terrain data and materials., in order to fit to the tutorial objectives.

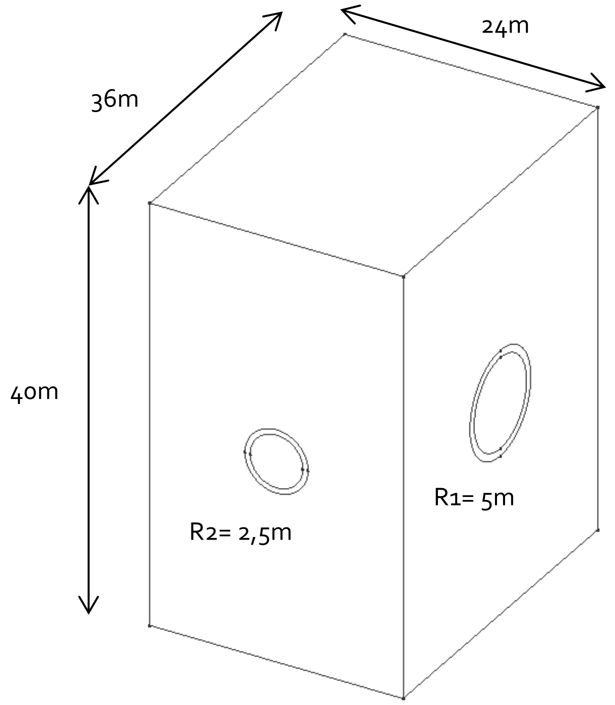
Note that, due to previous limitations, the dimensions extensions in horizontal axes might be considered as too small considering the areas of excavations.

1.1. *Problem Specifications*

General assumptions

- Static analysis,
- The lining is modelled with volume elements.

Dimensions



Material properties

The model of Mohr-Coulomb is used as plasticity criterion for calcareous marls, linear elasticity is used for the concrete of the lining.

	γ_h (kN/m3)	E (MPa)	ν	c (kPa)	φ (°)	ψ (°)
Calcareous marls	24	800	0.3	100	30	0
Concrete	27.5	25000	.2	-	-	-

2. 3D MESH

2.1. General settings

1. Launch CESAR-LCPC 3D
2. Set the units in the toolbox **Units** (at the top of the software window).
3. In the tree, select the leaf **General/Length** and set the unit **m** in the bottom left combo box.
4. In the tree, select the leaf **Mechanic/Force** and set the unit **MN** in the bottom left combo box.
5. In the tree, select the leaf **Mechanic/Displacement** and set the unit **mm**.
6. Click on ***Validate*** to close.

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

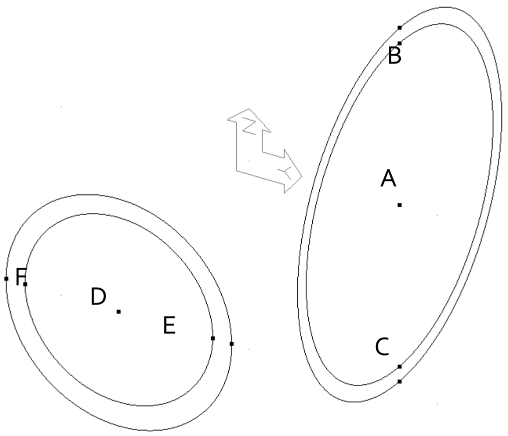
2.2. Geometry

Main gallery

1. Using ***Working plane***, set the working plane to Oxz
2. Using the circle tool . Draw 2 circles with radius of 4,3 m and 5 m and shared centre (0 ; 4 ; 0).
3. Detailed process for the 4,3 m radius circle.
   - Click on . The point definition dialog is displayed.
   - Define point A (0 ; 4 ; 0). ***Apply***.
   - Define point B (0 ; 4 ; 4,3). ***Apply***.
   - Define point C (0 ; 4 ; -4,3). ***Close***.
   - Click on .
   - Click successively point B, point C and finally point A. Repeat from point C to point B and finally point A.
4. Repeat previous operations for the 5,3 m radius circle.

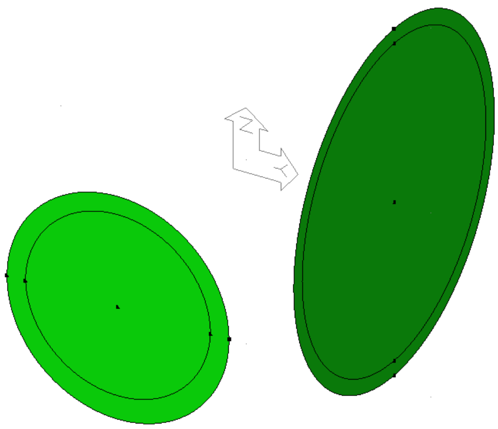
Adjacent gallery

1. Using ***Working plane***, set the working plane to Oyz
2. Using the circle tool ( ), draw 2 circles with radius of 2,5 m and 3 m and shared centre (6 ; 0 ; −1).
3. Detailed process for the 2,5 m radius circle.
   - Click on . The point definition dialog is displayed.
   - Define point D (6 ; 0 ; -1). ***Apply***.
   - Define point E (6 ; 2.5 ; -1). ***Apply***.
   - Define point F (6 ; -2.5 ; -1). ***Close***.
   - Click on .
   - Click successively point E, point F and finally point D. Repeat from point F to point E and finally point D.
4. Repeat previous operations for the 3 m radius circle.



Surfaces

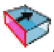
1. Select all the segments previously generated for the main gallery.
2. Use ***Plane surface***. Activate ***All regions on surface***. The surface regions are generated.
3. Repeat for the adjacent gallery.



Volume of the main gallery

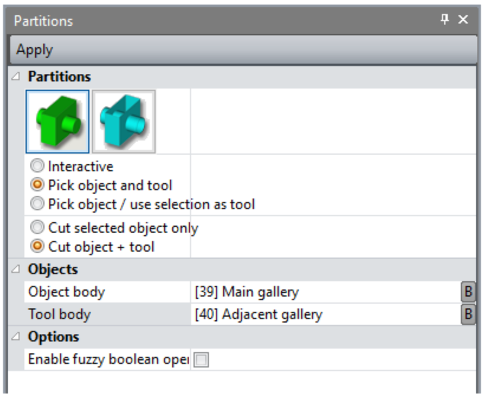
1. Select the surfaces of the main tunnel. For this selection, deactivate all ( )in the ***Selection toolbar*** and activate ***Select surfaces***.
2. Click on the ***Extrusion*** tool.
    - **Type of operation**: Translation
    - **Extrusion type:** Generation of volume bodies
    - **Number of operations:** 1
    - **Vector** : Vx = 0 ; Vy = --8 ; Vz = 0
    - Activate **Remove surface bodies**.
3. ***Apply***.

