



Tutorial 3g.05

Shallow foundation near slopes

| Ref: CESAR-TUT(3g.05)-v2021.0.1-EN

1. INTRODUCTION

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.

2. PREVIEW

A bridge pier has to be constructed in the corner of a 2 slopes embankment. Modelled as a footing, it is important to assess its load capacity.

This tutorial will guide the user through the 3D modelling and the analysis of the limit pressure.

2.1. Tutorial objectives

- Use of 3D meshing (tetrahedron mesher).
- Use of safety factor reduction procedures.
- Use of 3D post-processing tools.

2.2. Problem specifications

The foundation is a shallow one (15m long, 6m wide and 1m thick). It is placed at 2,5m from the slope edge.

The embankment is a 2-slopes one. It lays on a uniform terrain, 12m thick.

Materials

	γ (kN/m ³)	E (MPa)	ν	c (kPa)	ϕ (°)	ψ (°)
Embankment	19	30	0.3	5	25	5
Terrain	20	90	0.3	28	25	5
Footing	27.5	35000	0.2	-	-	-

Analysis steps

1. Build the 3D mesh using the tetrahedron mesh generator.
2. 1st calculation: Initial stress field (self-weight of the soil mass). We do not model the construction process of the embankment.
3. 2nd calculation: Uniform pressure loading on the footing. Use of the Safety factor procedure to find the limit pressure.

Geometry

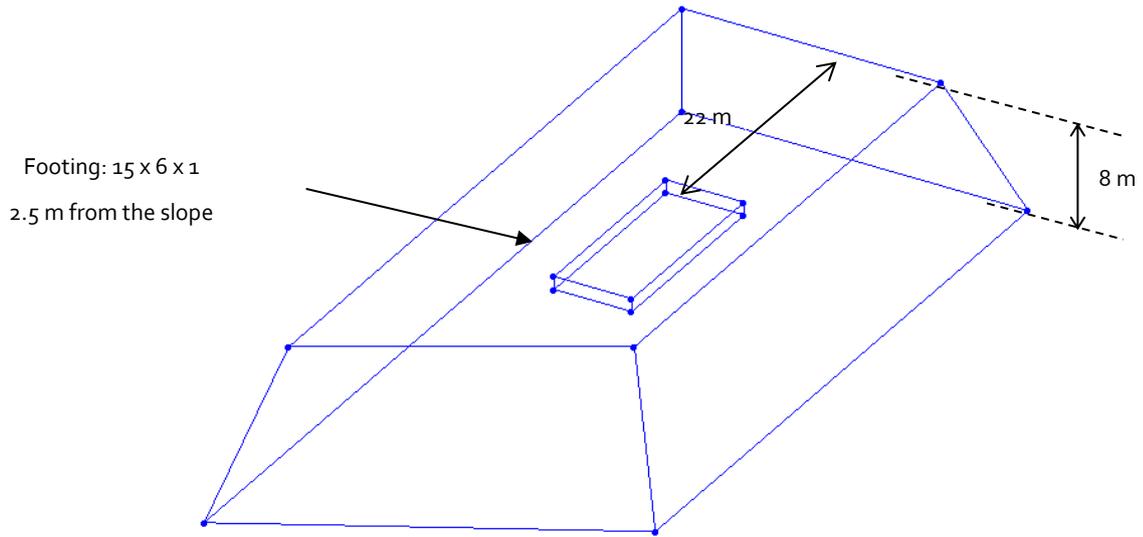


Figure 1: Geometry definition of the two-way slope – 3D view

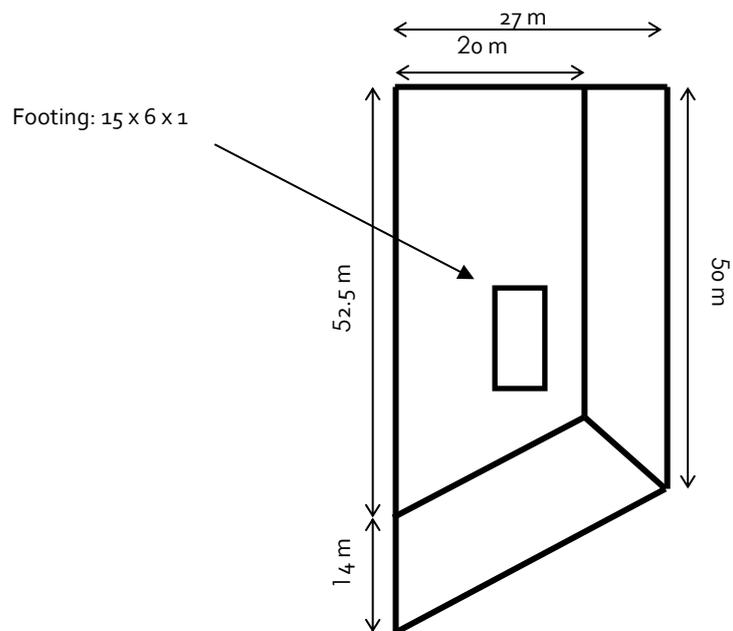


Figure 2: Geometry definition of the two-way slope – Top view

3. 3D MODELLING

3.1. General settings

1. Run CESAR 3D.

Preferences

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 o.o.
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.

3.2. 3D Meshing

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.

We will proceed step by step and use several of the CAD tools proposed in CESAR-LCPC, in order to generate the volumes, support of the mesh.

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

Drawing of the geometry:

The sequence proposed here is:

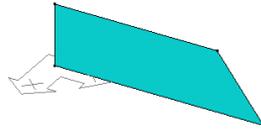
- Drawing of the main volume of the embankment by extrusion,
- Intersection of the created volume with a plane in order to generate the second slope,
- Definition of volumes of footing and terrain.

We start with the edges of the base of the embankment.

1. Activate . The point definition dialog is displayed.
2. Enter '0 ; 0 ; 0' as X, Y and Z, and press **Apply** button.
3. Tick **Linked points** to generate the segments between the consecutive generated points.
4. Enter '0 ; 27 ; 0' and press **Apply** button.
5. Enter '0 ; 21 ; 8' and press **Apply** button.
6. Enter '0 ; 0 ; 8' and press **Apply** button.
7. Enter '0 ; 0 ; 0' and press **Apply** button. The base of the embankment is drawn.

