# Cesar 3D

# Tutorial 3g.04

# Construction of a bolted tunnel

# 1. PREVIEW

In this tutorial we will model a reinforced tunnel in a complex stratum. First, we will create the soil mass. Then we create a basic model of tunnel, input properties of soil, rock bolts and shotcrete, then divide the tunnel and create a mesh. Finally, we will perform construction stage analysis and analyse the output results.

The main CAD tools for generation of the geometry are described. Also, we use the generation of reinforcement elements (rock bolts), linked to the tetrahedron mesh.

**Nota:** This tutorial has been fitted for an initiation to CESAR- v6 principles and features. Therefore, the geometry of the project has been simplified and the density of mesh (number of nodes) reduced for calculation speed. For the same purpose, the material parameters have been adapted.

## 1.1. *Problem Specifications*

General assumptions

- Static analysis,
- 3D geometry,
- The shotcrete is modelled with shell elements.

Material properties

The model of Mohr-Coulomb is used as plasticity criterion for the rock; linear elasticity is used for the concrete of the lining.

| | $\gamma$ (kN/m³) | E (MPa) | $\nu$ | c (kPa) | $\varphi$ (°) | $\psi$ (°) | K0 |
|---|---|---|---|---|---|---|---|
| Hard rock | 24 | 800 | 0,3 | 250 | 30 | 0 | 1 |

| | $\gamma$ (kN/m³) | E (MPa) | $\nu$ | e (m) |
|---|---|---|---|---|
| Shotcrete | 25 | 15000 | 0,2 | 0,15 |

Bolts are modeled as anchored bars with following properties.

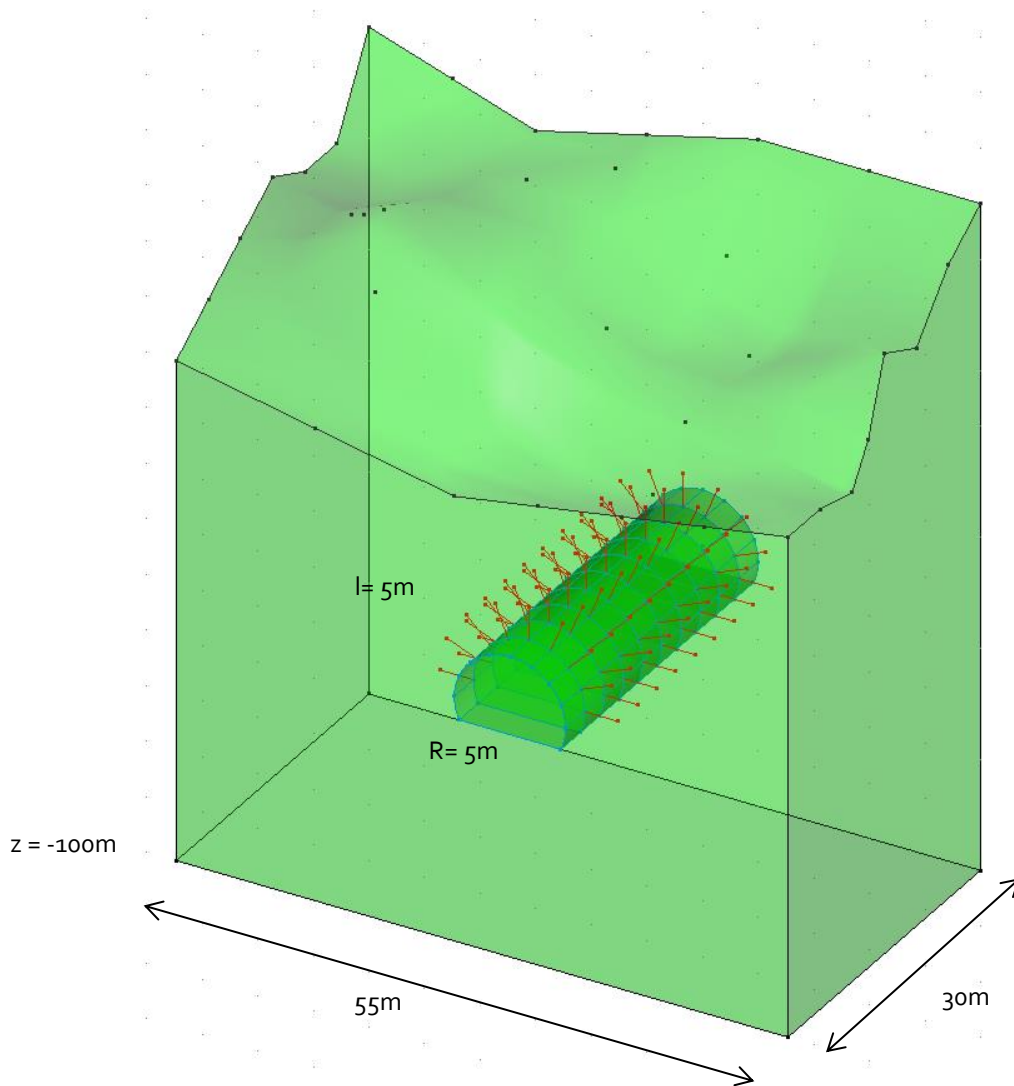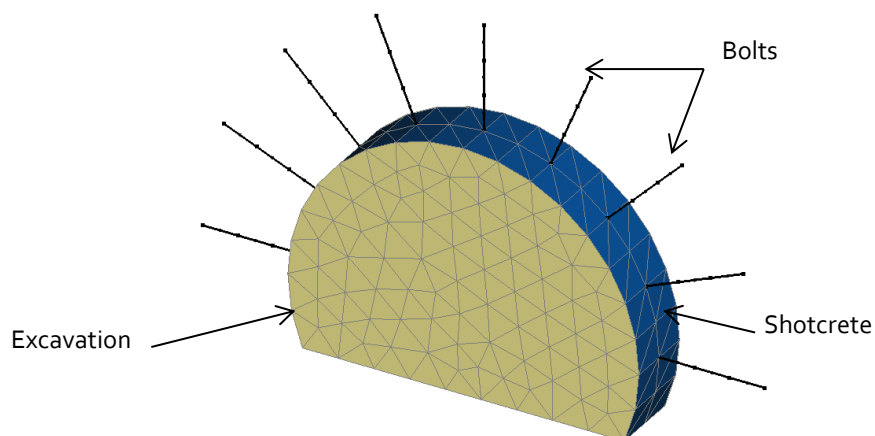| | E (MPa) | S (m²) |
|---|---|---|
| Bolts | 200000 | $5.10^{-4}$ |

Dimensions



*Figure 1: Dimensions*

The mesh requires 3 types of elements:

- volume elements = soil,

- 1D-elements = anchored bolts,

- shell elements = shotcrete.

## 2. 3D MODELLING

### 2.1. General settings

1. Run CESAR 3D.

Préferences

1. Set the preferences in the menu **Preferences** .
2. On the left, select the **Preferences** section and set the *Mesh creating function* to **Cubic**.
3. Click on **Apply** to close.

Units

1. Set the units in the menu **Units** .
2. On the left, select the **General** section and set the *Length* unit to **m** in the toolbox.
3. On the left, select the **Mechanic** section and set the *Force* unit to **MN**.
4. In the same way, select the *Displacement* unit to **mm**. Adjust the number of digits to 0.0.
5. Finally, select the leaf *Surface* and set the display in scientific digits.
6. Click on **Apply** to close.

💡 Use « Save as default" to define this unit system as the default user environment.

### 2.2. Geometry

CESAR v6 integrates a complete set of CAD tools. Import of external files is possible (dxf files as instance). We present hereafter the use of the integrated CAD tools.

A new project always starts in the tab **GEOMETRY**.

For the modeling of the excavation process, we will also distinguish 14 excavations, each being composed of:

- bolts (lineic elements),

- excavated soil (volume elements),

- shotcrete (shell elements).

Tunnel section

1. Using ⬚ **Working plane**, set the working plane to Oyz.

2. Activate 📜 **Script** and select TunnelM.xml (default one).

   - Select the tunnel shape ⬚, defined by 3 radius and 2 angles,
   - Untick "Asymmetry" and "Elliptical invert",
   - Set R1 = 5 m; A1 = 90°; R2 = 5m; A2 = 30°,

   - Excavation scheme = ⬚, full section,
   - **Apply**.
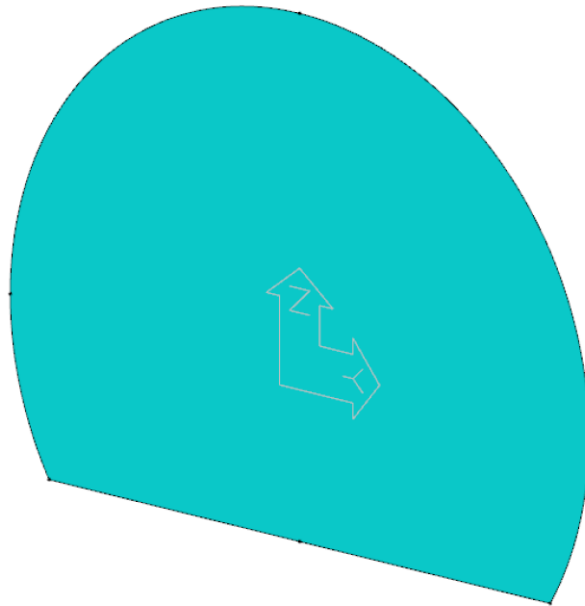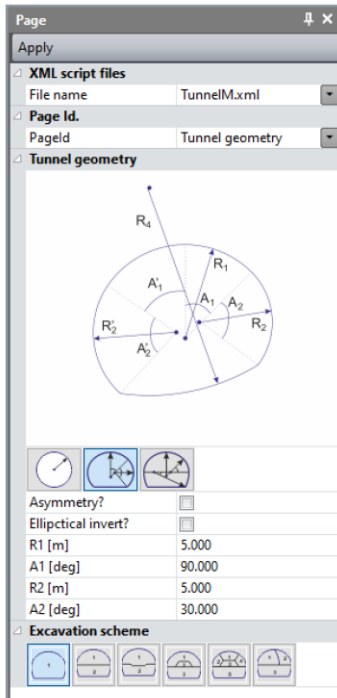
<u>Bolts</u>

Now we define the position of the 9 bolts.