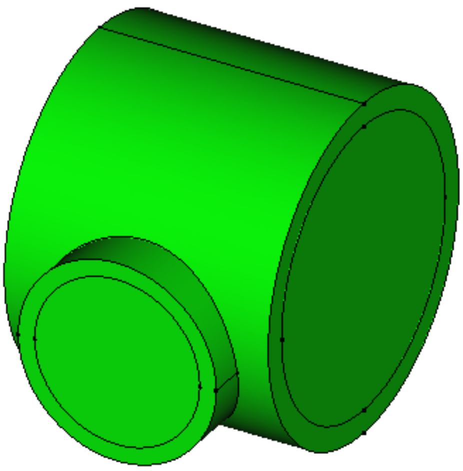
<u>Volume of the adjacent gallery</u>

1. Select the surfaces of the adjacent gallery.
2. Click on , the ***Extrusion*** feature.
   - **Type of operation**: Translation
   - **Extrusion type:** Generation of volume bodies
   - **Number of operations:** 1
   - **Vector** : Vx = -6 ; Vy = 0 ; Vz = 0
   - Activate **Remove surface bodies** (they are only supporting the volume bodies).
3. ***Apply***.

<u>Intersection of volume bodies</u>

1. Use  ***Bodies intersections***
   - Click  Solid/Solid partitions.
   - Activate Pick object and tool.
   - Activate Cut object and tool.
   - Select the 2 volumes of the main tunnel. Define them as "Object bodies".
   - Select the 2 volumes of the adjacent gallery. Define them as "Tool bodies".
2. ***Apply***.





State of the model after intersections of the 2 galleries

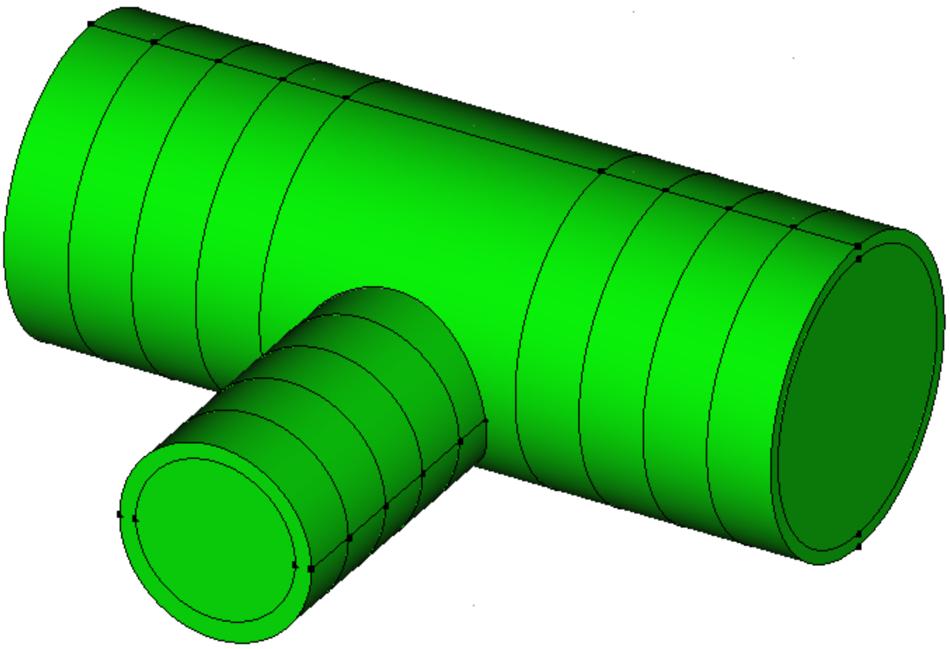
Excavated sections

We will now proceed to the generation of the volumes that will model the tunnel sections. For the main tunnel, these are 8 in number; each of them has a length of 2m. For the adjacent gallery, there are 5 sections; each of them has a length of 2m.

1. Select the 2 surfaces of the main tunnel gallery at y= 4 m. For this selection, deactivate all in the ***Selection toolbar*** ( ) and activate ***Select surfaces***.
2. Click on , the ***Extrusion*** feature:
   - **Type of operation**: Translation
   - **Number of operations**: 4
   - **Extrusion type:** Generation of volume bodies
   - **Vector** : Vx = 0 ; Vy = 2 ; Vz = 0
3. ***Apply***.
4. Select the 2 surfaces of the main tunnel gallery at y= -4 m.
5. Click on , the ***Extrusion*** feature:
   - **Type of operation**: Translation
   - **Number of operations**: 4
   - **Extrusion type:** Generation of volume bodies
   - **Vector** : Vx = 0 ; Vy = -2 ; Vz = 0
6. ***Apply***.
7. Select the 2 surfaces of the adjacent gallery at x= 6 m.
8. Click on , the ***Extrusion*** feature:
   - **Type of operation**: Translation
   - **Number of operations**: 5
   - **Extrusion type:** Generation of volume bodies
   - **Vector** : Vx = 2 ; Vy = 0 ; Vz = 0
9. ***Apply***.

Note that after these extrusion operations, the new volumes bodies are merged.

1. Select the new volumes bodies.
2. Use ***Explode bodies***.
3. ***Apply***.

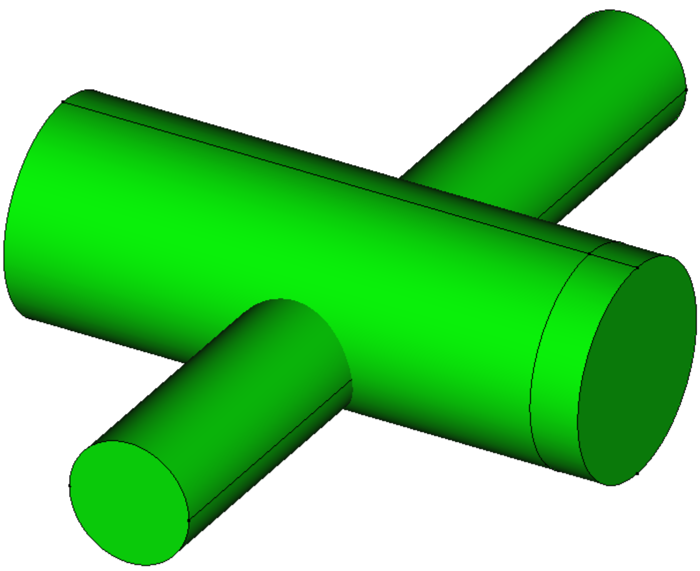


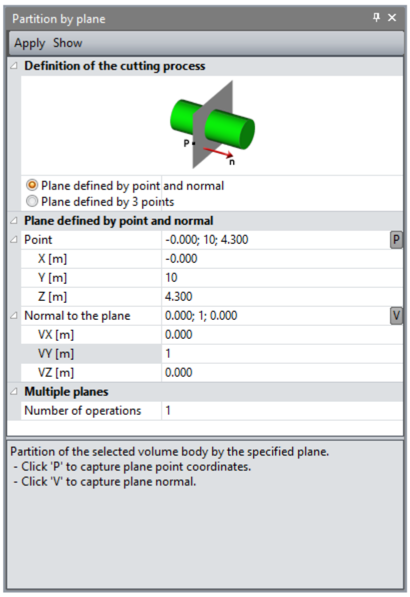
State of the model after extrusions in Ox and Oy directions

Other processes for intersection of the tunnel by a plane

Use of ***Intersection by plane***.

1. Open ***Intersection by plane***.
2. Indicate the coordinates of a point inside the plane.
3. Indicate the normal vector to the plane.
4. Select the volumes.
5. Click on ***Apply***.