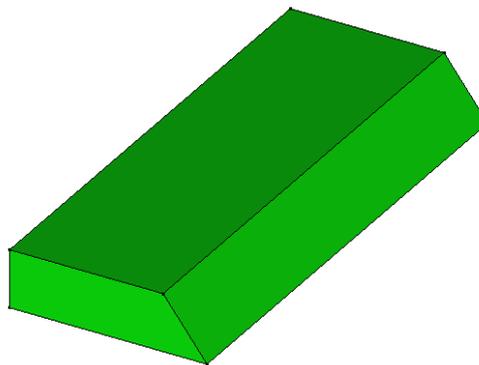
We can now define the associated surface.

1. Select the previously defined edges.
2. Activate  **Plane Surface**.
3. **Apply**.



We can now extrude it.

1. Select the previously defined surface.
2. Activate  **Extrusion**.
 - Operation type: *Translation*.
 - Extrusion type: *Generation of volume bodies*.
 - Number of operation: 1.
 - Vector (Vx, Vy, Vz): **66.5 ; 0 ; 0**
 - Remove surface bodies (the support one is useless after operation).
3. **Apply**.

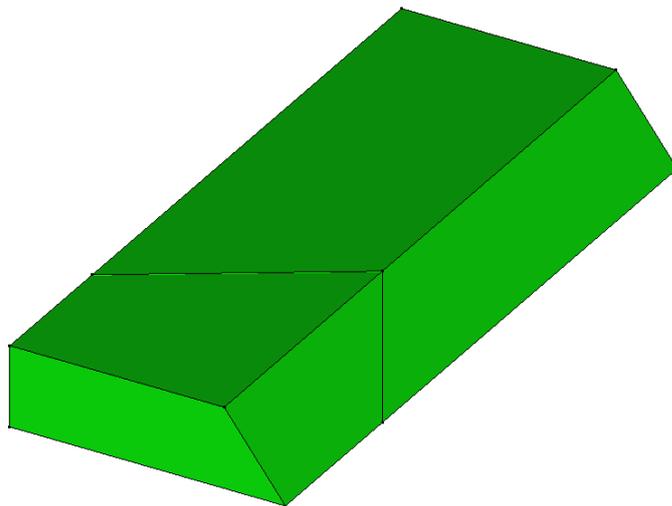
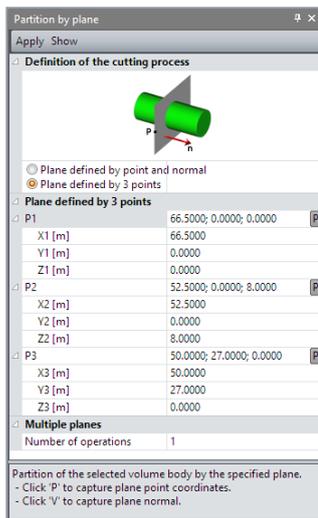


The second slope is the result of the intersection of this volume with a plane. The plane is defined by 3 points that we are defining now.

4. Activate  **Points**.
5. Untick **Linked points** as we do not need to generate the segments between the consecutive generated points.
6. Enter '**52.5 ; 0 ; 8**' as X, Y and Z, and press **Apply** button. Point #1 is created.
7. Enter '**50 ; 27 ; 0**' and press **Apply** button. Point #2 is created.

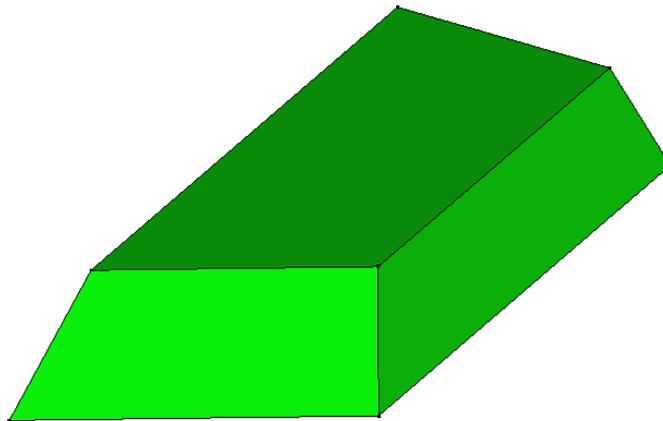
The Point #3 already exists as part of the extruded volume. Its coordinates are '**66.5 ; 0 ; 0**'.

1. Activate  **Partition by plane**.
 - Select **Plane defined by 3 point**.
 - Interactively define the 1st point. Activate the tool **P**. The grid box gets blue. Click on Point #1. Its coordinates are updated in the grid: '**52.5 ; 0 ; 8**'
 - Interactively define the 2nd point. Activate the tool **P**. The grid box gets blue. Click on Point #2. Its coordinates are updated in the grid: '**50 ; 27 ; 0**'
 - Interactively define the 3rd point. Activate the tool **P**. The grid box gets blue. Click on Point #3. Its coordinates are updated in the grid: '**66.5 ; 0 ; 0**'
2. Select the volume body.
3. **Apply**. The intersection is done.



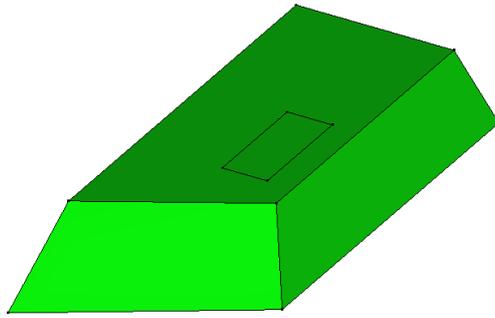
We delete the useless volume.

1. On the selection toolbar, activate **Select bodies**.
2. Ensure that *Volume body* is activated in **Selection options**.
3. Select the volume body. Activate **Delete**.
 - Tick **Delete the body edges** so that unnecessary edges and points are also deleted.
 - **Apply**.



We implement now the position of the footing.

