Cesar 3D

Tutorial 3g.02

Loading of a pile foundation

1. PREVIEW

The problem considered is the analysis of a pile group for a bridge foundation subjected to a vertical and a horizontal load. This foundation will be simplified to a model including the column, the pile cap and the piles.

In the scope of this tutorial, the soil mass is also simplified as a unique soil mass. The soil-pile interaction is modeled with interface elements.

Version used: CESAR 2021 – CESAR 3D

1.1. Tutorial objectives

- Use of 3D meshing (tetrahedron filling and extrusion).
- Modelling of soil-stric joint elements
- Use of 3D post-processing tools.
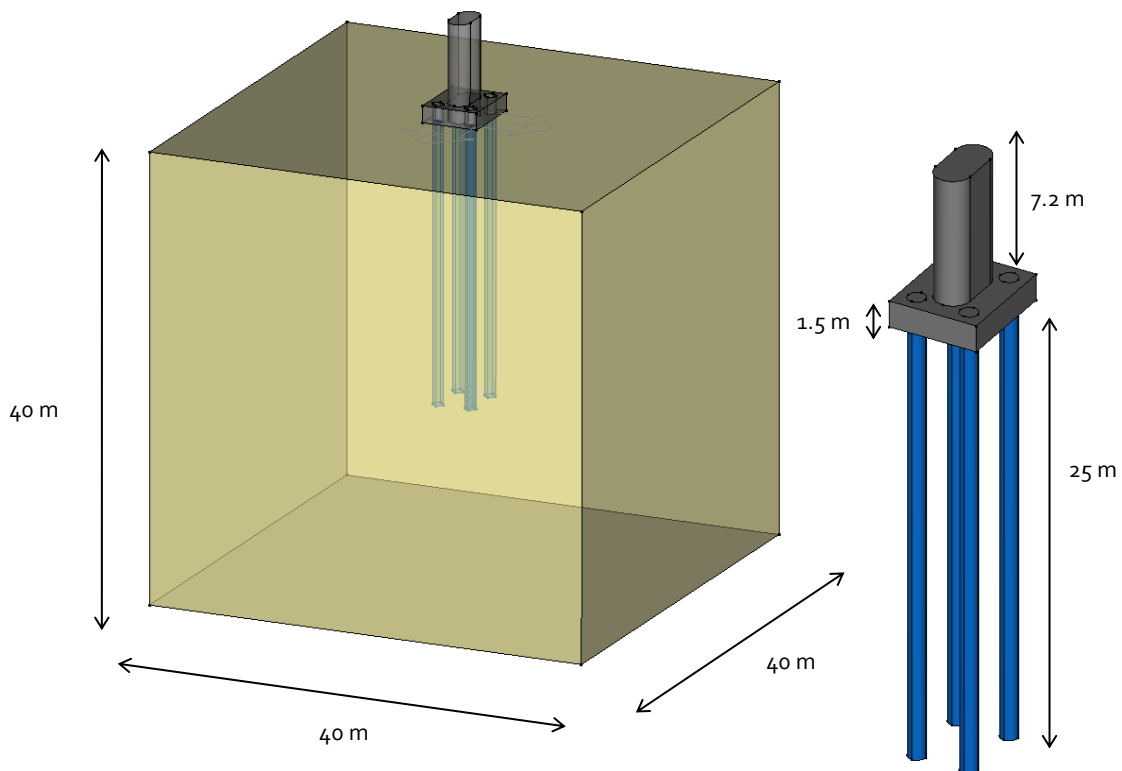
1.2. Problem specifications

3D geometry
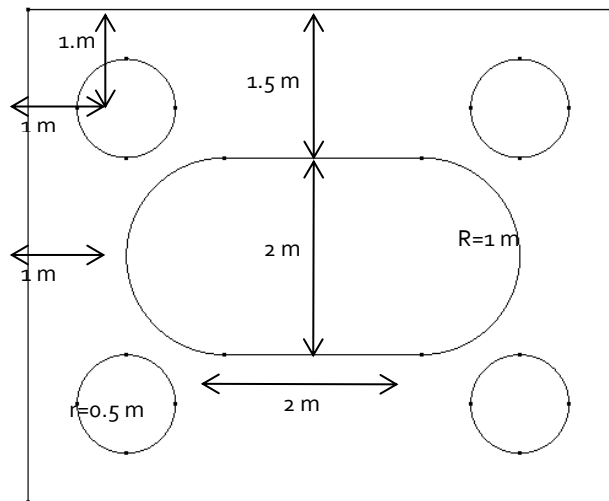

Figure 1: 3D geometry overview

Figure 2: Details of the cap geometry

Materials

	γ (kN/m3)	E (MPa)	ν	c (kPa)	ϕ (°)	ψ (°)
Concrete	25	35000	0.2	-	-	-
Silty clay	19	100	0.3	20	35	5

Boundary conditions and loading

- Standard boundary conditions are adopted: horizontal fixities on vertical planes, vertical fixities at the base of the model.
- In first instance, a uniform vertical pressure of 0,2 MPa is applied on the foundation. Then a uniform horizontal pressure of 0,1 MPa is applied.

1.3. Analysis steps

1. Initialise the stresses in the soil using the reduced density and $K_0 = 0.5$.
2. Apply the loads in 4 steps.

2. 3D MODELLING

Several methods are available to generate the volume mesh:

- By filling volumes with tetrahedrons,
- By simple vertical extrusions, a simple plane geometry is required.

In the present tutorial, we will detail the first method.