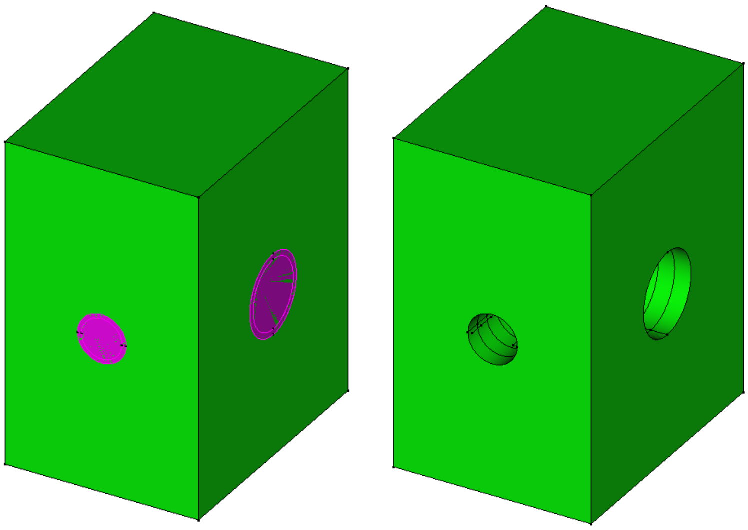




Soil mass

We define the soil volume that contains the 2 galleries. For this purpose, we define a "box" that will be intersected by the existing volumes.

1. Define the base plane at z = -20m.
   - Click on . The point definition dialog is displayed.
   - Define point G (16 ; 12 ; -20). ***Apply***.
   - Define point H (16 ; -12 ; -20). ***Apply***.
   - Define point H (-16 ; -12 ; -20). ***Apply***.
   - Define point H (-16 ; 12 ; -20). ***Close***.
   - Link these points with ***Lines***.
   - Select these segments and generate the surface using ***Plane surface***.
2. Extrude this base plane to z = 20m.
   - Select the surface previously defined.
   - Click on , the ***Extrusion*** feature.
   - Define the following parameters:
     - **Type of operation**: Translation
     - **Number of operations**: 1
     - **Extrusion type:** Generation of volume bodies
     - **Operation data**: Vx = 0 ; Vy = 0 ; Vz = 40
   - ***Apply***.
3. Use ***Bodies intersections***
   - Select all the volume bodies of the main and adjacent gallery.
   - Click Solid/Solid partitions.
   - Activate **Pick object / use selection as tool**.
   - Using "B", pick the volume of soil mass previously defined as **"Object body"**.
4. ***Apply***.



Galleries selected used as cutter and resulting volume body (with hidden galleries)

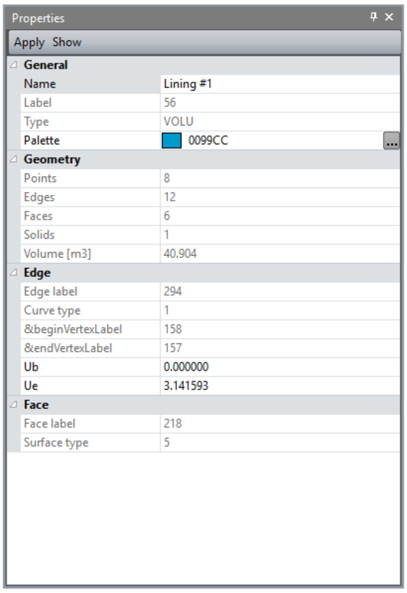
Identification of the bodies

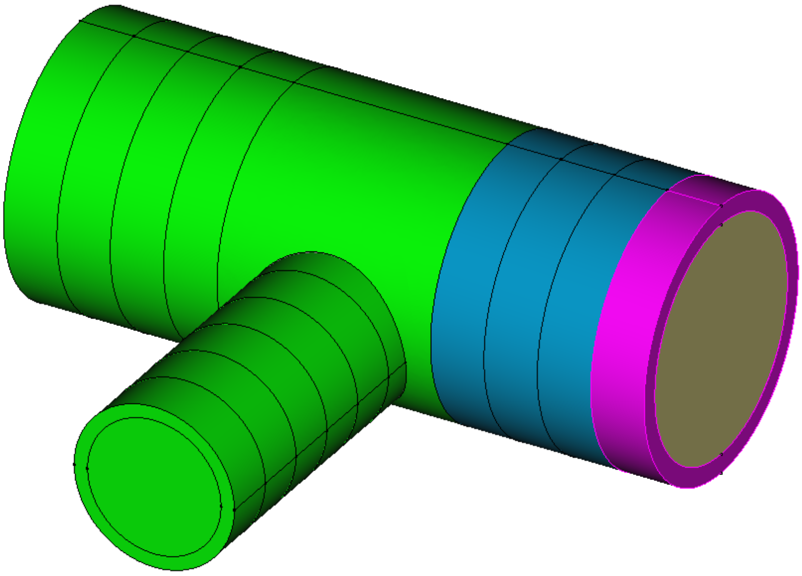
This step is optional.

In order to simplify the identification of the several volumes previously generated, we can name them and modify their color.

1. Select one of the volume bodies.
2. Activate tool ***Bodies properties***. By a right click on the body, we update the table of properties. In this table, it is possible to:
   - Specify a name,
   - Set any color.
3. Click ***Apply***. It saves the modifications and affect them to the selected volume body.

For the current example, we set as **Section #i** and **Lining #i** the bodies of the main gallery.





2.3. 3D meshing

Mesh density

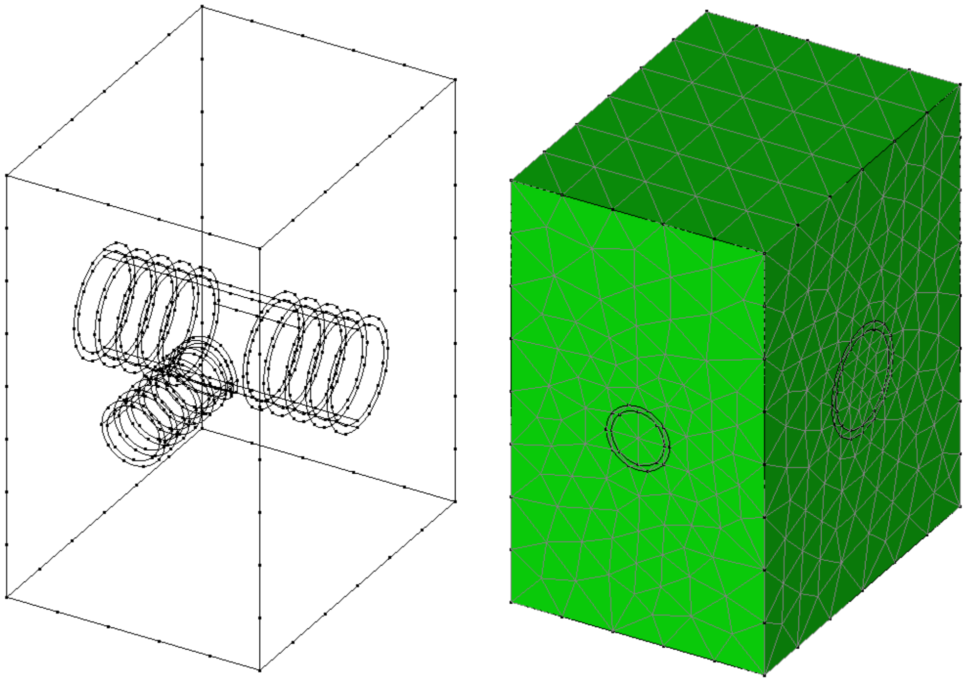
We will define dense mesh in the areas of high stresses, i.e. in the vicinity of the excavated galleries.