1.  Select the upper half circle (2 arcs). Cut each arc in 4 equal segments with the cutting tool ✂
    (Number of divisions = 4).
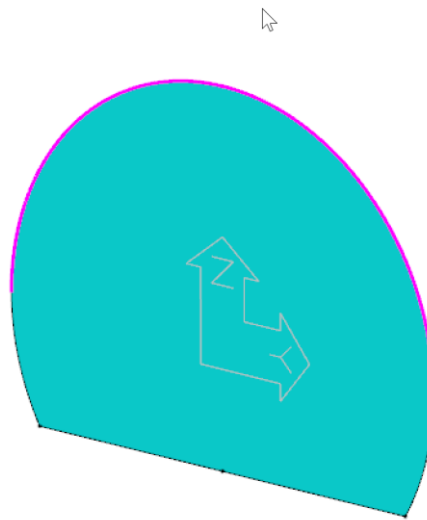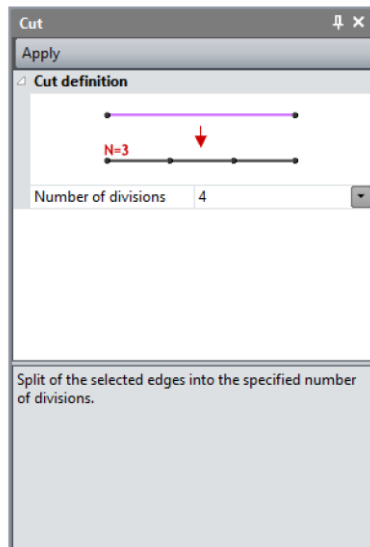


2.  Activate ✏ **1D-bodies**.
    - Select **Interactive** definition method,
    - Select **Anchored** body type,
    - Select **Length/Angle** as dynamic input.
    - Click on the first point at the middle of the tunnel section (see figure below)
    - Set L = 3 m and A = 360 deg,
    - Press {Enter}, the 1D body is generated
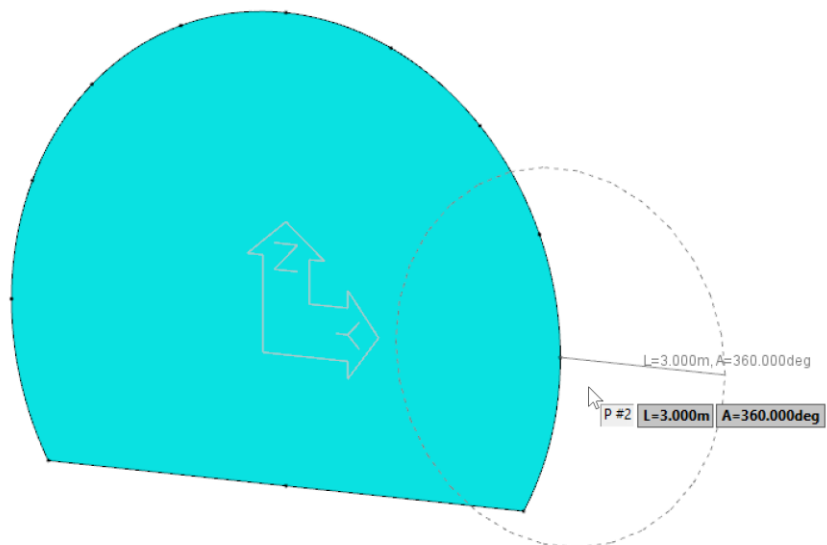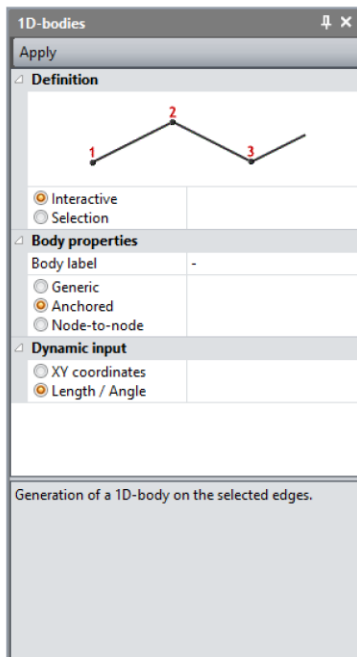


*State of the model before "second click"*
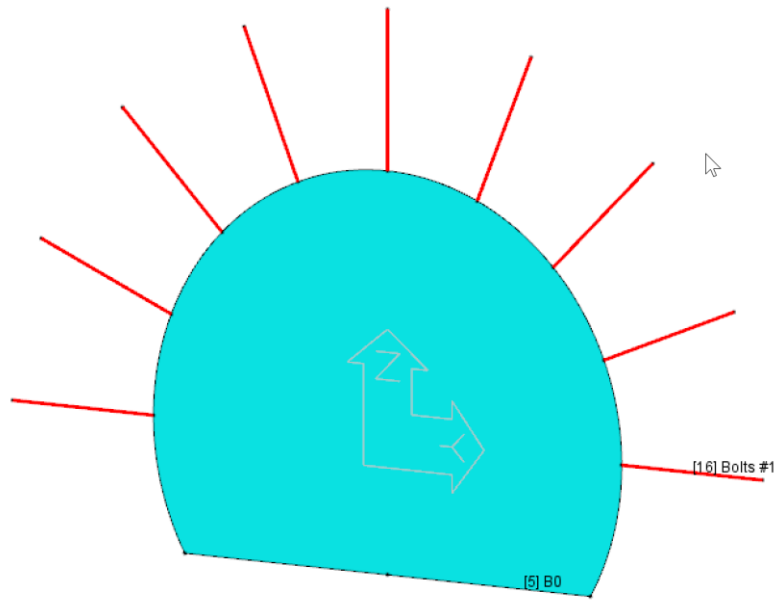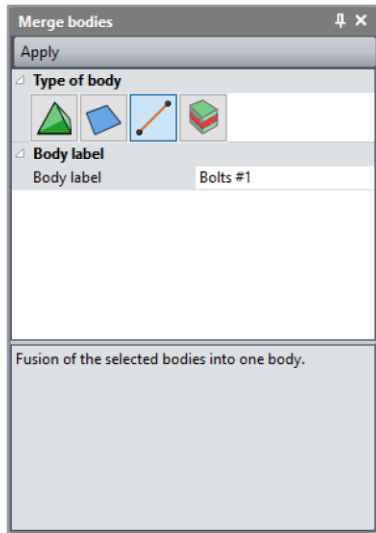
3.  Select it and open the tool 🖳T Translation/Rotation/Symmetry in the "Body operations" section.

    - Select **Rotation**,
        o **Number of operations** is 8.
        o **Angle** of rotation is 22.5°.
        o Define the axis of rotation with 2 points: Point F is P1 (0, 0 , 22). Point P2 coordinates are (1, 0 , 22).
        o **Apply**
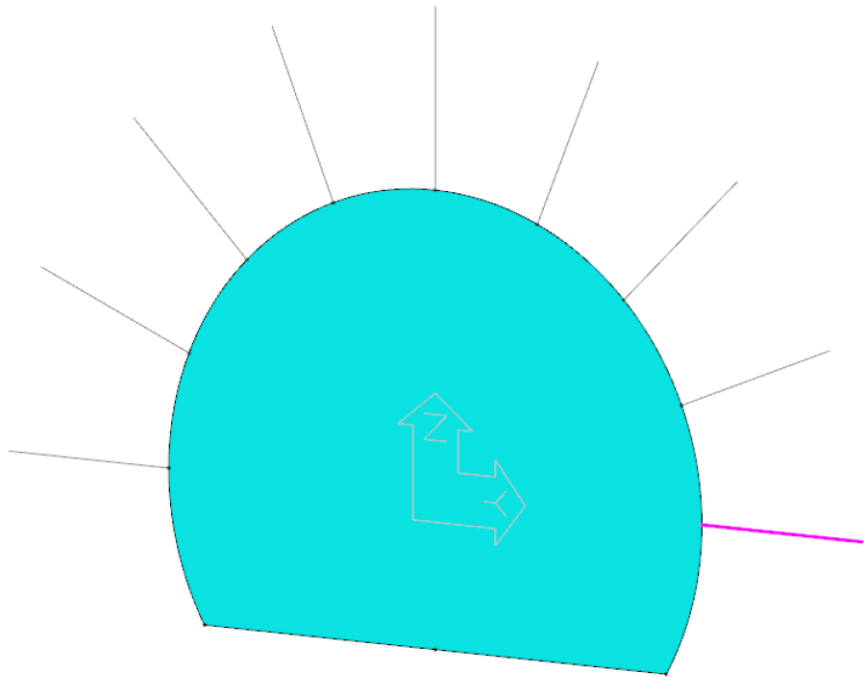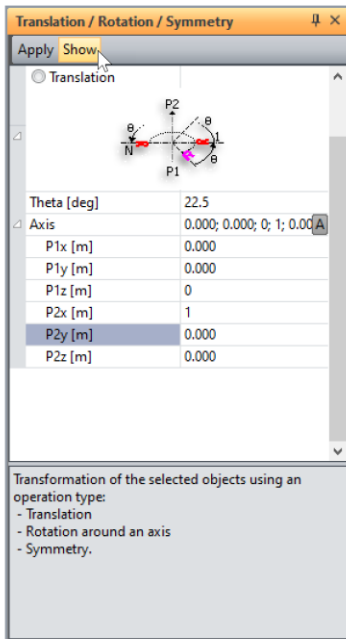


*State of the model with "show" option and before "Apply"*

For an easier edition, user can merge all the bolts, 1D-bodies, and set a specific color (red for example). It will help their selection in next actions.

1.  Select all the 1D-bodies

2.  Activate ⬚🗗 **Merge bodies**.

    - Select ╱ as type of body,
    - Set the label "Bolts #1".
    - **Apply**.

3.  Select this group of 1D bodies, in the vertical color palette, select the red colour. Bolts are now displayed in red.

**Merge bodies**

Apply

◢ **Type of body**

◢ **Body label**

Body label      Bolts #1

Fusion of the selected bodies into one body.

[16] Bolts #1

[5] B0

Excavation

1. Select the tunnel section. For this selection, deactivate all (✗) in the **Selection toolbar** and activate  **Select surfaces**.

2. Click on  , the **Extrusion** feature.

- **Type of operation:**    Translation
- **Type of extrusion:**    Generation of volume bodies
- **Operation data:**       Vx = 1 ; Vy = 0 ; Vz = 0
- **Apply**.

3. Select the generated volume body. For this selection, deactivate all (✗) in the **Selection toolbar**, activate  **Select bodies**.

4. Open the tool  **Translation/Rotation/Symmetry** in the "Body operations" section.

- **Type of operation**:        Translation
- **Operation data**:           Vx = -1 ; Vy = 0 ; Vz = 0
- **Number of operations**:     1.
- **Apply**.