1. Activate **Points**.
 1. Enter '**22 ; 11.5 ; 8**' as X, Y and Z, and press **Apply** button.
 2. Tick **Linked points** to generate the segments between the consecutive generated points.
 3. Enter '**22 ; 17.5 ; 8**' and press **Apply** button.
 4. Enter '**37 ; 17.5 ; 8**' and press **Apply** button.
 5. Enter '**37 ; 11.5 ; 8**' and press **Apply** button.
 6. Enter '**22 ; 11.5 ; 8**' and press **Apply** button. This closes the rectangle of the footing print on the embankment.

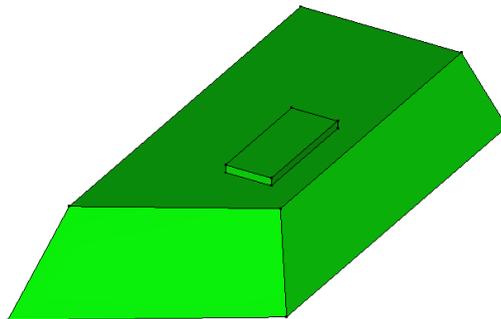


We can now define the associated surface.

1. Select the edges previously defined.
2. Activate  **Plane Surface**.
3. **Apply**.

The volume of the footing (1m thick) is now created by extruding this face.

1. Select the previously defined surface.
2. Activate  **Extrusion**.
 - Operation type: *Translation*.
 - Extrusion type: *Generation of volume bodies*.
 - Number of operation: 1.
 - Vector (Vx, Vy, Vz): 0 ; 0 ; 1
3. **Apply**.



We can now draw the substratum volume body.

In order to ease the visualization and selection, we start to hide existing parts of the model.

1. On the selection toolbar, activate  **Select bodies**.
2. Ensure that *Volume body* is activated in  **Selection options**.
3. Activate  **Hide selection**.

We proceed the volume body generation.

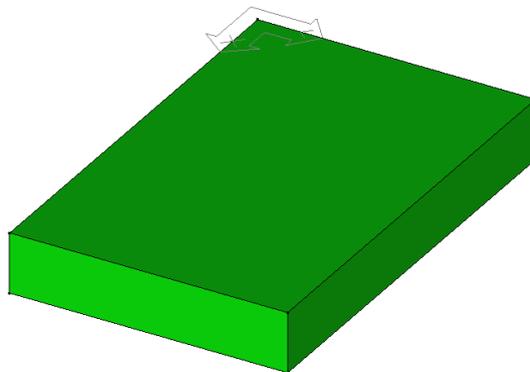
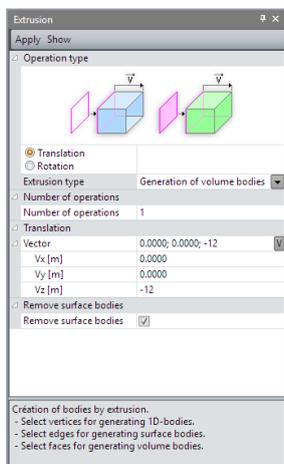
1. Activate  **Points**.
2. Enter '0 ; 0 ; 0' as X, Y and Z, and press **Apply** button.
3. Tick **Linked points** to generate the segments between the consecutive generated points.
4. Enter '0 ; 55 ; 0' as X, Y and Z, and press **Apply** button.
5. Enter '85 ; 55 ; 0' and press **Apply** button.
6. Enter '85 ; 0 ; 0' and press **Apply** button.
7. Enter '0 ; 0 ; 0' as X, Y and Z, and press **Apply** button.

We can now define the associated surface.

1. Select the edges previously defined.
2. Activate  **Plane Surface**.
3. **Apply**.

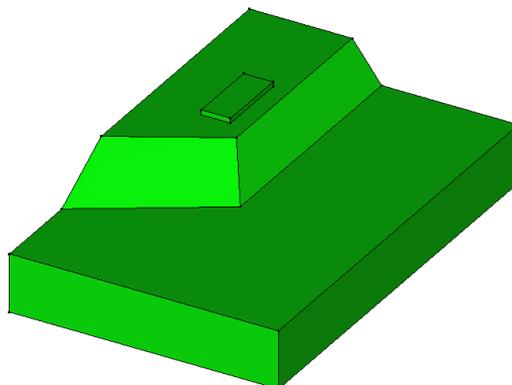
The volume of the footing being a regular one (12m thick) we can create it by extruding this face.

1. Select the previously defined surface. Ensure that  **Select bodies** tool is activated on the selection toolbar, and *Surface body* is activated in  **Selection options**.
2. Activate  **Extrusion**.
 - Operation type: *Translation*.
 - Extrusion type: *Generation of volume bodies*.
 - Number of operation: 1.
 - Vector (Vx, Vy, Vz): 0 ; 0 ; -12
3. **Apply**.



We can now display the 3 volumes bodies, parts of our model.

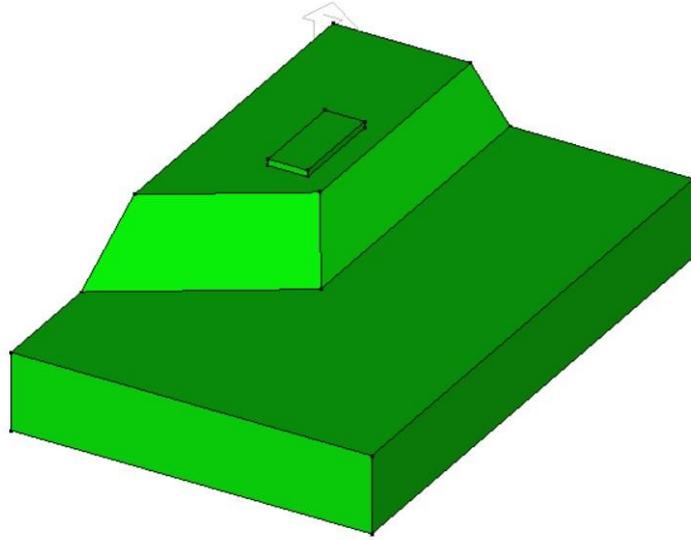
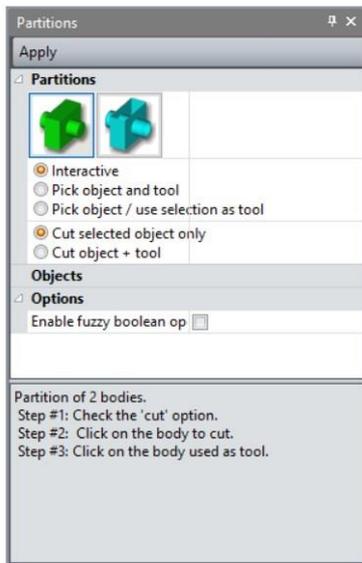
1. Select the volume bodies. Ensure that  **Select bodies** tool is activated on the selection toolbar, and *Volume body* is activated in  **Selection options**.
2. Activate  **Show all**.