1. Go to the stage MESH on the project flow bar to start the definition of divisions along edges.

2. Select all the edges of the tunnel galleries (main and adjacent). Click on ***Fixed length density***. to divide these segments with a fixed length. Set it to **1 m**. Click on ***Validate***.

   ⚠ The software algorithm will adjust the length for the best fit with the input value of length.

3. Select all the external edges of the soil volume. Click on ***Fixed length density***. to divide these segments with a fixed length. Enter **5 m.** in the dialog box. Click on ***Validate***.



Model after density settings and after 3D meshing

3D mesh generation

For the tutorial purpose, we will generate a linear interpolated mesh instead of a quadratic interpolated mesh. This choice reduces the duration of calculations but in reduces the accuracy of results.

⚠ It is highly recommended to use quadratic interpolated mesh in nonlinear analysis.

1. Select all the volume bodies.

2. Open the tool ***Volume meshing***
   - Select "Linear interpolation" as **Interpolation type**.
   - Set "Tetrahedron mesh" as **Mesh type**.
   - Set **Density factor** to 0.8
   - Click on ***Validate***.

CESAR-LCPC proposes 3 levels for the meshing procedure of external surfaces of the volume. It enables to generate a coarse or dense mesh. The choice is made in Preferences>Program settings (Linear = coarse, cubic = dense).

Now that the mesh is completed, we check it.

3. Click on ***Mesh information***.
4. The toolbox displays the number of nodes and elements and the type of elements.
5. Click on ***Element quality check***. The toolbox is updated. If elements are deformed with wrong Jacobian, they appear in the toolbox.

Groups definition:

This step is optional but it eases the recognition of the group of elements if more than one has been generated.

1. Come back to GEOMETRY step.
2. Activate tool ***Bodies properties***.
3. Select the volume corresponding to the soil mass. Enter **Soil** as a name. Click on **Validate**.
4. Select the volume corresponding to the 1st excavated section of the main gallery. Enter **Section #1** as a name. Click on **Validate**.
5. Select the volume corresponding to the 1st lining of the main gallery. Enter **Lining #1** as a name. Click on ***Validate***.
6. Repeat previous operations 3 and 4 and name groups **Section #2** and **Lining #2**.
7. Continue operations...

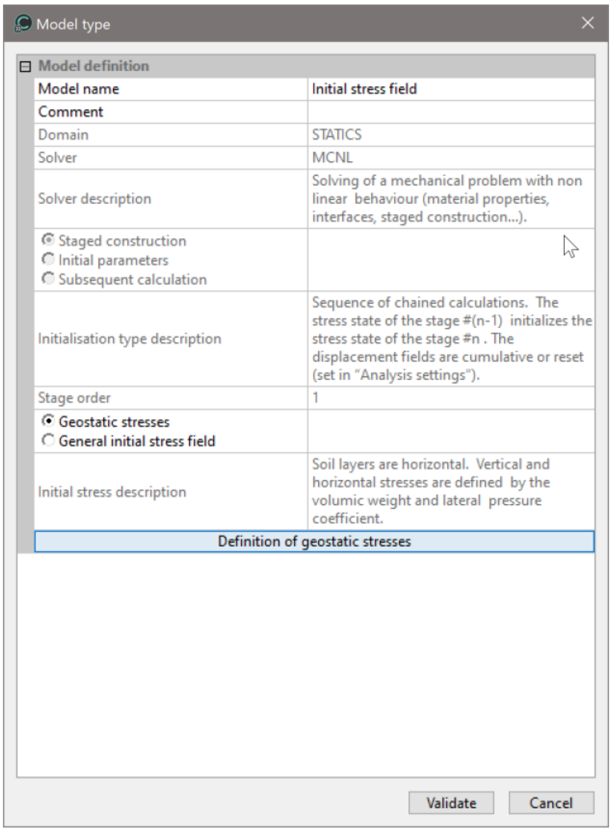
3. CALCULATION SETTINGS

3.1. *Calculation properties for the initial stress field: stage #0*

The staged construction process requires the definition of an initial stress field before applying loads. For this project, we follow the procedure defined in the document "Getting started with CESAR-LCPC v5", Chapter "Initial stress field". The initial stress field is initialized with the "Ko procedure".

Model definition

1. On the right side of the working window, the "Study tree view" 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 **Initial stress field** as "Model name".
3. Select **MCNL** as "Solver".
4. Tick **Staged construction**.
5. Tick **Geostatic stresses** as initialization type.
6. Click on ***Validate***.

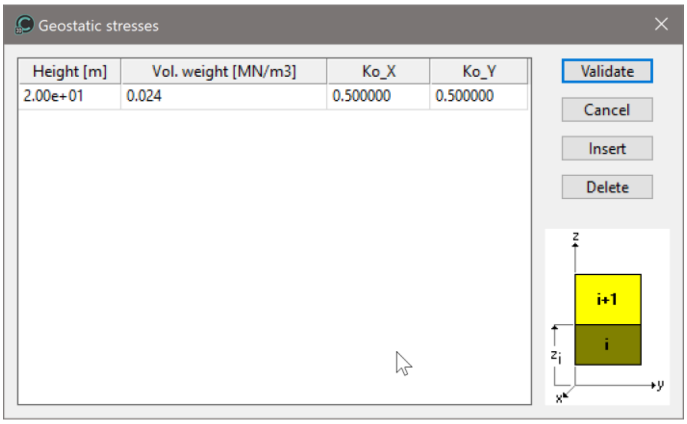


Initial geostatic stress field

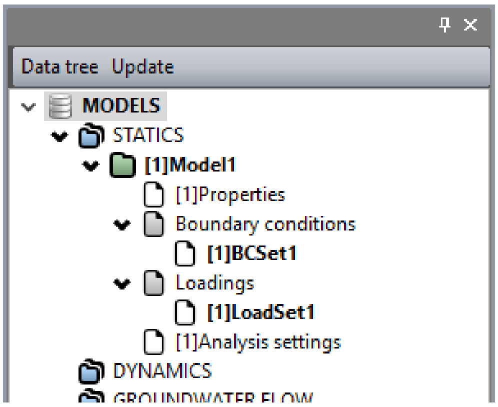
1. Click ***Definition of geostatic stresses***.
2. Click ***Insert*** to set a new soil layer.
3. Input following values:

Height (m)	Volumic weight (MN/m^3)	Ko_x	Ko_y
20	0.024	0.5	0.5

4. ***Validate***.



The date tree is now as illustrated below.



<u>Material database :</u>

We set here all the sets of material properties that will be used during the construction process : material of the soil mass and lining concrete.