5. Merge these two volume bodies as they represent the 2m excavation.

- Activate the tool  **Merge bodies**.
- Give a name to the new body. Body label = "Excavation #1" for example.
- **Apply**.
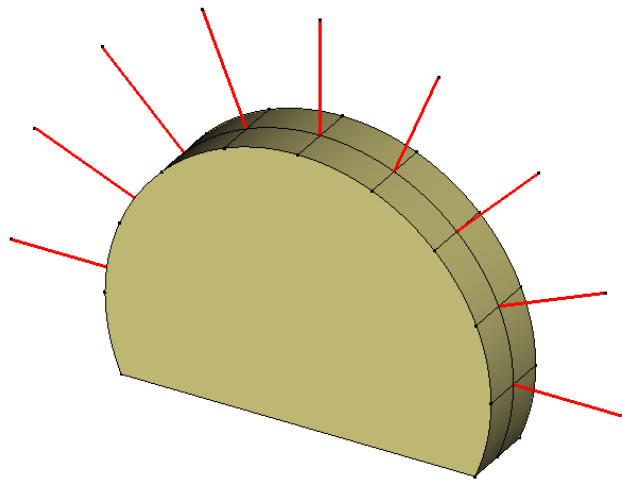


*View of the excavation and bolts*

Shotcrete

1. Set the **Selection options** 

- Items = faces
- Support body = Volume body

2. Select all external faces of the excavation.

3. Use  **Surface body on volume**.

- Enter a body label, "Shotcrete #1" for example.
- **Apply**.
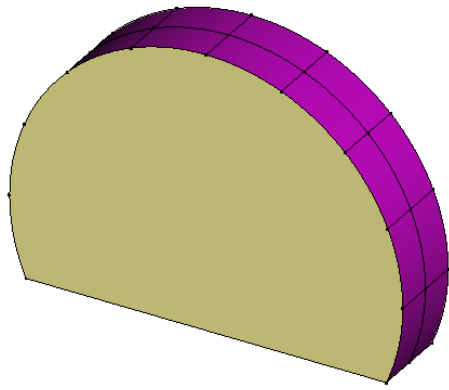
*Selection of faces on the volume body*
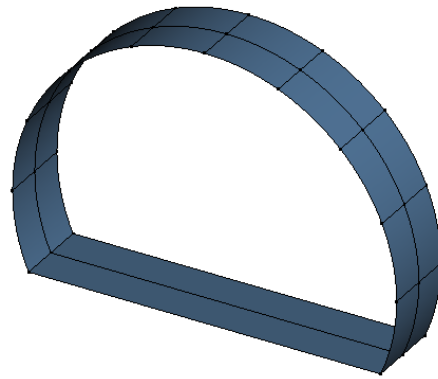


*Surface bodies created*

Gallery

The gallery with 15 excavations, shotcrete and crown of bolt reinforcement is accomplished by translation of the previous generated patterns.

1.  Select the following bodies :
    - Volume body "Excavation #1"
    - 1D-bodies "Bolts #1".
    - Surface body : Shotcrete #1

2.  Open the tool **T** *Translation/Rotation/Symmetry*.
    - **Type of operation**:        Translation
    - **Operation data**:           Vx = -2 ; Vy = 0 ; Vz = 0
    - **Number of operations**:     14
    - *Apply*.

3.  The gallery is completed.
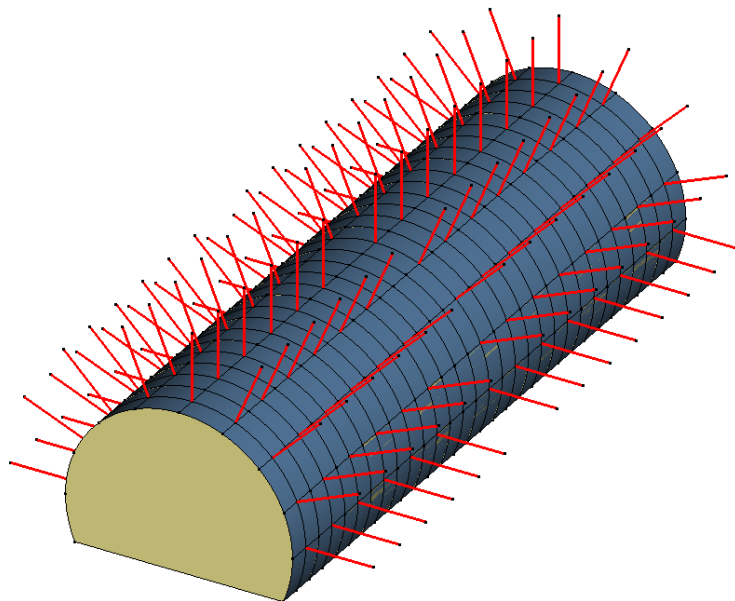
💡 Note here that the names of the bodies are automatically incremented. User can check these names in the data tree (see illustration). User can modify these names using the ⓘ*Properties* tool in the *Edit* section of the GEOMETRY toolbar.
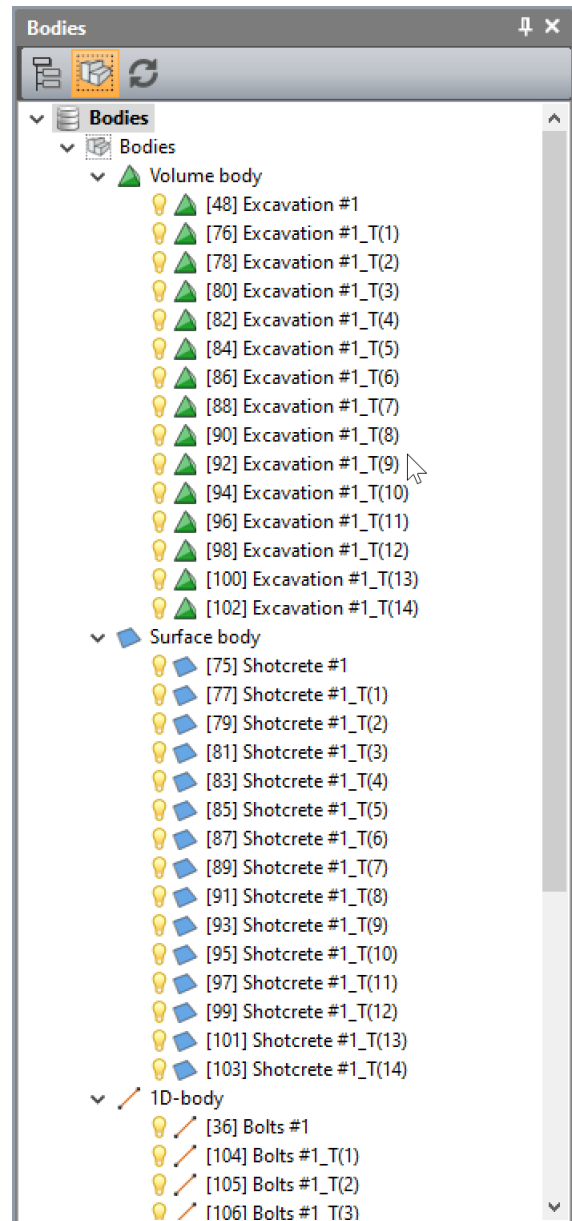
<u>Soil volume</u>

For an easy manipulation of the new entities, we hide all the bodies (volumes, surfaces and 1D bodies) previously generated for the gallery.

1. Select all the bodies.
2. In the selection toolbar, click ☁ *Hide selection*.

First operation consists of importing of a base geometry contained in an existing cleo3D file (this file can be a former CESAR 3D study or a dxf file).

3. Go in the menu **File>Import**.
4. Browse and find the file "Terrain.cleo36", attached to this Tutorial.
5. *Open*.

Edges and points are imported. We will use them to create the upper terrain limit.

1. Click on ◆ *Bspline surface*. Keep the default settings.
2. Select all edges and points.
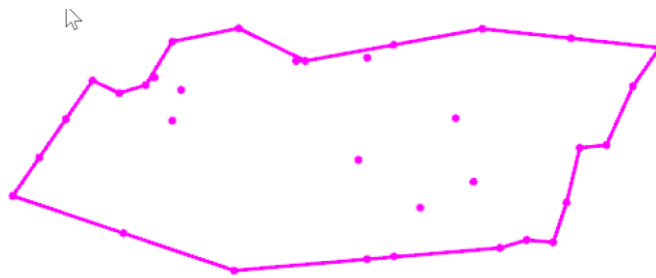


3. *Apply*.
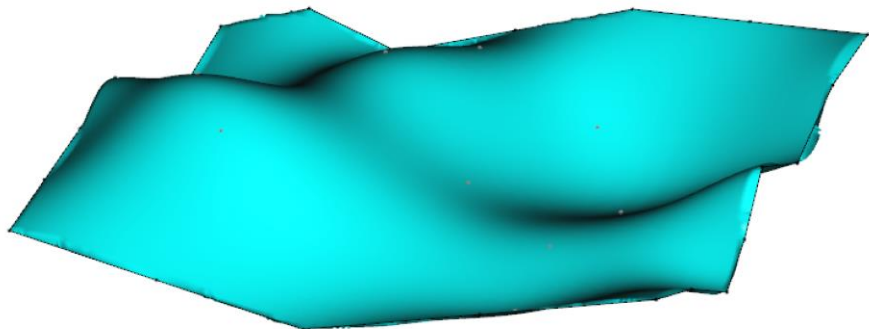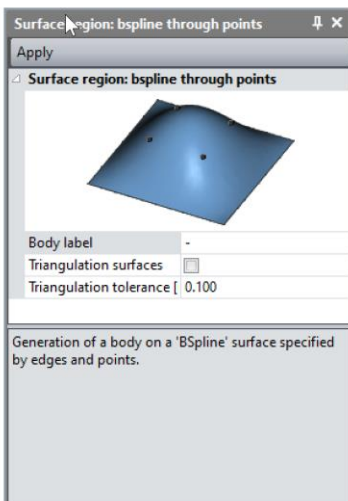


Lower surface generation at z= 0.

1. Click on ✗ . The point definition dialog is displayed.
2. Define point A (1 ; -30 ;-25). *Apply*.
3. Define point B (1 ; 25 ; -25). *Apply*.
4. Define point C (-29; 25 ; -25). *Apply*.
5. Define point D (-29; -30 ; -25). *Apply*.