Volumes intersection

In previous steps, we defined 3 volume bodies. At current state, these volumes are independent. We will now link them.

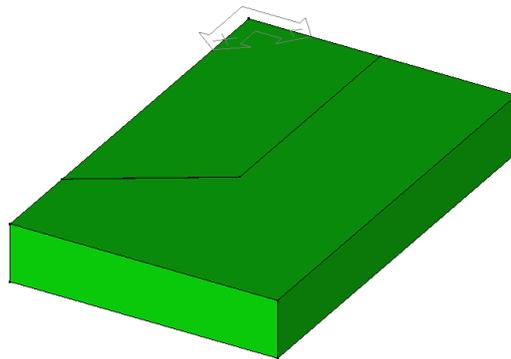
1. Click on  **Bodies intersection**
 - Activate **Interactive** option and **Cut selected object only** option;
 - With the mouse click on the footing, as body #1, then click on the embankment, as body #2.
 - Repeat operation with the embankment, as body #1, then click on and the substratum, as body #2.



We can check the operation on the substratum body.

2. On the selection toolbar, activate  **Select bodies**.
3. Ensure that *Volume body* is activated in  **Selection options**.
4. Select the embankment body. Activate  **Show selection**.

The print of the embankment on the substratum is displayed.



Mesh density

Mesh refinement is important nearby the areas where high level of strains is expected. This will lead to more accurate results. In the case of slope stability analysis with c-phi reduction, the results are highly influenced by this density of mesh.

In the present tutorial, we will apply a density that is suitable for the analysis and that will lead to reasonable computation times. We will use a progressive density definition to generate a progressive evolution of size from small segments in high strains areas, to large segments on the boundary edges.

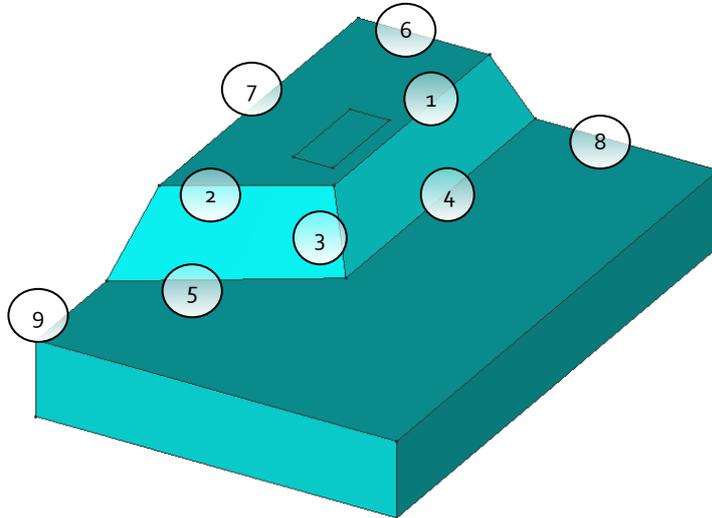


Figure 3: Sketch of the numbering of edges

We start with the mesh density of the embankment and substratum bodies.

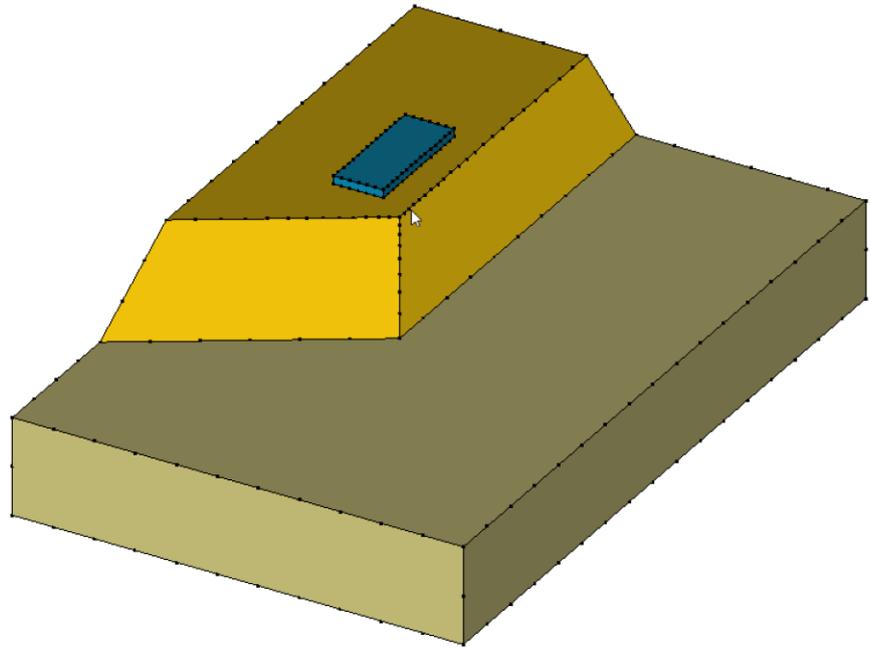
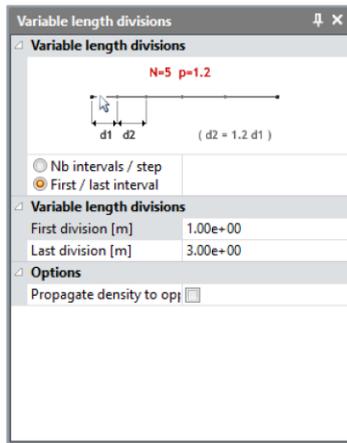
1. Go to the tab **MESH** on the project flow bar to start the definition of divisions along lines.
2. Select all the external edges of the embankment and substratum bodies.
3. Click on  **Fixed length density** to divide these segments with a fixed length. Set it to **5m**. Click on **Apply**.