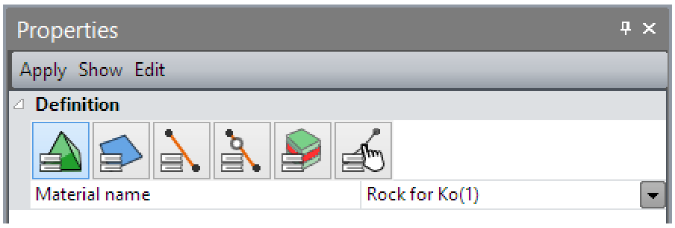
1. Go to step ***PROPERTIES***.
2. On the toolbar, click  ***Properties of volume bodies***.
3. Modify the default name of the properties set into "Concrete".
   - In **Elastic parameters**, chose "Liner isotropic elasticity" as constitutive model. Set ρ, E and ν (values are in table below).
4. Add a property set with tool . Give a name to the new set: "Soil".
   - In **Elastic parameters**, chose "Liner isotropic elasticity" as constitutive model. Set ρ, E and ν (values are in table below).
   - In **Plastic parameters**, chose "Drucker-Prager without hardening" as constitutive model. Set c, φ and ψ
5. ***Validate*** then ***Close***.

	ρ (kg/m3)	E (MPa)	ν	c (MPa)	φ (°)	ψ (°)
Concrete	2750	25000	0.2	-	-	-
Calcareous marls	2400	800	0.3	0.1	30	0

Assignment of properties 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 volume bodies***.
   - Select all the volume bodies of the model.
   - Select the and the set of parameter "Calacreous marls" in the list.
   - Apply.



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.

Optional. It is possible to modify the default name assigned to the boundary condition, BCSet1. Press [F2] ; enter **Standard fixities** for example.

Loading set:

There is no loading set to be defined in this first stage as we are initialising the geostatic stresses.

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: Multi frontal
3. Click on **Validate**.

User can also access calculation parameters settings in the calculation tree by a right cli on the model.

The option "Calculation with secondary storage" is required when the matrix size of the model will be larger than the random memory (RAM) of the computer.

3.2. *Calculation properties for the analysis of excavation #1*

In this stage, we excavate the full section of the main gallery. To unbalance the model, excavation forces on the section limits are applied.

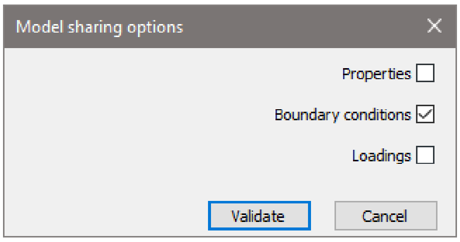
When creating a new analysis phase, it is handy to use parameters of the previous one by copy of the model. Subsequently, we make modifications where necessary:

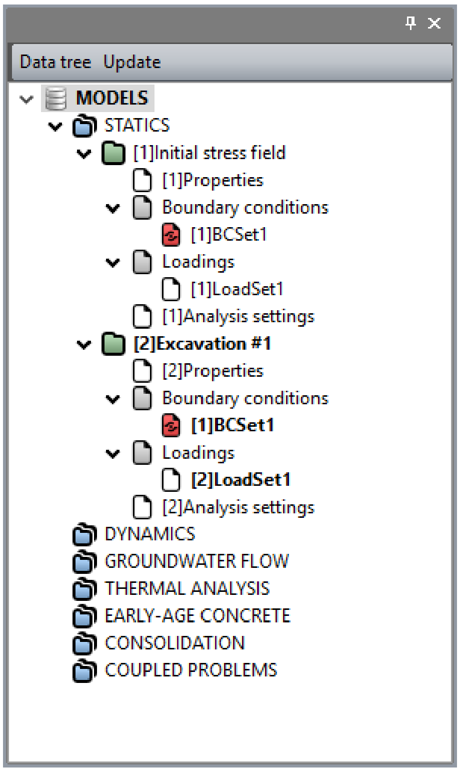
- In ***PROPERTIES*** as groups are activated,

- In ***LOADS*** in order to activate the excavation forces.

<u>Model definition</u>

1. In the "Tree view", right click on the model **Initial stress field**.
2. Click on ***Copy of the model***. A new toolbox is open for definition of the Model.
    - Enter **Excavation #1** as "Model name".
    - Click on ***Validate***.
3. A toolbox is displayed. User set here the options for sharing "Properties", "Boundary conditions" and "Loadings" of existing model. By default, all these entities are copied and share.
    - Select the "Boundary conditions" entry





View of the calculation tree

Active/inactive elements

The excavation of the full section leads to set at inactive state the bodies **Section #1** and **Lining #1**.

1. Go to the tab ***PROPERTIES***.
2. Select the volume bodies **Section #1** and **Lining #1**.
3. Click on ***Activate/deactivate bodies***.
4. In the grid, tick "Inactive".
5. ***Apply***. The volume body are now marked with the neutral colour.

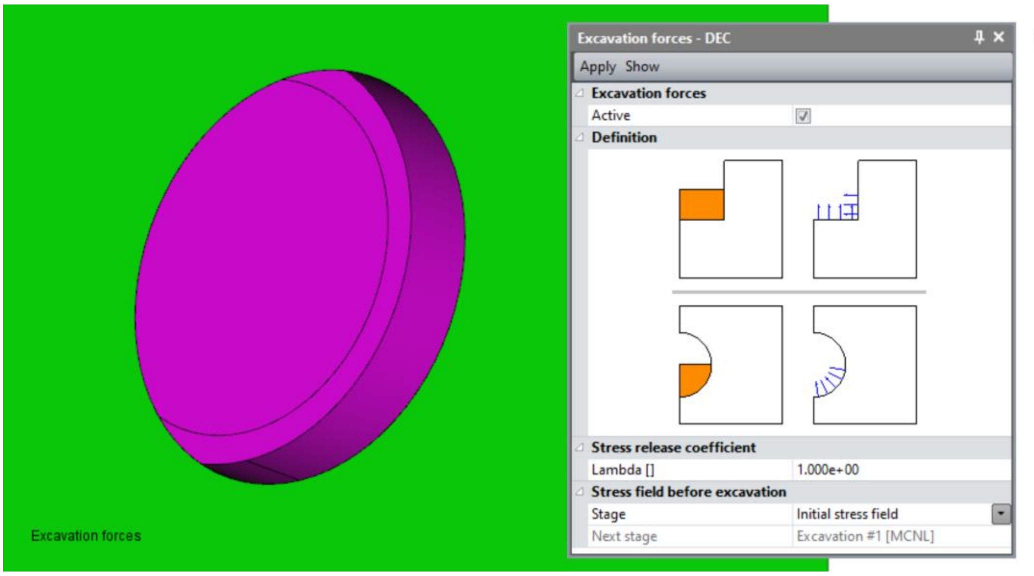
Boundary conditions:

There is no change in the boundary conditions.

Loading set: Excavation forces #1

The load set consists here to define the excavation forces resulting from the removal of the soil mass.

1. In the model tree, activate the default *Loading set* of the model **Excavation #1**.
2. Press [F2]; enter **Excavation forces #1** as a name.
3. Activate the ***LOADS*** tab.
4. On the toolbar, activate ***Excavation forces***.
   - Tick "Active"
   - Set the value of lambda to 1,
   - As stress origin, select **Initial stress field** model as stress field origin.
5. Click on ***Apply***. The excavation forces are applied.