6. Use the *Lines* tool. Link successively point A, B, C, D and finally point A.

7. Select all lines and use the tool *Plane surface* . The surface at z= om is generated.
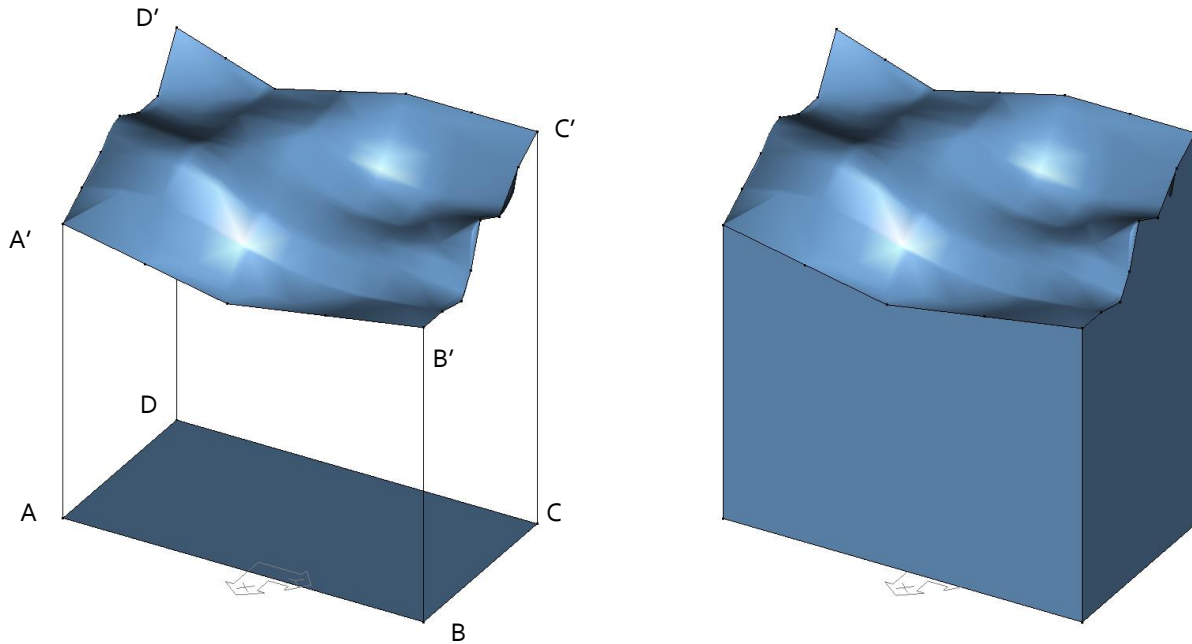


Complete the volume:

1. Draw the edges AA', BB', CC' and DD'.

2. Select all edges on each face of the volume and generate the surface using *Plane surface* . Use isoparametric views to help for the selection.

3. Select all the surfaces that close the volume (6). Use *Volume bounded by faces* to generate the volume.
   - Give "Soil" as body label.
   - *Apply*.
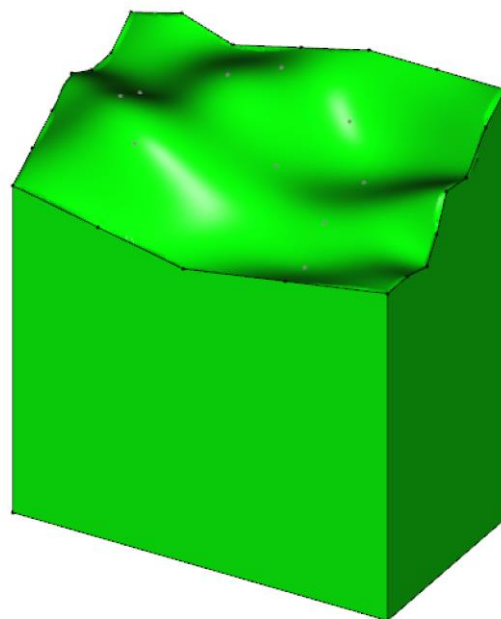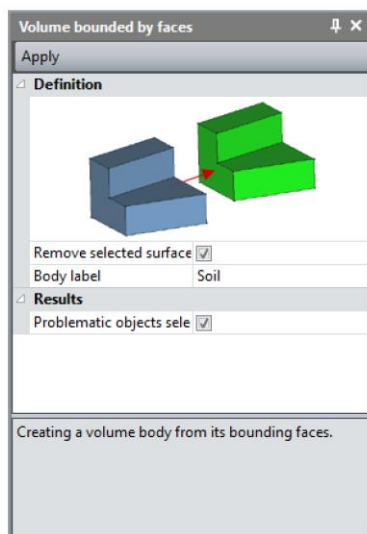
Volumes intersection

2 distinct volumes set have been defined: volume of the soil and volumes of the gallery (excavations). We will now generate their intersection.

1. Select all the volumes of the gallery.

2. Click on ![icon] **Bodies intersection**

   - Activate **Pick object / use selection as tool** option ;
   - As object body; pick the "Soil" body using tool B .

3. **Apply**.

## Fix geometry

**Check**  **Check / select**  **Fix**

| | |
|---|---|
| **Database** | |
| Faces and volumes not used by bodies | ☑ |
| Shapes not attached to the database | ☑ |
| **Coincident shapes** | |
| Geometrically coincident shapes | ☑ |
| **Check edges** | |
| Degenerated edges | ☑ |
| Edges with length less than [m] | 0.000 |
| Edges with bad parameters | ☑ |
| **Check faces** | |
| Degenerated faces | ☑ |
| Faces with bad orientation | ☑ |
| Faces with bad tolerance | ☑ |
| **Check volumes** | |
| Solids with bad faces orientation | ☑ |
| Extruded solids with errors | ☑ |
| **Check bodies** | |
| Bodies with errors | ☑ |
| **Results** | |
| Faces with bad tolerance | 1 |

## 2.3. 3D meshing

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

Mesh density

1. Go to the tab **MESH** on the project flow bar to start the definition of divisions along lines.

2. Select the complete model. Click on ⛔ **Fixed length density**. to divide these segments with a fixed length. Enter **5 m.** in the dialog box. Click on **Validate**.

3. Select all the edges of the tunnel gallery. Click on ⛔ **Fixed length density**. to divide these segments with a fixed length. Set it to **1m**. Click on **Apply**.

4. Select all the edges of the bolts. Click on ⛔ **Fixed length density**. to divide these segments with a fixed length. Set it to **0.5m**. Click on **Apply**.

⚠ The software algorithm will adjust the length for the best fit with the input value of length.
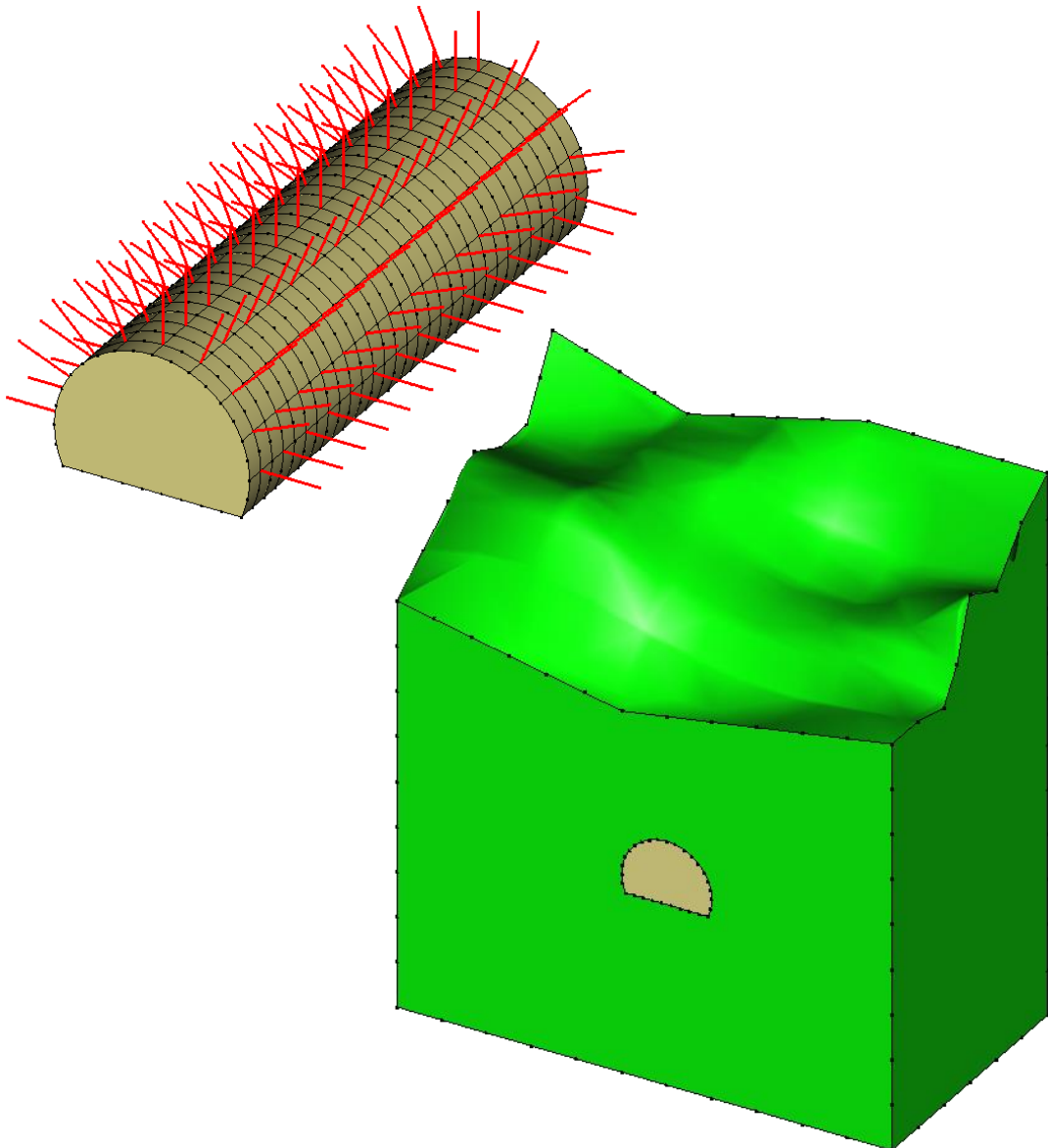
💡 An easy way to select all the edges of the tunnel galley and bolts, is to use the 🔲 *Front view* in isoparamatric views and make a selection box (see figure below).
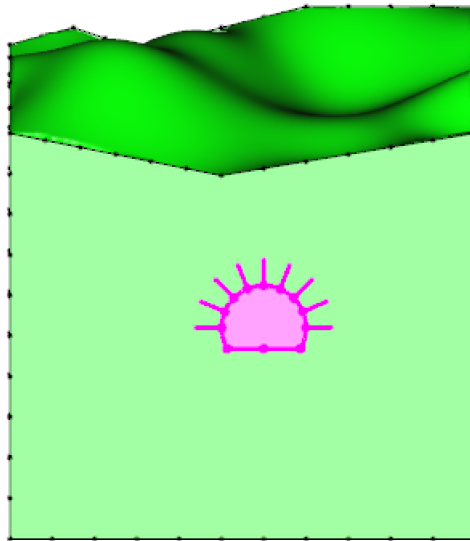


3D mesh generation

We start with the bolts generation.

1. Select the 1D-bodies of the bolts. Use the selection by color tool and select the red ones.

2. Use ✏ *Mesh of 1D-bodies*.

3. Set the interpolation type to **Linear interpolation**.

4. *Apply*.

Now we define these line elements as anchored elements so that they will be connected with the tetrahedron mesh.