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

1. Click on  **Progressive density** to divide the segment with a variation of lengths.
 - Tick **First division** and **Last division** to define the method.
 - Enter **1 m** as First division and **3 m** as **Last division**.
2. Click on the edge #1 near the 2 slopes intersection.

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

3. Click on the edge #2 near the 2 slopes intersection.
4. Click on the edge #3 near the 2 slopes intersection.
5. Click on  **Progressive density** to divide the segment with a variation of lengths.
 - Tick **First division** and **Last division** to define the method.
 - Enter **3 m** as First division and **5 m** as **Last division**.
6. Click on the edge #4 near the 2 slopes intersection.
7. Click on the edge #5 near the 2 slopes intersection.



Now we define the mesh density for the footing.

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

Meshing:

1. Go to tab **MESH**.
2. Activate  **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 0,9.
 - Mesh creating function is **Quadratic**.
 - **Apply**.

 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 interpolation = coarse, cubic interpolation = dense).

Now that the mesh is completed, we can check it.

1. Activate  **Mesh properties**.
2. The toolbox displays the number of nodes and elements and the type of elements.
3. Click on **Elements quality check**. The toolbox is updated. If elements are deformed with wrong Jacobian, they appear in the toolbox.

 Note that the number of nodes and elements of the tutorial model is 3490 and 17 241 respectively.

Bodies edition:

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

1. On the **Selection Toolbar**, activate **Select bodies**.
2. Activate  **Body properties**.
3. Right-click on the body corresponding to the slope. Enter *Slope* as name. Specify a color. Click on **Apply**.
4. Right-click on the body corresponding to the footing. Enter *Footing* as name. Specify a color. Click on **Apply**.
5. Right-click on the body corresponding to the terrain. Enter *Terrain* as name. Specify a color. Click on **Apply**.

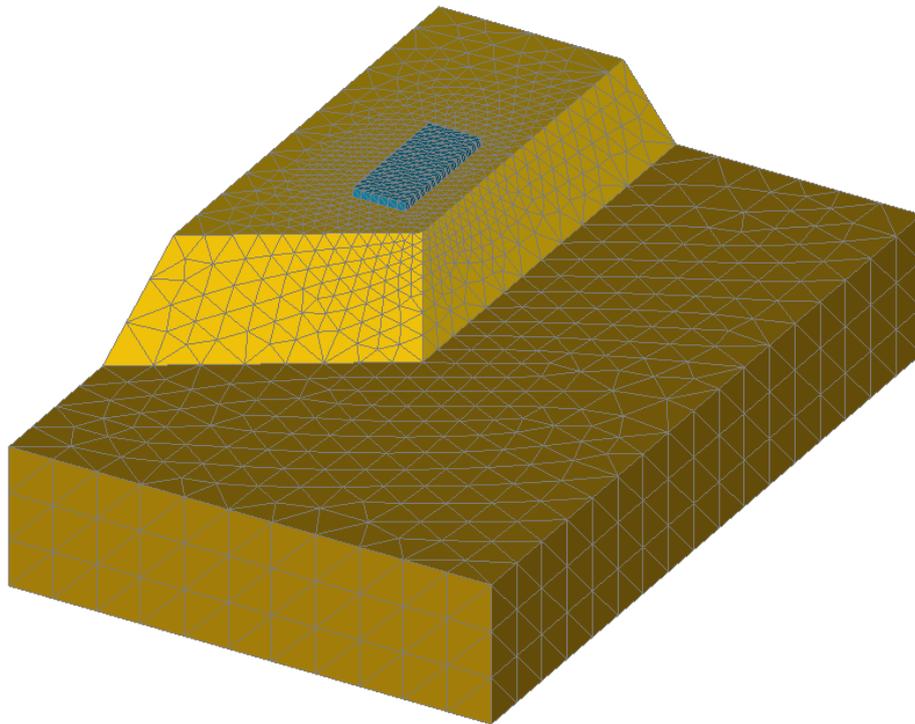


Figure 4: View of the complete 3D mesh

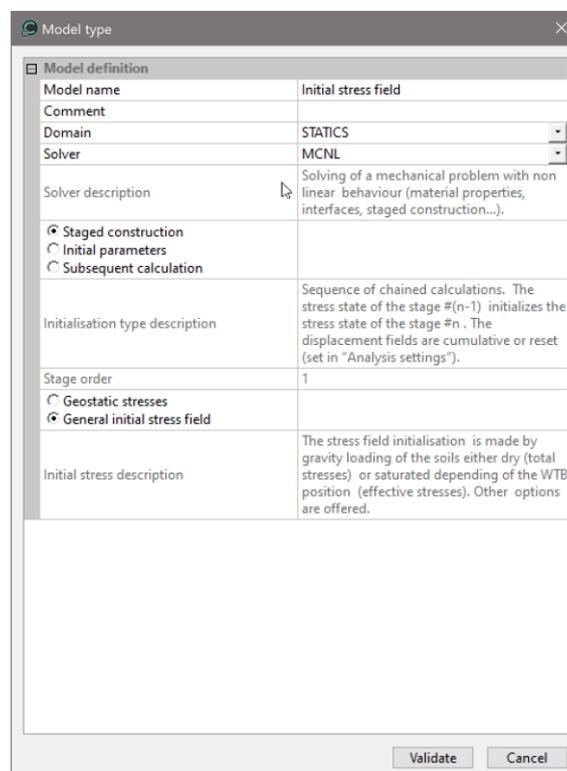
4. CALCULATION SETTINGS

4.1. Calculation properties for the initial stress field

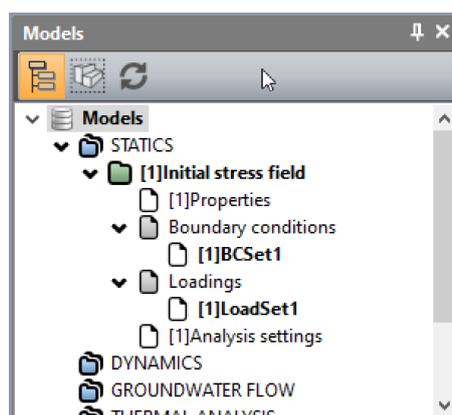
As we do not model the embankment construction, the initial stress field is initialized with the "Self-weight procedure" (cf. manual "Getting started with CESAR-LCPC", Chapter "Initial stress field").