View of the excavation forces

Calculation parameters

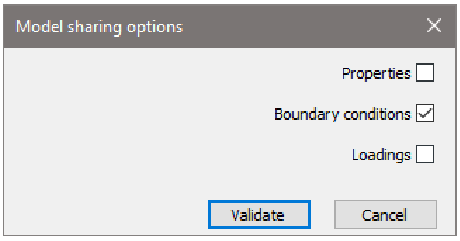
There is no change here.

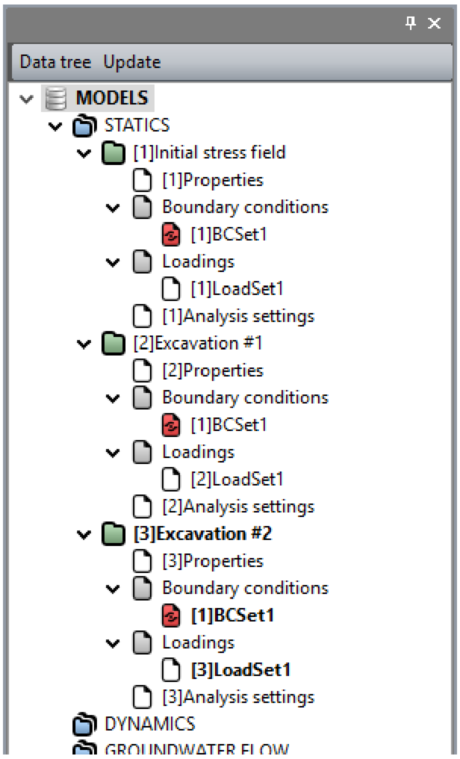
3.3. *Calculation properties for the analysis of excavation #2*

In this step, we excavate the second section of the main gallery and we setup the concrete lining on the section #1 previously excavated.

Model definition

1. In the "Tree view", right click on the model **Initial stress field**.
2. Click on ***Copy of the model***. A new toolbox is open for definition of the Model.
   - Enter **Excavation #2** as "Model name".
   - Click on ***Validate***.
3. A toolbox is displayed. User set here the options for sharing "Properties", "Boundary conditions" and "Loadings" of existing model. By default, all these entities are copied and share.
   - Select the "Boundary conditions" entry





View of the calculation tree

Active/inactive elements

The excavation of the full section leads to set at inactive state the bodies **Section #2** and **Lining #2**.

1. Go to the tab ***PROPERTIES***.
2. Select the volume bodies **Section #2** and **Lining #2**.
3. Click on ***Activate/deactivate bodies***.
4. In the grid, tick "Inactive".
5. ***Apply***. The volume body are now marked with the neutral colour.

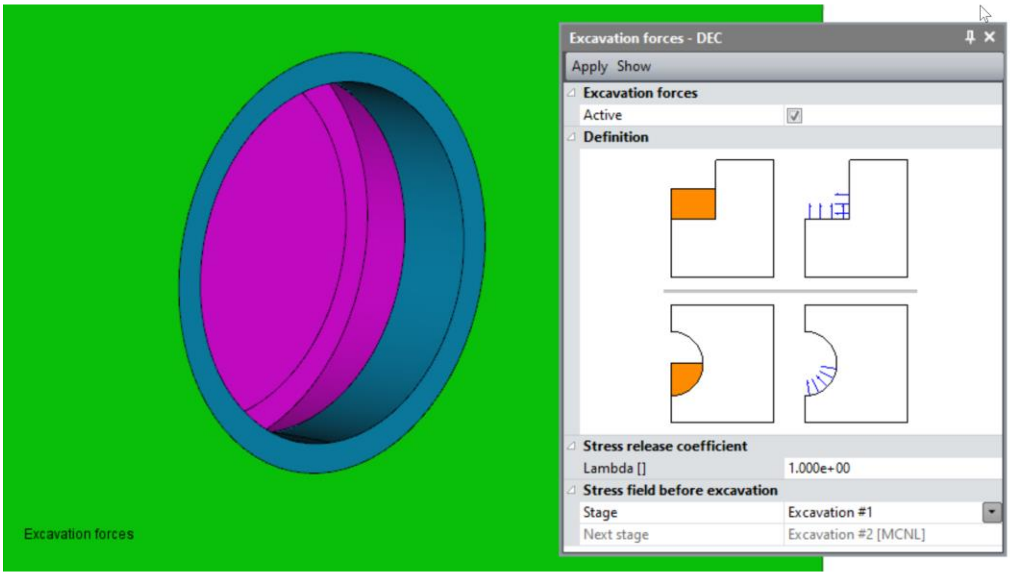
Boundary conditions:

There is no change in the boundary conditions.

Loading set: Excavation forces #2

The load set consists here to define the excavation forces resulting from the removal of the soil mass and to activate the self-weight of the activated lining.

1. In the model tree, right click on "Loadings" gives access to the command ***Add a loading set***. A dialog windows opens.
    - Give **Excavation forces #2** as name to this loading set.
    - ***Validate***.
2. In the model tree, right click on the loding set **Excavation forces #2** copied from previous stage. It gives access to the command ***Delete a loading set***. Delete it. **Excavation forces #2** is now the active loading set.
3. Activate the ***LOADS*** tab.
4. On the toolbar, activate ***Excavation forces***.
    - Tick "Active"
    - Set the value of lambda to 1,
    - As stress origin, select **Excavation #1** model as stress field origin.
5. Click on ***Apply***. The excavation forces are applied.

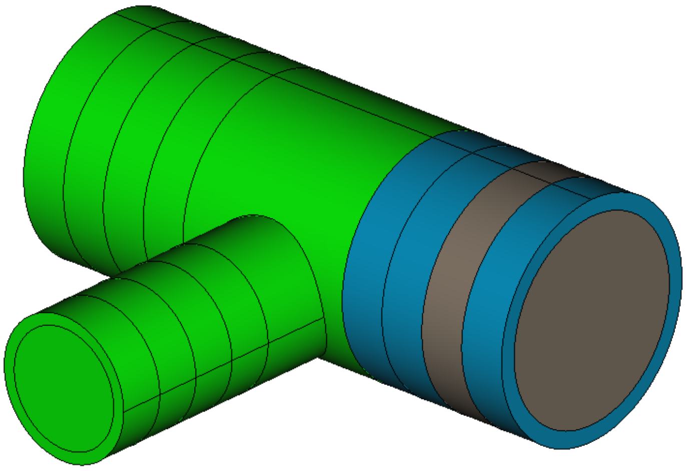


View of the excavation forces

Loading set: Self-weight of **Lining #1**.

The setup of the lining in section #1 requires to set at active state the body **Lining #1**.