1. Open the tool ✏ *Anchored elements*.

2. Select the 1D-bodies of the bolts. Use the selection by color tool and select the red ones.

3. Apply.
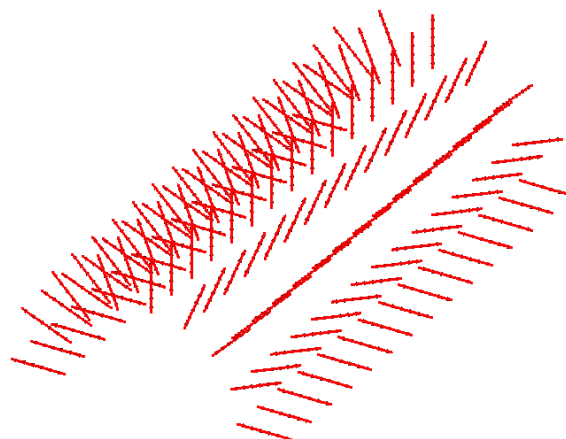


We can now mesh the volumes.

💡 CESAR-LCPC proposes 3 levels for the meshing procedure of external surfaces of the volume. It enables to definition the progression of the meshing algorithm from small to large elements. The choice is made in Preferences> Mesh > Mesh creating function (Linear = coarse, cubic = dense).

1.   Select all the volume bodies.

2.   Open the tool  *Volume meshing*
   - Select "Linear interpolation" as **Interpolation type**.
   - Set "Tetrahedron mesh" as **Mesh type**.
   - Select the default Tetmesh generator and set the **Density factor** to 1.0.
   - Mesh creating function is **Cubic**.
   - *Apply*.
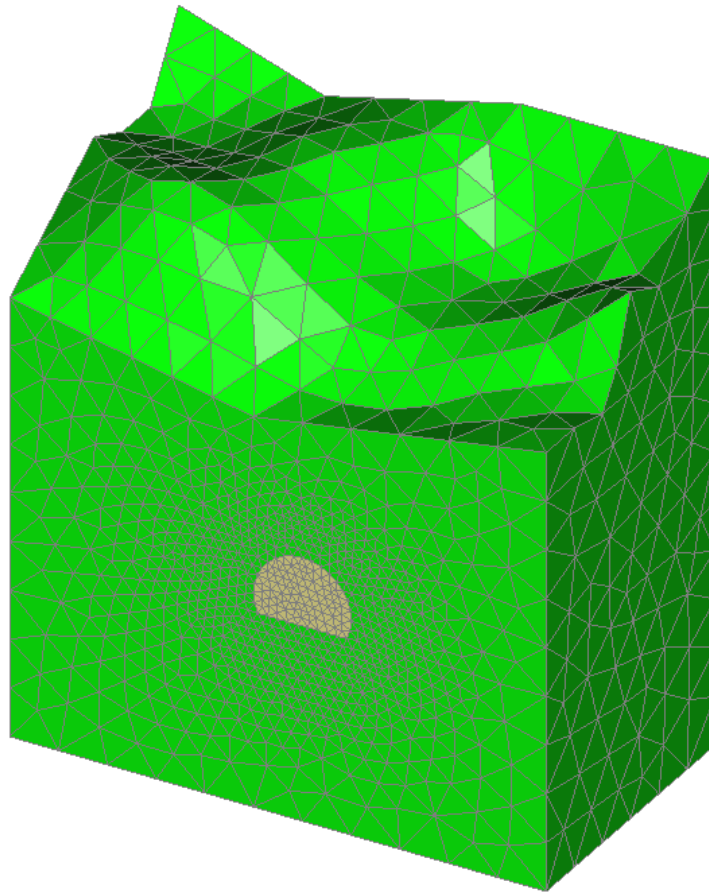


| Mesh properties | ⊓ ✕ |
|---|---|
| Update   Elements quality check | |
| ▱ **Modelled bodies** | |
| Number of bodies | 31 |
| ▱ **Mesh size** | |
| Number of groups | 166 |
| Total number of nodes | 13439 |
| Total number of elemen | 72038 |
| 1D-elements | 810 (L) |
| Surface elements | 0 (-) |
| Volume elements | 71228 (L) |
| ▷ **Mesh properties** | |
| ▱ **Elements quality check** | |
| Number of elements ent | 0 |
| Number of elements par | 0 |
| Number of very distorted | 0 |
| Number of distorted surf | 0 |

💡 User can evaluate the number of nodes and elements as well as the quality of the mesh.

1.   Open tool  *Mesh Properties*.
2.   Click **Elements quality check**.

Finally, we can generate the shell elements.

1.  Select all the surface bodies.

2.  Open the tool 📐 *Surface meshing*

    - Select "Linear interpolation" as **Interpolation type**.
    - Set "Triangle" as type of mesh.
    - Mesh creating function is **Cubic**.
    - *Apply*.
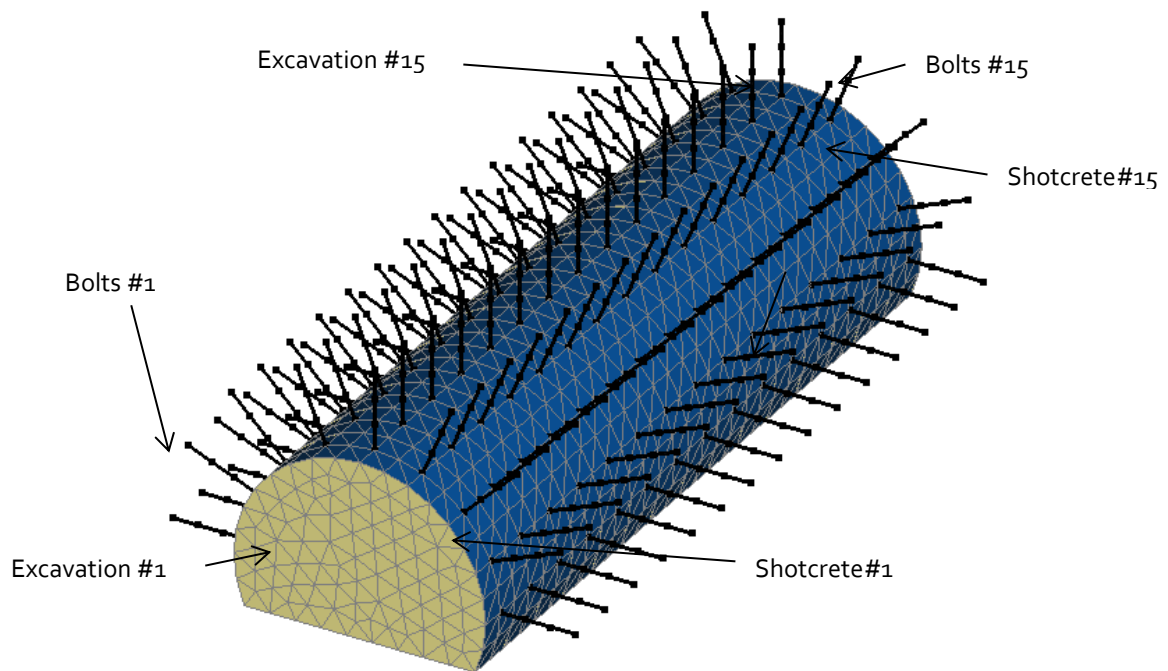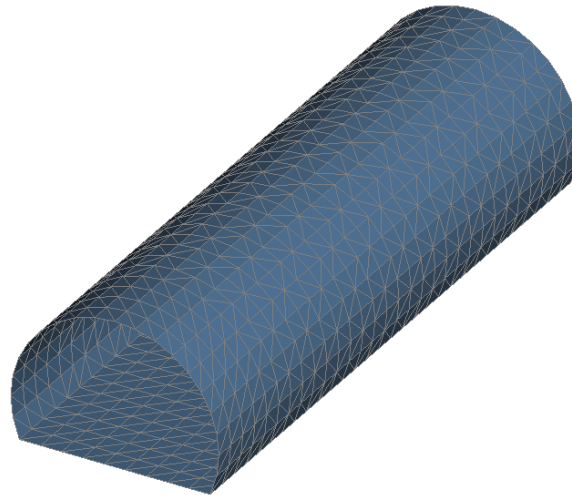
# 3. CALCULATION SETTINGS

## 3.1. Calculation properties for the initial stress field

The staged construction process requires the definition of an initial stress field before applying loads.

As the top of the soil mass is not horizontal, we cannot apply the "Ko procedure".

### 3.1.1 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 **Plane strain** as model configuration, with **Staged construction**.

5. Tick **General initial stress field** as initialisation type.

6. Click on **Validate**.

| Model type | |
|---|---|
| **Model definition** | |
| Model name | Initial stress field under gravity |
| Comment | |
| Domain | STATICS |
| Solver | MCNL |
| Solver description | Solving of a mechanical problem with non linear behaviour (material properties, interfaces, staged construction...). |
| ⦿ Staged construction ○ Initial parameters ○ Subsequent calculation | |
| Initialisation type description | Sequence of chained calculations. The stress state of the stage #(n-1) initializes the stress state of the stage #n . The displacement fields are cumulative or reset (set in "Analysis settings"). |
| Stage order | 1 |
| ○ Geostatic stresses ⦿ General initial stress field | |
| Initial stress description | The stress field initialisation is made by gravity loading of the soils either dry (total stresses) or saturated depending of the WTB position (effective stresses). Other options are offered. |

The date tree is now as illustrated below.

**Study**

Data tree   Update

- **MODELS**
  - **STATICS**
    - **[4]Initial stress field under gravity**
      - [4]Properties
      - Boundary conditions
        - **[6]Bound.Condition**
      - Loadings
        - [4]Self-weight of rock mass
      - [4]Results

### 3.1.2 Material database

Material properties for the solid bodies:

We initially define the material library of the study.

1. Go to the tab **PROPERTIES**.

2. Click on ◮ **Properties for volume bodies**.

3. Give a name for the properties set name ("Rock" for example).

4. In **Elasticity parameters**, chose "Isotropic linear elasticity" and define $\rho$, E and $\nu$.

5. In **Plasticity parameters**, chose "Mohr-Coulomb without hardening" and define c, $\varphi$ and $\psi$.

6. Click on **Validate** and **Close**.

| | $\rho$ (kg/m3) | E (MN/m²) | $\nu$ | c (MN/m²) | $\varphi$ (°) | $\psi$ (°) |
|---|---|---|---|---|---|---|
| Rock for K0 | 2400 | 800 | 0,49 | 0,25 | 30 | 0 |
| Rock | 2400 | 800 | 0,3 | 0,25 | 30 | 0 |

💡 In order to generate K0 = 1 in this rock mass, we will apply the self-weight and a Poisson's ratio equal to 0,49. Thus K0 = $\nu/(1-\nu) \approx 1$.