4.1.1 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 **Staged construction**.
5. Tick **General initial stress field** as initialization type.
6. Click on **Validate**.



The date tree is now as illustrated below.



4.1.2 Material database

Material properties for the solid bodies:

We initially define the material library of the study.

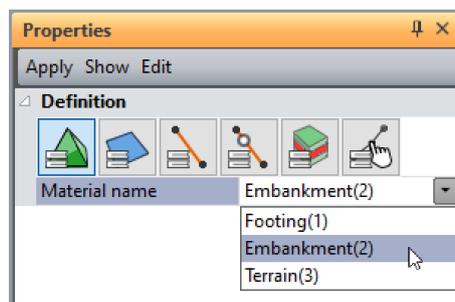
1. Go to tab **PROPERTIES**.
2. Activate  **Properties for volume bodies**.
3. Give a name for the properties set name (*Footing* for example).
 - In **Elasticity parameters**, select "Isotropic linear elasticity" and define ρ , E and ν .
 - Click on **Validate**.
4. Click  and add a property set for the *Embankment*.
 - In **Elasticity parameters**, select "Isotropic linear elasticity" and define ρ , E and ν .
 - In **Plasticity parameters**, select "Mohr-Coulomb without hardening" and define c , ϕ and ψ .
 - Click on **Validate**.
5. Repeat the previous operation for the *Terrain* properties,
6. Click on **Validate** and **Close**.

	ρ (kg/m ³)	E (MN/m ²)	ν	c (MN/m ²)	ϕ (°)	ψ (°)
Footing	2 750	35 000	0.2	-	-	-
Embankment	1 900	30	0.3	0.005	35	5
Terrain	2 000	90	0.3	0.028	25	5

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 the volume body corresponding to the embankment.
 - Select the *Embankment* properties set in the list.
 - **Apply**.
3. Repeat the operation for the *Footing* and the *Terrain*.



Active/inactive bodies

In this stage, we consider that the footing is not activated.

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

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:

1. Go to the tab **LOADS**.
2. On the toolbar, activate  **Gravity forces**.
3. Select all the bodies.
4. Click on **Apply**. The display of "GRAVITY FORCES" at the bottom left of the working window indicates the activation in this loading set.



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

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
 - Algorithm type: Pardiso
3. Click on **Validate**.



The algorithm Pardiso is the most powerful one. If user thinks that his computer may be not correctly sized for the calculation (for example with small RAM size), he may prefer the use of the algorithm type "Multifrontal". The "Direct-Skyline method" is the most robust one but also the slowest one.

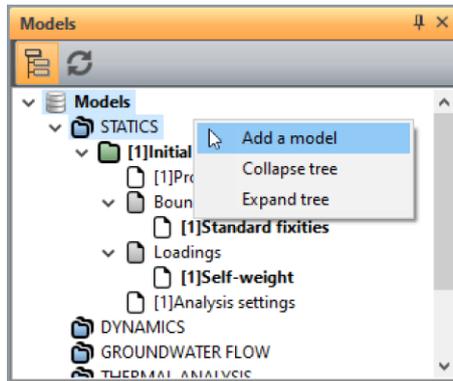


When using "Multifrontal method". The option "Calculation with secondary storage" is required when the matrix size of the model will be larger than the random memory (RAM) of the computer.

4.2. Calculation properties for the uniform pressure loading

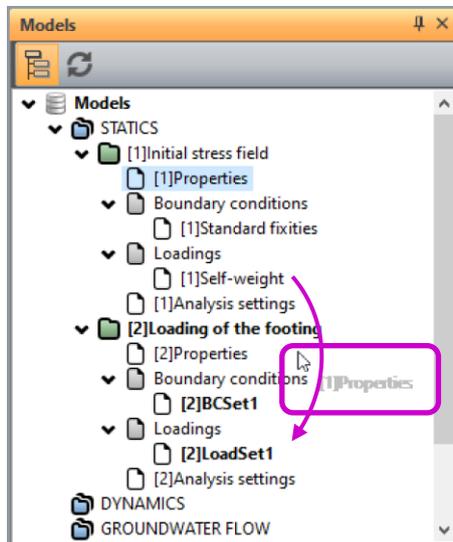
Model definition:

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 **Loading of the footing** as "Model name".
 - Select **Staged analysis** as initialization type.
3. Click on **Validate**.



We can now copy the definitions sets of the model 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 **Uniform pressure loading**.
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**.