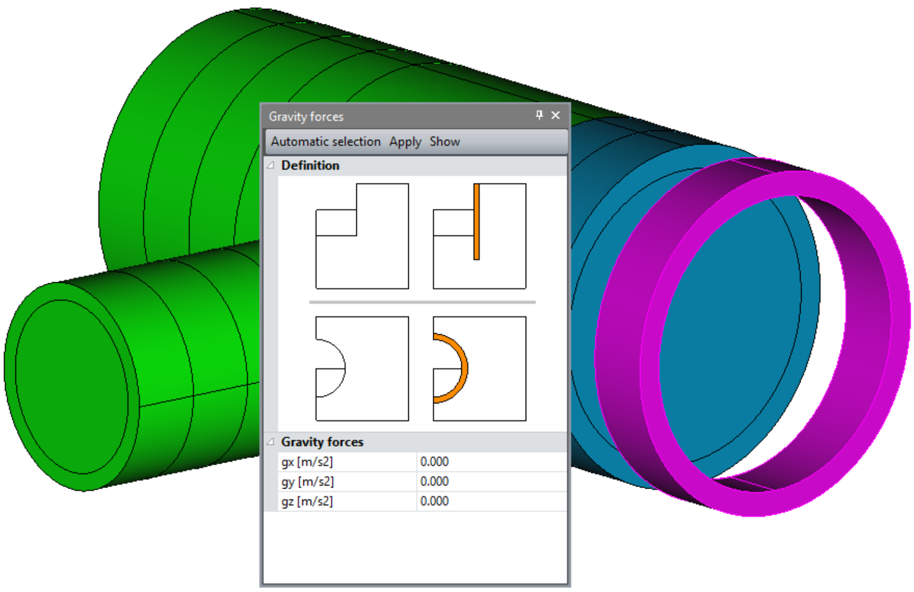
1. Go to the tab ***PROPERTIES***.
2. Select the volume bodies **Lining #1**.
3. Click on ***Activate/deactivate bodies***.
4. In the grid, tick "Active".
5. ***Apply***. The volume body are now displayed with its original colour.



View of the model (with the soil body being hidden)

We create a new loading set. Thus, we are able to pilot independently each loading set (if required).

1. In the model tree, right click on "Loadings" gives access to the command ***Add a loading set***. A dialog windows opens.
   - Give **Self-weight of lining #1** as name to this loading set.
   - ***Validate***.
2. Activate the feature ***Gravity forces***.
   - Click ***Automatic selection***. This tool automatically detects the activated bodies at this calculation stage (i.e. **Lining #1**)
   - ***Validate***.



View of the Lining #1 body selected for definition of its self-weight (with soil mass body being hidden)

Calculation parameters

There is no change here.

4. SOLVE

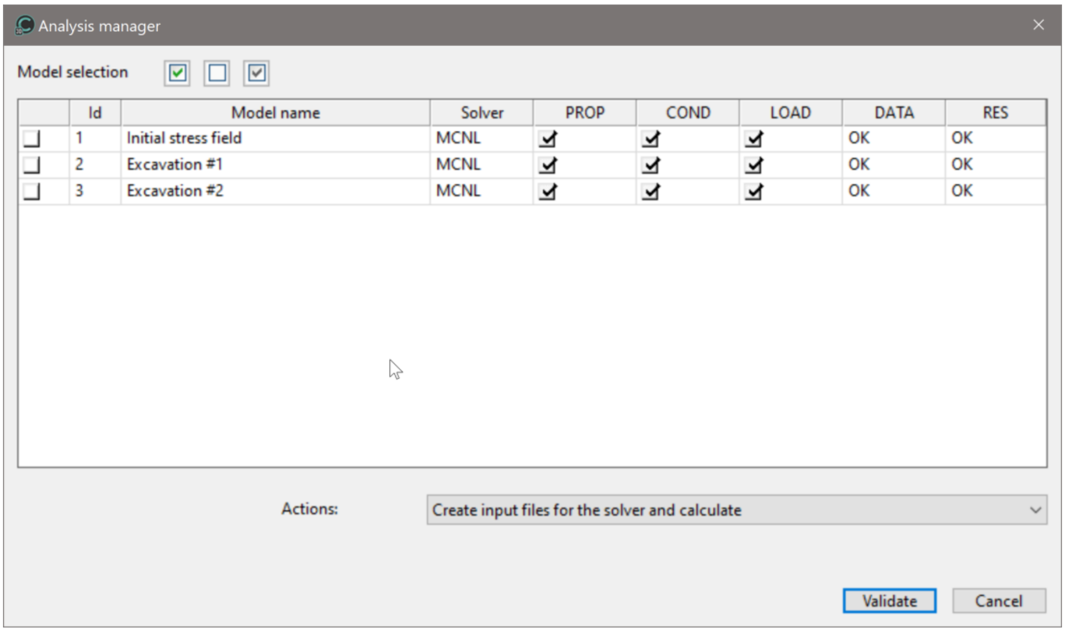
In the scope of the tutorial, we finish here and run the calculations.

User will understand that the process defined for modelling excavations #1 and #2 of the main gallery can be reproduced in the same way to complete the tunnel excavation as well as it can be adapted to model the excavation of the secondary gallery.

We launch the calculations simultaneously. It is of course possible to launch the calculations individually.

1. Go to the ***ANALYSIS*** tab.
2. Click on ***Calculation options***.
   - Check that **No renumbering** is ticked.
   - ***Apply***.
3. Click on ***Analysis manager***.
4. Select the 3 stages.
5. Select **Create input files for the solver and calculate**. Click on ***Validate***.
6. 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.



View of the calculation manager at the end of the calculations

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).