Material properties for the bolts:

1. Click on ▱ **Properties for 1D bodies**.

2. Give a name for the properties set name ("Bolts" for example).

3. As **Type**, chose "Bar" and define the rigidity E, and the section S.

4. Click on **Validate** and **Close**.

| Name of the body | Type | E (MPa) | S (m²) |
|---|---|---|---|
| Bolts | Bar | 200 000 | $5.10^{-4}$ |
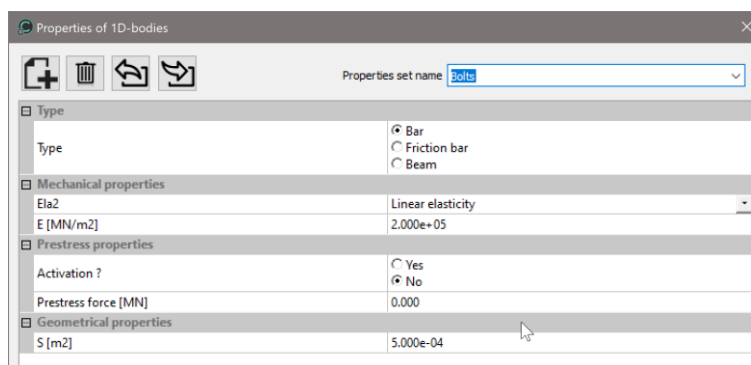


Material properties for the shotcrete:

1. Click on ◰ **Properties for surface bodies**.

2. Give a name for the properties set name ("Shotcrete" for example).

3. Set the elasticity parameters and the thickness.

4. Click on *Validate* and *Close*.
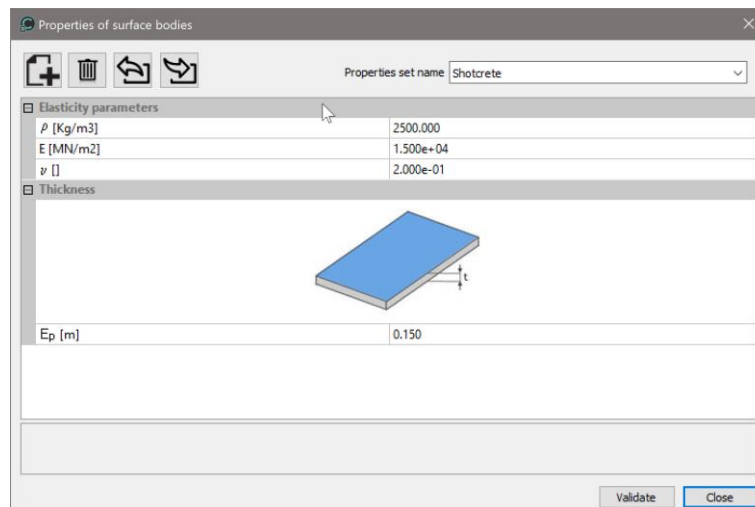
| | ρ (kg/m3) | E (MN/m²) | ν | Thickness (m) |
|---|---|---|---|---|
| Shotcrete | 2500 | 15000 | 0,2 | 0,15 |



Assignment of properties sets:

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

1. Click on [icon] *Assign properties* tool.

2. On the left side, a new toolbox is displayed. Click on [icon] *Properties for volume bodies*.

3. Select all the volume bodies of the model window and the set of parameter "Rock for Ko" in the list.
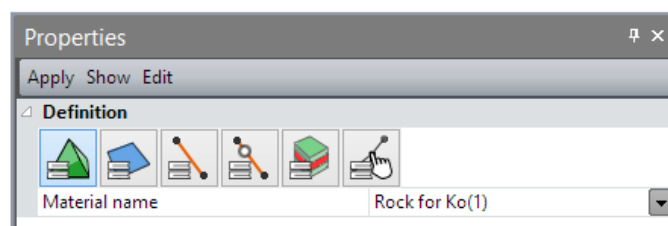
4. *Apply*.



5. On the left side, click on [icon] *Properties for surface bodies*.

6. Select all the surface bodies of the model window and the set of parameter "Shotcrete" in the list.

7. *Apply*.

8. On the left side, click on [icon] *Properties for 1D bodies*.

9. Select all the 1D bodies of the model window and the set of parameter "Bolts" in the list.

10. *Apply*.

<u>Active/inactive bodies</u>

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

1.  Select the 1D and surface bodies representing the **Bolts** and **Shotcrete**.

2.  Click on  *Activate/deactivate bodies*.

3.  In the grid, tick "Inactive".

4.  *Apply*. The 1D and surface bodies are now marked with the neutral colour. Only the rock mass remains activated.

<u>Boundary conditions:</u>

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.

<u>Loading set:</u>

1.  Go to the tab *LOADS*.

2.  On the toolbar, activate *Gravity forces*  .

3.  Select all the bodies.

4.  Click on *Apply*.

💡 Optional. It is possible to modify the default name assigned to the loading set. Press [F2]; enter **Self-weight** for example.

<u>Calculation parameters:</u>

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**.

💡 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 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:
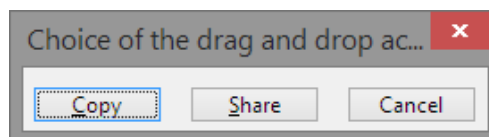
- In properties as groups are activated,

- In loads in order to activate the excavation forces.
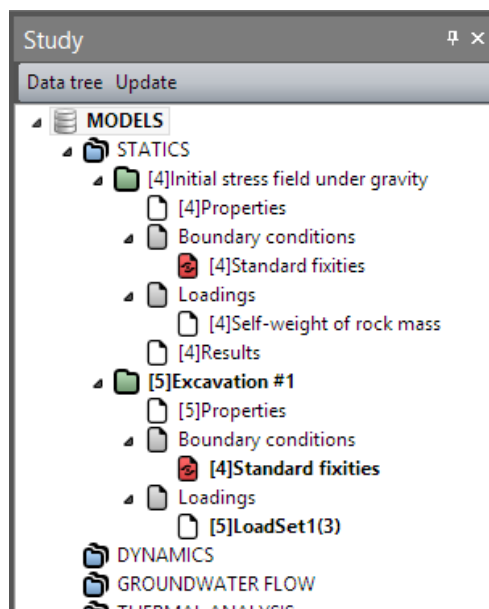
Model definition

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

We can now copy the definition sets from previous model.

1. Select the *Properties* set of the model **Initial stress field**.
2. By drag and drop, place it on the new *Properties* set of model **Excavation #1**.
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***.

Choice of the drag and drop ac...    ✕

[ Copy ]    [ Share ]    [ Cancel ]

The date tree is now as illustrated below.

Study                                    📌 ✕

Data tree  Update

▲ 🗏 **MODELS**
  ▲ 🗀 STATICS
    ▲ 🗀 [4]Initial stress field under gravity
      📄 [4]Properties
     ▲ 📄 Boundary conditions
       🔴 [4]Standard fixities
     ▲ 📄 Loadings
      📄 [4]Self-weight of rock mass
     📄 [4]Results
    ▲ 🗀 [5]Excavation #1
     📄 [5]Properties
     ▲ 📄 Boundary conditions
      🔴 **[4]Standard fixities**
     ▲ 📄 Loadings
      📄 [5]LoadSet1(3)
  🗀 DYNAMICS
  🗀 GROUNDWATER FLOW
  🗀 THERMAL ANALYSIS

## Assignment of properties set

We modify here the properties of the Rock mass with a Poisson's ration equal to 0.3.

1. Go to the tab **PROPERTIES**.

2. Click on ▭ **Assign properties** tool.

3. On the left side, a new toolbox is displayed. Click on ▱ **Properties for volume bodies**.

4. Select all the volume bodies of the model window and the set of parameter "Rock" in the list.

5. **Apply**.
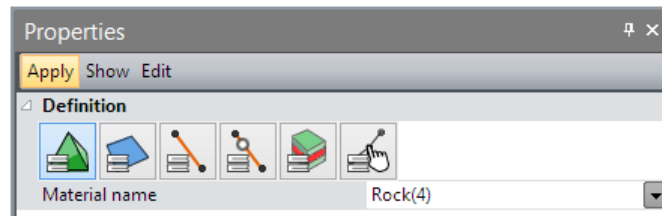
## Active/inactive elements

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

1. Select the volume body representing the **Section #1**.

2. Click on ▭▮ **Activate/deactivate bodies**.

3. In the grid, tick "Inactive".

4. **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

We generate the new loading set.

The load set consists here to define the excavation forces resulting from the remove of the rock mass.

1. In the model tree, activate the default *Loading set* of *Loadings* 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**.
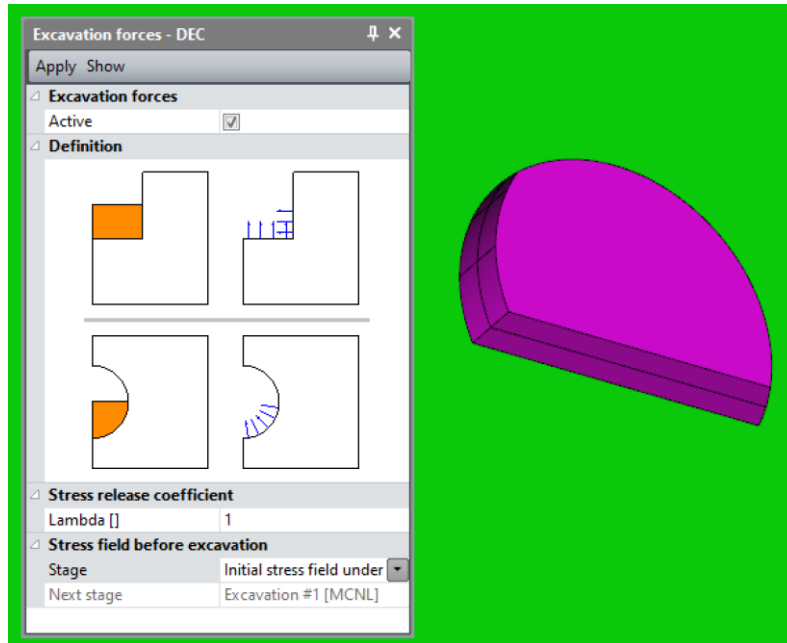
Calculation parameters:

There is no change here.

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

In this stage, the excavation front is going deeper, the shotcrete and the bolts are activated to stabilize the soil mass.

Model definition:

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

We can now copy the definition sets from previous model.

1. Select the *Properties* set of the model **Excavation #1**.
2. By drag and drop, place it on the new *Properties* set of model **Excavation #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***.
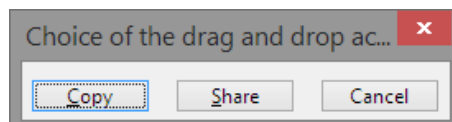


Active/inactive elements

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

1. Go to the tab ***PROPERTIES***.
2. Select the volume body representing the **Section #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.

Installation of the shotcrete and bolts requires setting at active state the **Shotcrete #1**.and the **Bolts #1**.