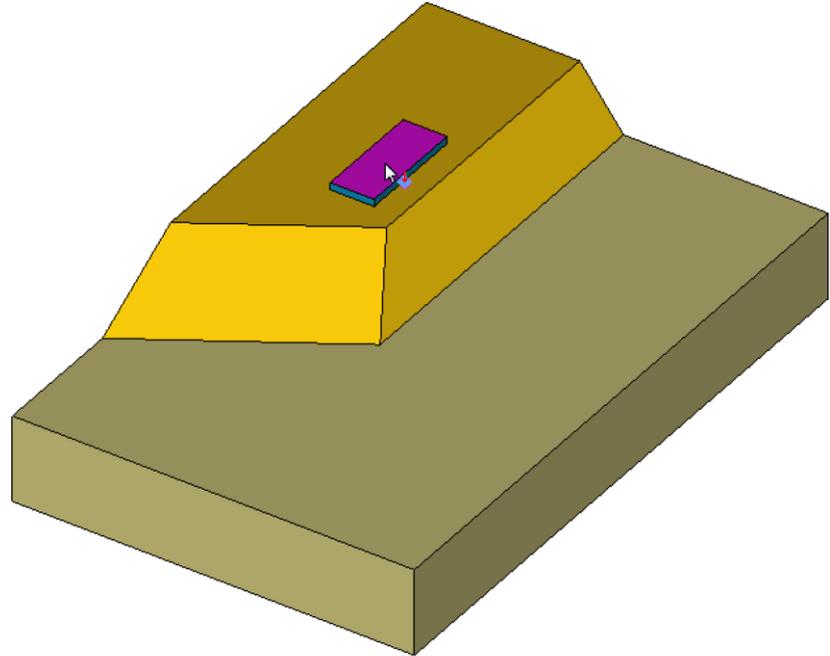
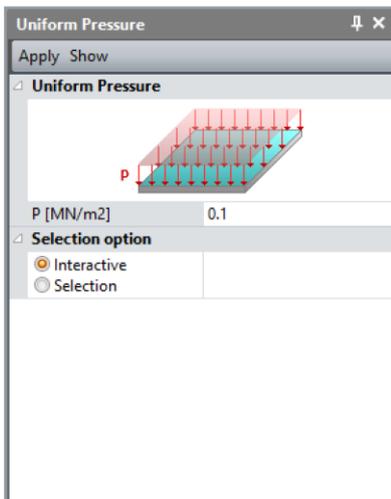


Boundary conditions:

There is no change in the boundary conditions.

Loading set:

1. Go to the tab **LOADS**. On the toolbar, activate  **Uniform pressure**.
2. Enter the value of 0,50 MN/m².
3. Select *Interactive* as selection option. As the affectation is interactive, click the mouse on the surface of the Footing body.



Optional. It is possible to modify the default name assigned to the loading set, LoadSet1. Press [F2] ; enter **Uniform pressure** for example.

Calculation parameters:

1. In **ANALYSIS**, activate **Analysis settings** .
2. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 5
 - Max number of iterations per increment: 500
 - Tolerance: 0,01
 - Method of resolution: 1- initial stresses
 - Solver type: Pardiso
 - Calculation type: Standard
 - Tick "Displacement initialisation", so that the settlements due to the initial stress field are shift to 0.
3. Close using **Validate**.



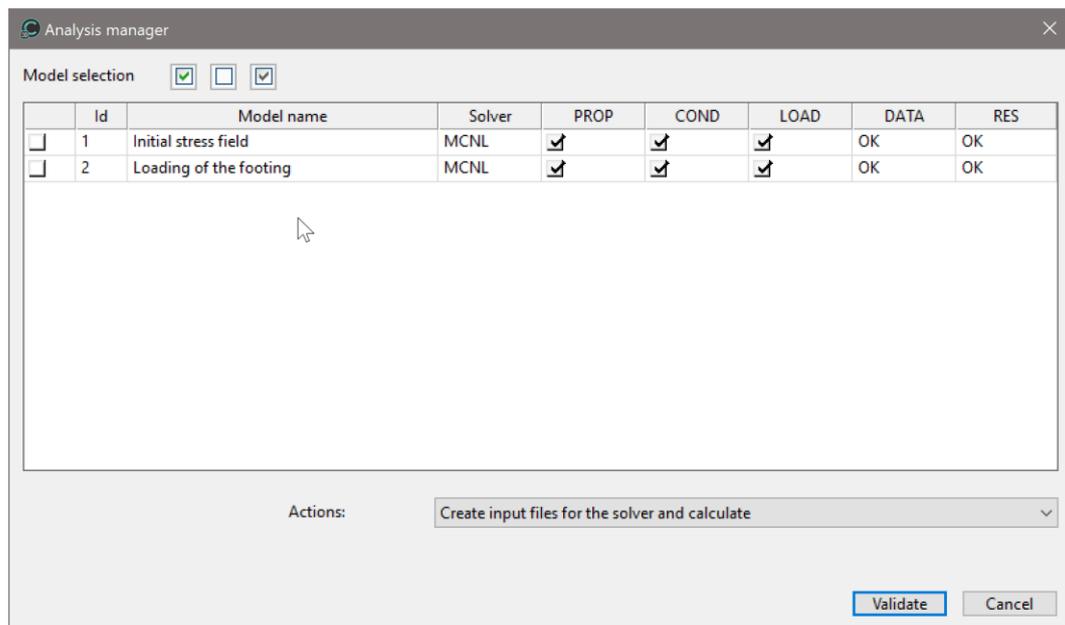
The option "Calculation with secondary storage" is required when the matrix size of the model will be larger than the random memory (RAM) of the computer.

5. 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 2 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).

6. FINITE ELEMENT RESULTS

The result is highly influenced by the mesh density, the type of interpolation (linear or quadratic) and the precision on the convergence algorithm. We show here several types of result that could be displayed for a sharp analysis of the model behaviour.

In the **RESULTS** tab, we display isosurface of total displacements, reached at this level of loading.

1. Go to the **RESULTS** tab.
2. In **Results selection**, select Increment #5.
3. Activate  **Type of results to display**.
 - Select **|u|, Total displacements** in Contour option,
 - **Apply**.
4. Activate  **Legend**.
 - Select **Contour plot** as legend type,
 - **Apply**.