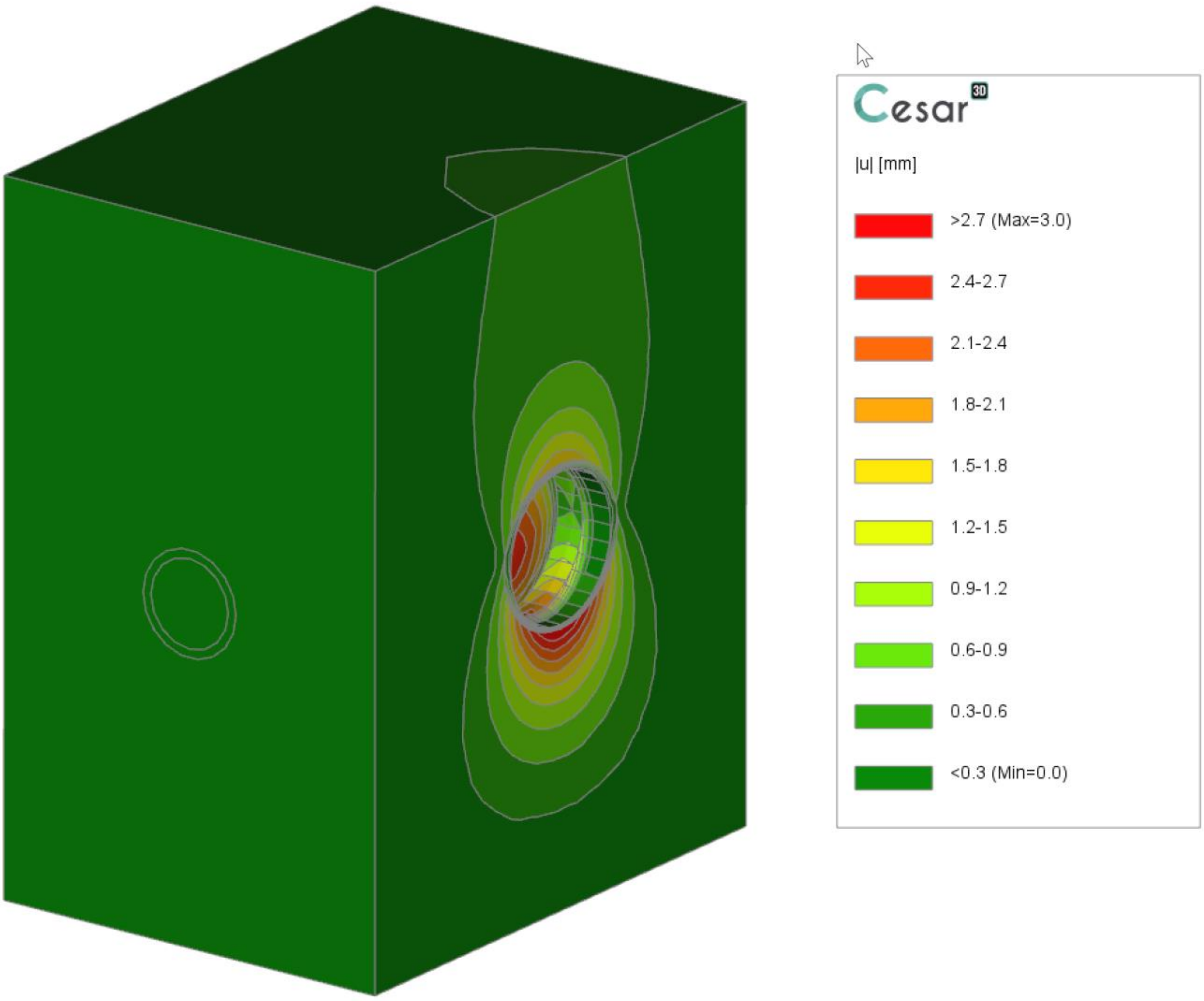
Calculation times depend of the computer performances (CPU and RAM configuration).

5. Results

User can display several scalar: displacements, stresses, strains.

We display the vertical stresses inside the model after analysis of **Excavation #2**.

1. Activate the tab ***Results***
2. Click on ***Type of results to display***.
   - Select **|u|, Total displacements** in Contour option,
   - ***Apply***.
3. Click on ***Scalar settings***.
   - Select **Areas** as Style of contour plots.
   - Activate **Contour lines**, and select **Grey** as color set,
   - ***Apply***.
4. Click on ***Legend***.
   - Select **Contour plot** as legend type,
   - ***Apply***.

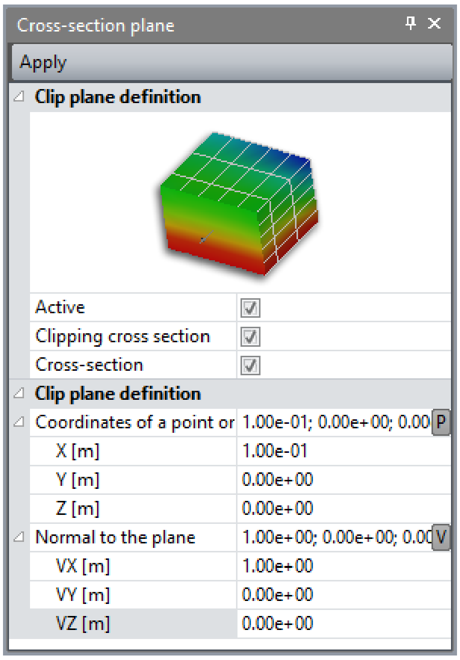


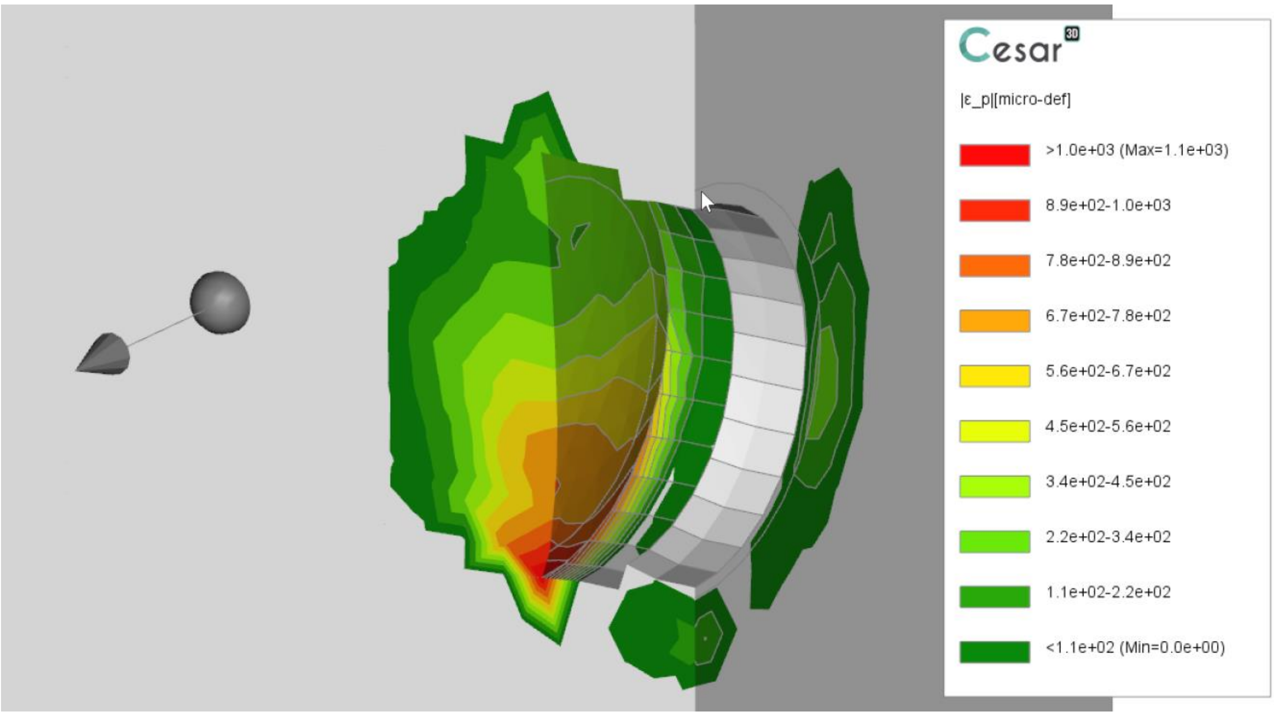
View of total displacements in the model at the end of Excavation #2

We now display the plastic strains and investigate inside the model with the tool "Cross section plane".

1. Click on ***Type of results to display***.
   - Select **|ε_p|, Norm of plastic strains** in Contour option,
   - ***Apply***.

2. Click on ***Cross section plane***.
   - Tick **Active**, **Clipping cross section** and **Cross section**,
   - Set the position to the point (0.1; 0; 0),
   - Set the direction of the cut plane following the vector (0; 1; 0).
   - ***Apply***.





View of plastic strains inside the model using the cutting plane feature

Edited by:

itech

8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tel.: +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com