1. Select the surface body representing the **Shotcrete #1** and the 1D bodies for **Bolts #1**.
2. Click on ▯▶▮ **Activate/deactivate bodies**.
3. In the grid, tick "Active".
4. **Apply**. These bodies are now marked with their initial colour.

💡 For ease of the selection, user will hide the rock mass and use the selection toolbar and especially the **Selection options** tool ▱.
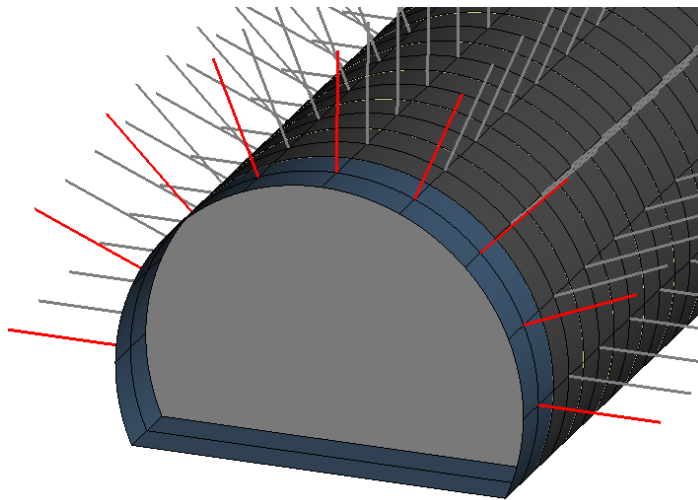


Boundary conditions:

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 remove of the rock mass and to activate the self-weight of the shotcrete (considering that the bolts do not weigh).

1. In the model tree, activate the default *Loading set* of the model **Excavation #2**.
2. Press [F2]; enter **Excavation forces #2** 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 **Excavation #1** model as stress field origin.
5. Click on **Apply**.
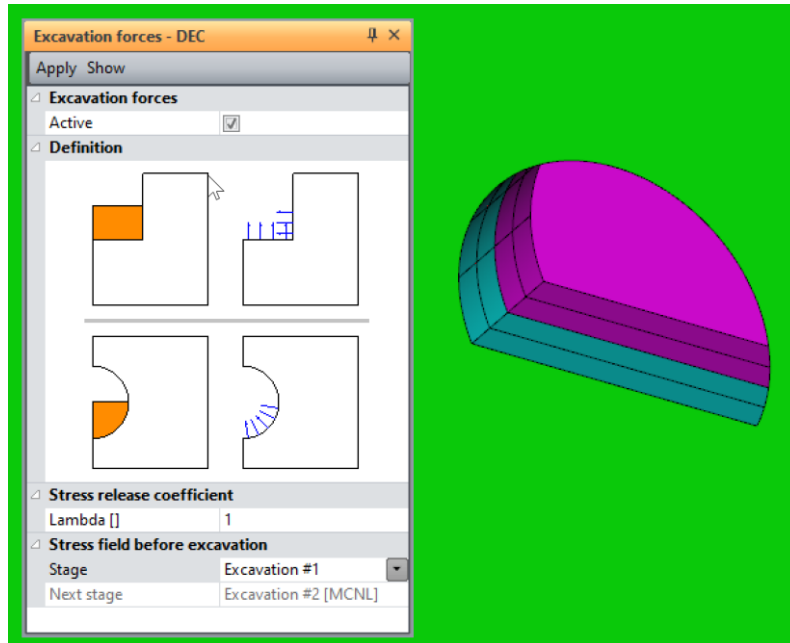
Loading set: Activation of the lining #1

1.  In the model tree, right click on *Loadings* and click **Add a loading set**.
2.  Press [F2]; enter **Self-weight of concrete #1** as a name for this new loading set.
3.  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 **Shotcrete #1** body.
    -   *Apply*.

Calculation parameters

There is no change here.

## 3.4. *Calculation properties for the analysis of excavation #3*

In this stage, the excavation front is going deeper, the shotcrete and the bolts are activated to stabilize the soil mass.

Model definition:

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

We can now copy the definition sets from previous model.

1.  Select the *Properties* set of the model **Excavation #2**.
2.  By drag and drop, place it on the new *Properties* set of model **Excavation #3**.
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**.
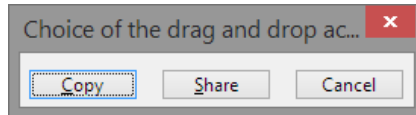
### Active/inactive elements

The excavation of the full section leads to set at inactive state the **Section #3**.

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

Installation of the the shotcrete and bolts requires setting at active state the **Shotcrete #2**.and the **Bolts #2**.

### Boundary conditions:

There is no change in the boundary conditions.

### Loading set: Excavation forces #3

The load set consists here to define the excavation forces resulting from the remove of the rock mass and to activate the self-weight of the shotcrete (considering that the bolts do not weigh).

1. In the model tree, activate the default *Loading set* of the model **Excavation #3**.
2. Press [F2]; enter **Excavation forces #3** 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 **Excavation #2** model as stress field origin.
5. Click on **Apply**.
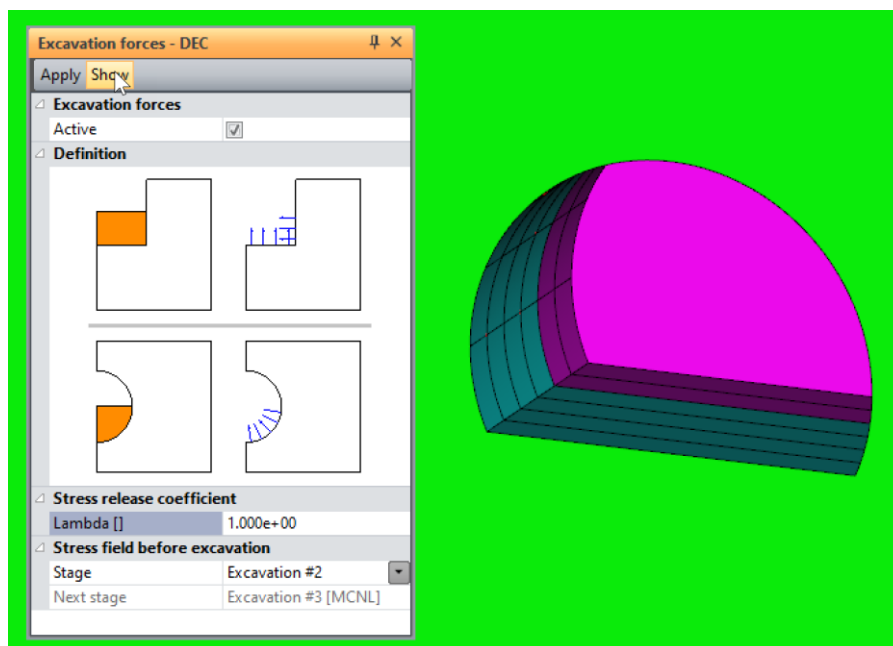
Loading set: Activation of the lining #2

1. In the model tree, right click on *Loadings* and click **Add a loading set**.
2. Press [F2]; enter **Self-weight of concrete #2** as a name for this new loading set.
3. 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 **Shotcrete #2** body.
   - *Apply*.

Calculation parameters
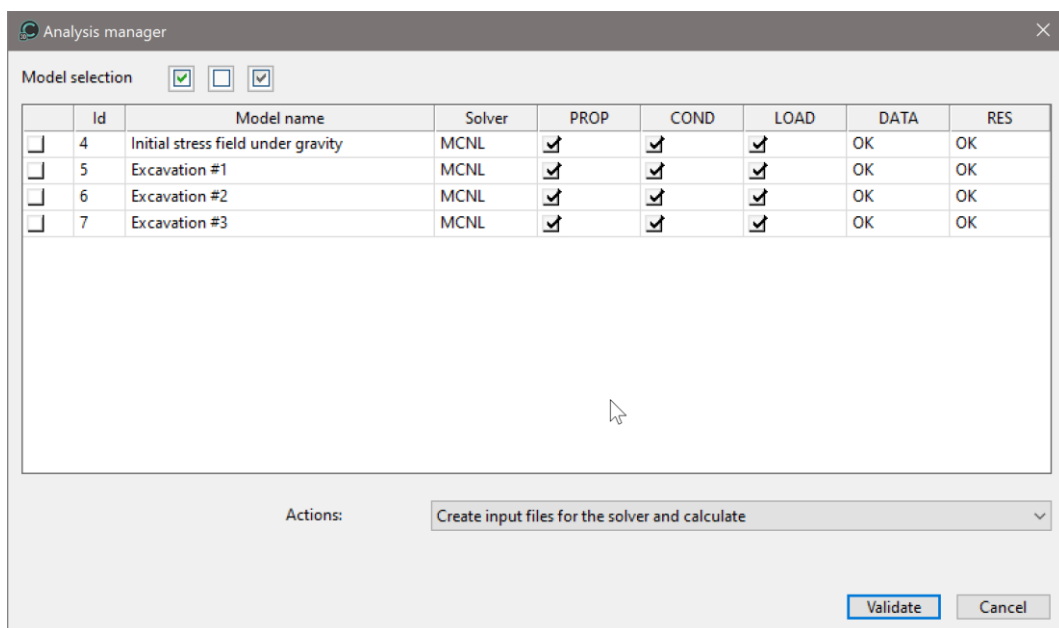
There is no change here.

# 4. SOLVE

User can easily understand that the process defined for modelling excavations #2 and #3 can be reproduced in the same way to complete the tunnel. In the scope of the tutorial, we finish here and run the calculations.

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 4 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.

| | Id | Model name | Solver | PROP | COND | LOAD | DATA | RES |
|---|---|---|---|---|---|---|---|---|
| ☐ | 4 | Initial stress field under gravity | MCNL | ✔ | ✔ | ✔ | OK | OK |
| ☐ | 5 | Excavation #1 | MCNL | ✔ | ✔ | ✔ | OK | OK |
| ☐ | 6 | Excavation #2 | MCNL | ✔ | ✔ | ✔ | OK | OK |
| ☐ | 7 | Excavation #3 | MCNL | ✔ | ✔ | ✔ | OK | OK |

Actions: Create input files for the solver and calculate

Validate    Cancel

💡 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. FINITE ELEMENT RESULTS

We display the vertical stresses inside the model after analysis of **Excavation #3**.