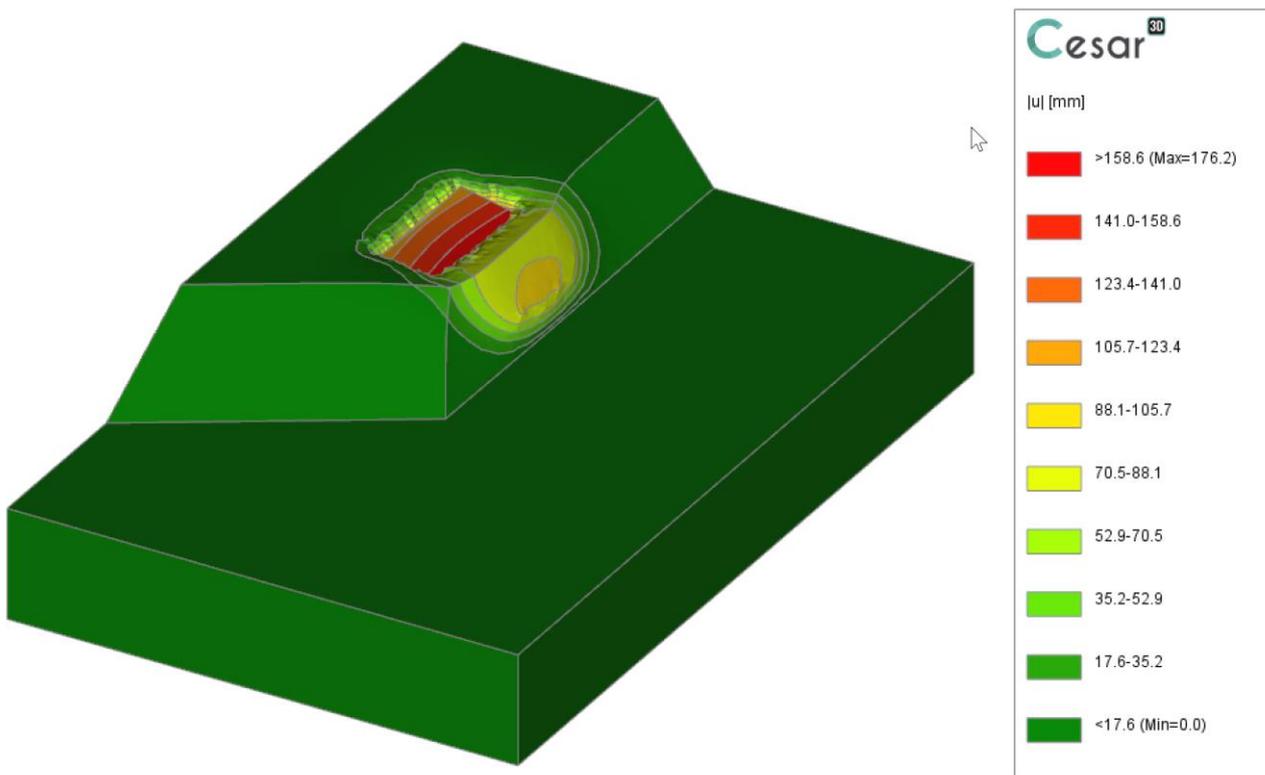


Figure 5: Deformed model with isosurfaces of total displacements

We can now display the isosurface of vertical displacements.

1. Activate  **Type of results to display**.
 - Select **u_z, vertical displacements** in Contour option,
 - **Apply**.
2. Activate  **Scalar settings**.
 - Untick **Activate**.
 - **Apply**.

3. Activate  **Isosurface**.
- Tick **Show isolevel surfaces**.
 - In **Operations**, select **Automatic**.
 - Set 10 as **Number of values**.
 - Click ⚡ for automatic values generation.
 - **Apply**.

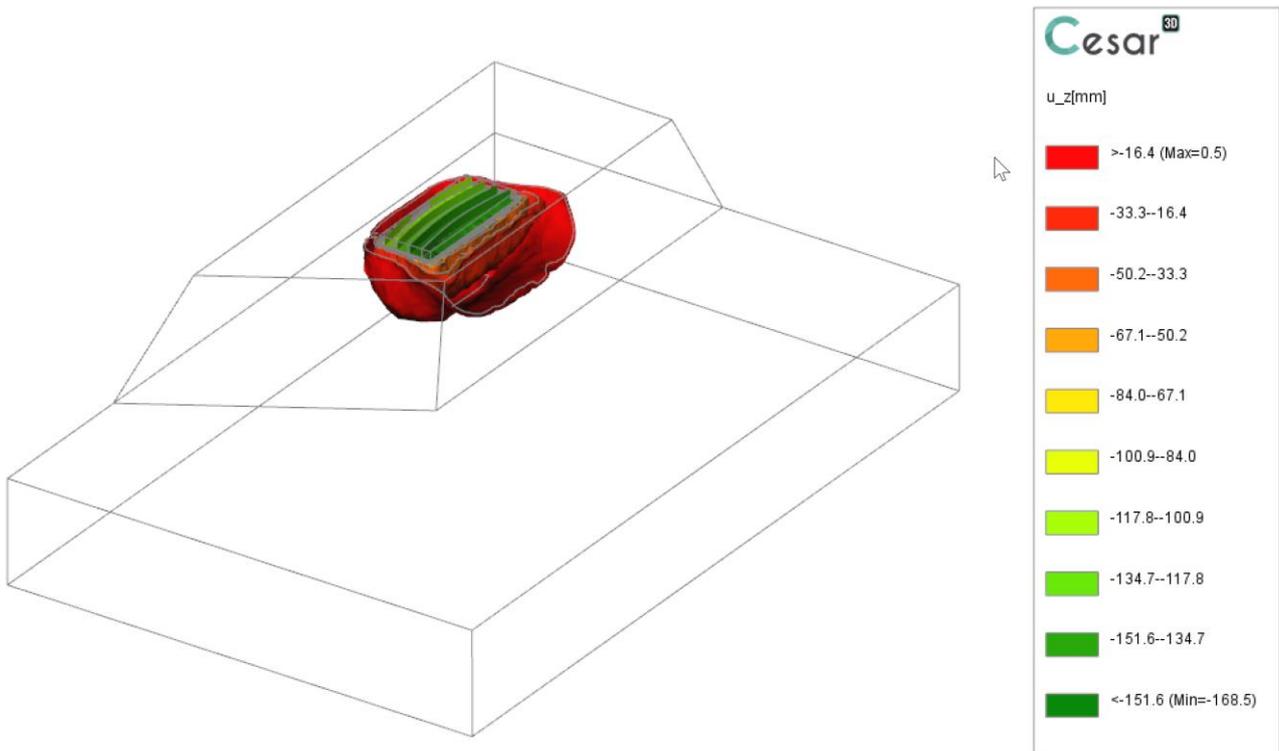


Figure 6: Isovvalue surfaces of the vertical displacements

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