1. Activate the tab *RESULTS*

2. Click on ⬛ *Type of results to display*.
   - Select **Principal stress S1** 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*.
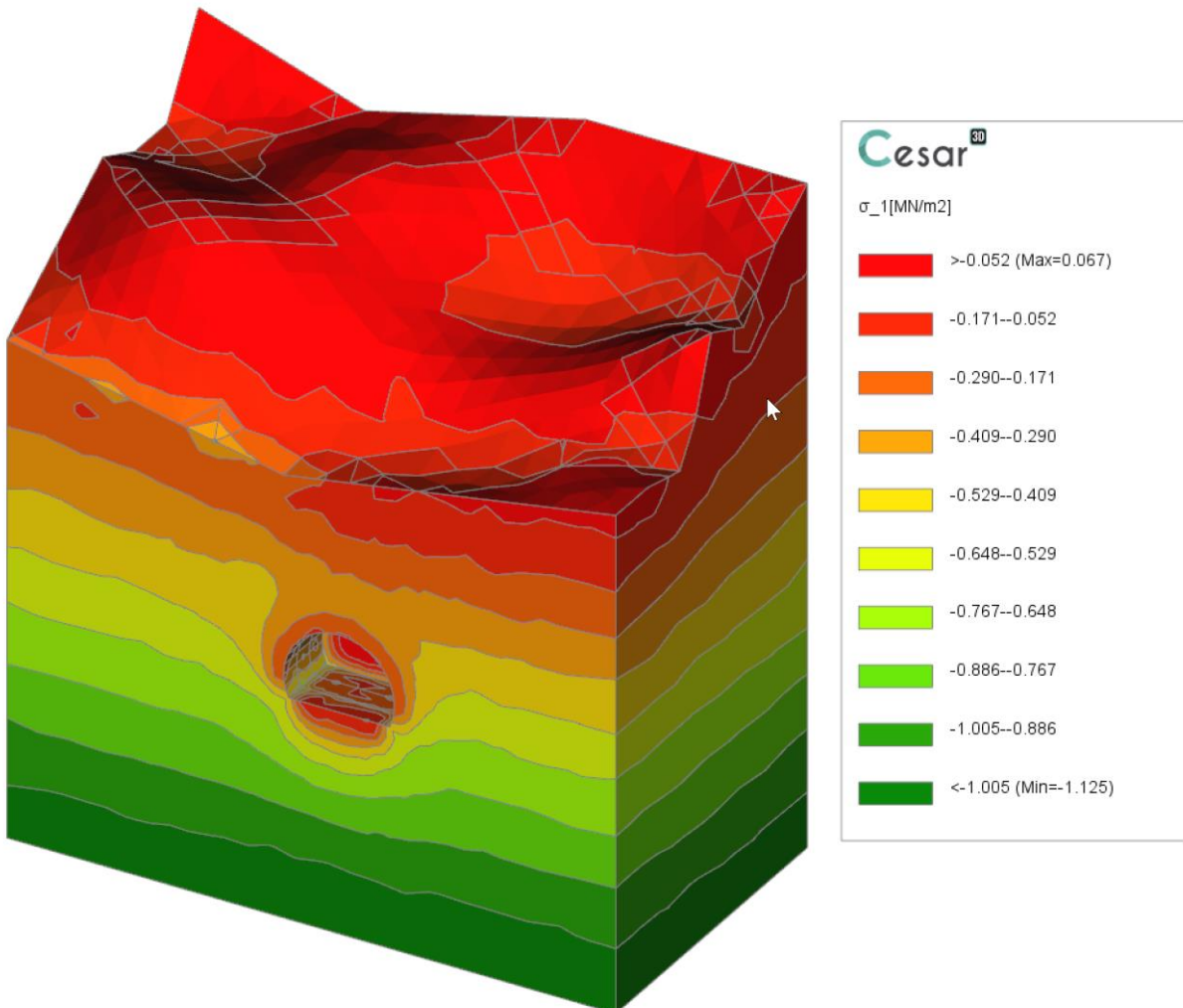
We display now the total displacements and investigate inside the model with the tool "Cross section plane".

Using S**election options** ☑️, user will select only volume bodies.

1. Click on 👁 🌈 ***Type of results to display***.
   - Select **Total displacement** in Contour option,
   - ***Apply***.

2. Click on 🔲 ***Cross section plane***.
   - Tick **Active**, **Clipping cross section** and **Cross section**,
   - Set the position to the point (32; 0; 100),
   - Set the direction of the cut plane following the vector (0; 1; 0).
   - ***Apply***.
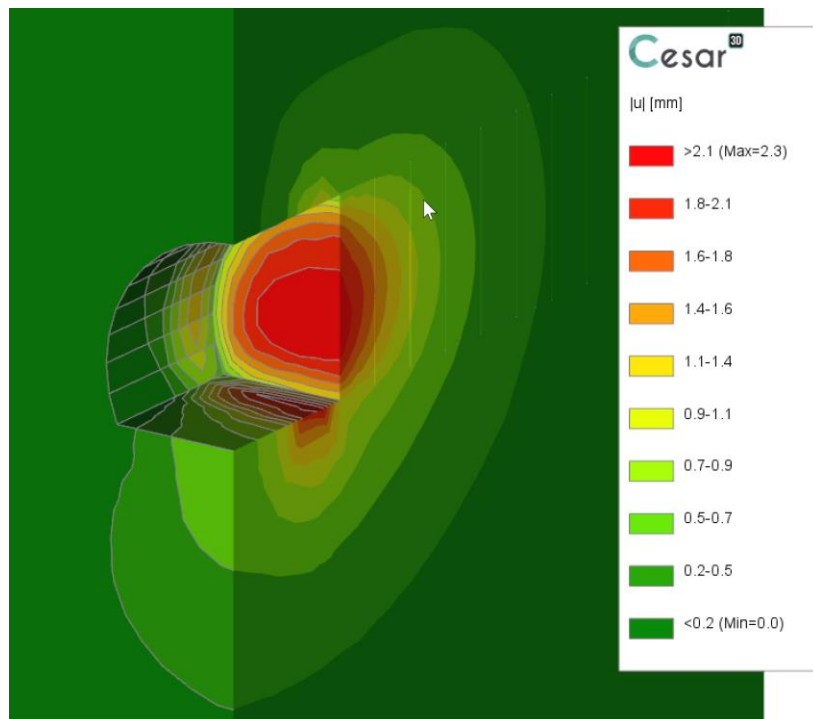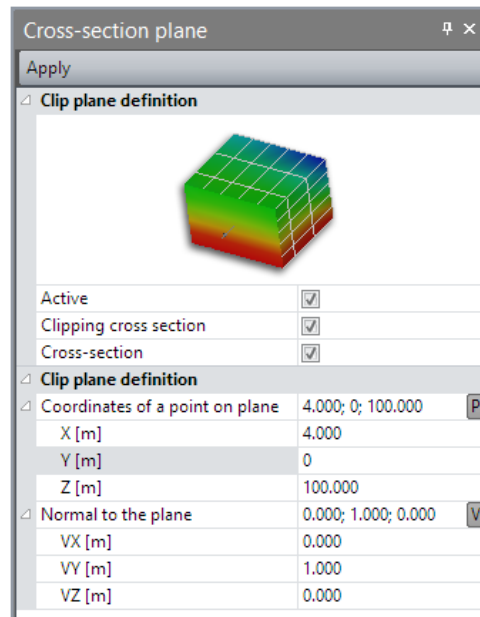
Finally, we show how to get accurate information in shell and bolts.

1. Select bodies **Bolts #1** and **Bolts #2**. For this selection, select **1D bodies** in *Selection Options* ⬚↖

2. Click on 🌈 *Type of results to display*.
   - Activate **Beam results**,
   - *Apply*.
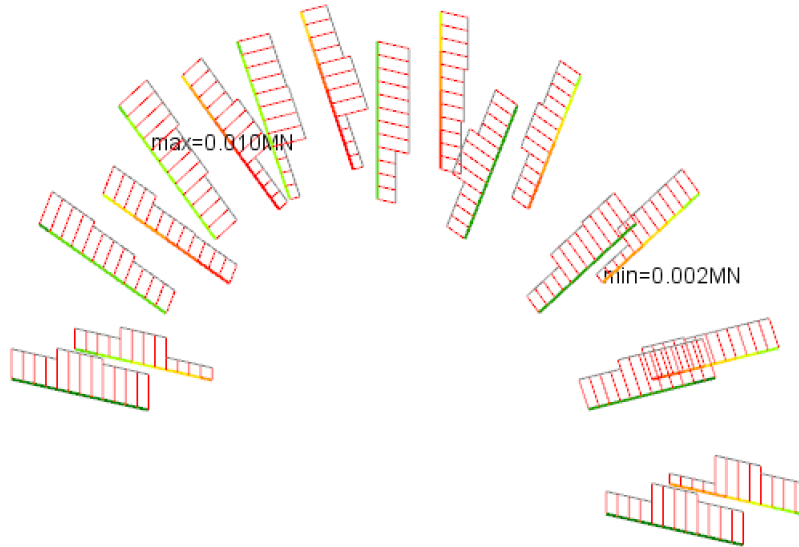


1. Select bodies **Shocrete #1** and **Shocrete #2**. For this selection, select **Surface bodies** in *Selection Options* ⬚↖:

2. Click on 🌈 *Type of results to display*.
   - Activate **Shell forces Nz** in Contour plots.
   - *Apply*.
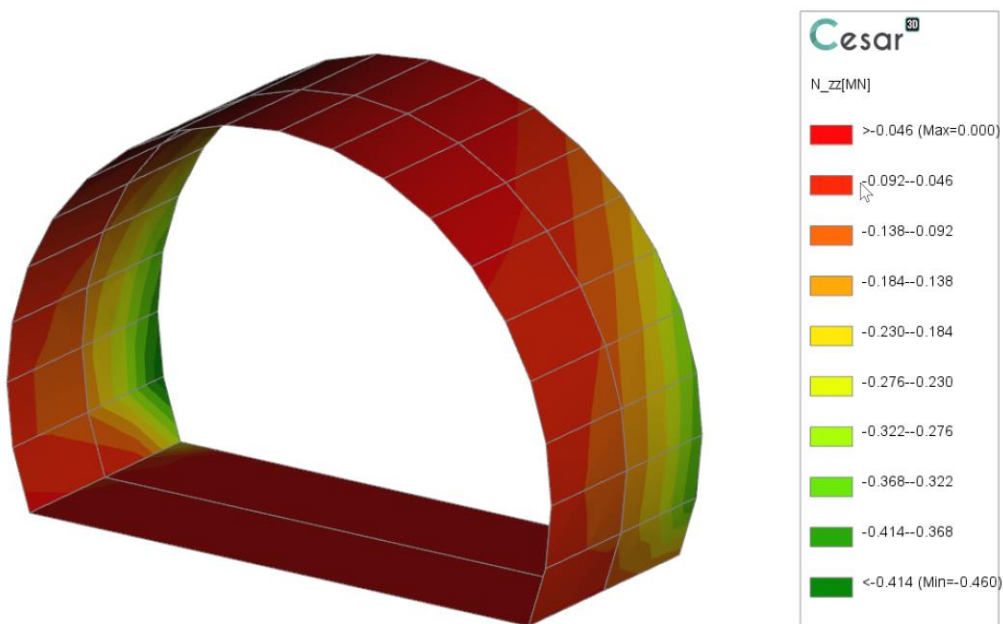
Edited by:



8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tel.: +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com