2.1. General settings

1.   Run CESAR 3D.

<u>Preferences</u>

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

<u>Units</u>

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 kN.
4.   In the same way, select the **Displacement** unit to mm. Adjust the number of digits to 0.0.
5.   Click on *Apply* to close.

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

2.2. Geometry

CESAR-LCPC proposes a full set of CAD tools for a complete drawing of the 3D geometry (surfaces, volumes, intersections).

<u>Drawing of the segments</u>

User can use tools ✗ *Points*, *Lines* and *Circle* for the geometry generation with reference to Figure 2.

As alternative and quicker method, use can also import a dxf file:

1.   Click on **File>Import>Geometry**
2.   Select *dxf* as file extension
3.   Browse and access to the folder "...\Tutorial12\"
4.   Select the file *tutorial12.dxf*.
5.   *Open*.
6.   A message box asks to validate the type of units. Validate the default one, click *Yes*.
7.   *Save* the file as *Tutorial12.cleo35*.

Piles, pile cap and column

1. Select all the group of segments of the pile cap and the column.

1. Generate the surfaces using [icon] *Plane surface*.
   - Activate option **All regions on surface**,
   - *Apply*.

1. Select these surfaces. For this selection, deactivate all ([icon]) in the *Selection toolbar* and activate [icon] *Select surfaces*.

2. Click on [icon], the *Extrusion* feature.
   - *Type of operation:* Translation
   - *Type of extrusion:* Generation of volume bodies
   - *Operation data:* Vx = 0 ; Vy = 0 ; Vz = 1 m
   - *Apply*.

3. Select the surface of the column base.

4. Click on [icon], the *Extrusion* feature.
   - *Type of operation:* Translation
   - *Type of extrusion:* Generation of volume bodies
   - *Operation data:* Vx = 0 ; Vy = 0 ; Vz = 7.2 m
   - *Apply*.

5. Select the 4 surfaces of the piles section.

6. Click on [icon], the *Extrusion* feature.
   - *Type of operation:* Translation
   - *Type of extrusion:* Generation of volume bodies
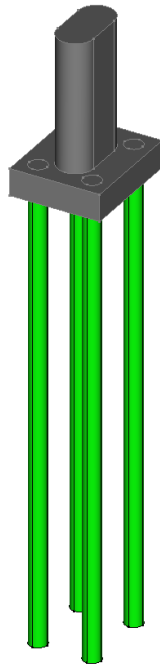   - *Operation data:* Vx = 0 ; Vy = 0 ; Vz = -25 m
   - *Apply*.

Figure 3: Geometry after import of the dxf file and generation of the surfaces

Soil volume

For the soil, we define a cubic box with 40m as main dimension.

1. Use the ▨ *Box* tool.
   - Set the body label as *Soil mass*
   - Select the option **2 points at corners**
   - Set first point P1: X1 = 0; Y1 = 0; Z1 = 0
   - Set second point P2: X2 = 40 m; Y2 = 40 m; Z1 = -40 m;
   - *Apply*.

Volumes intersection

We want to imprint the cap at the soil top and insert the piles in the soil mass. The tool is the *Foundation* (Column + cap + piles) and the object to be intersected is the *Soil mass*.

1. Use tool ▨ *Merge bodies.*
   - Set the column, the cap and the piles,
   - Set the name of the new boy as *Foundation*,
   - *Apply*.

2. Activate tool ▨ *Bodies intersections*.
   - Select option **Pick object and tool**,
   - Select the *Soil mass* as **Object**,
   - Select the *Foundation* as **Tool**,
   - *Apply*.

3. Select the volume of the soil. Define it as **"Cut objects"** using ⊕.
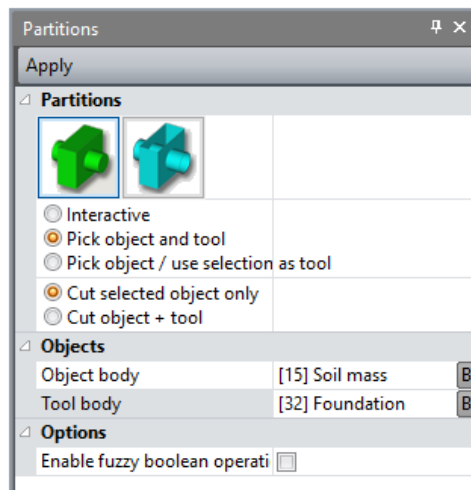
Groups definition:

Now that all the volume bodies are generated, we can manage them and group them by use, material, order of appearance in the phased analysis.... This is optional but is eases the model preparation and analysis.

1. Select the Foundation body. Click on ▨ *Explode bodies*.

We group the column and the pile cap (same material properties).

2. Select these volumes. Activate ▨ *Merge bodies*. Enter **Column+Pile cap** as body label. Click on **Apply**.

User can merge the 4 piles or let them independent. The second option will ease the analysis per pile.

3. Select the volumes corresponding to the piles. Activate ⬚ *Merge bodies*. Enter **Piles** as body label. Click on **Apply**.

A facultative step consists in affecting colours to these groups by using the pallet colour tool.

Interface bodies generation

The interface elements are connecting the soil mass with the piles shaft.

1. We first need to select the faces at the interface.
   - For this selection, deactivate all (🔧) in the *Selection toolbar* and activate 🔲 *Select faces*.
   - Select the faces on the piles shaft and the piles toe.

2. Click on 🟩 *Interface bodies*.
   - Select Contact (12 or 16 nodes) elements,
   - Set name as *Shaft interfaces*,
   - *Apply*.

3. Repeat this operation for each pile or make it for all at once. Distinction is the ability to set different interface parameters for each pile.
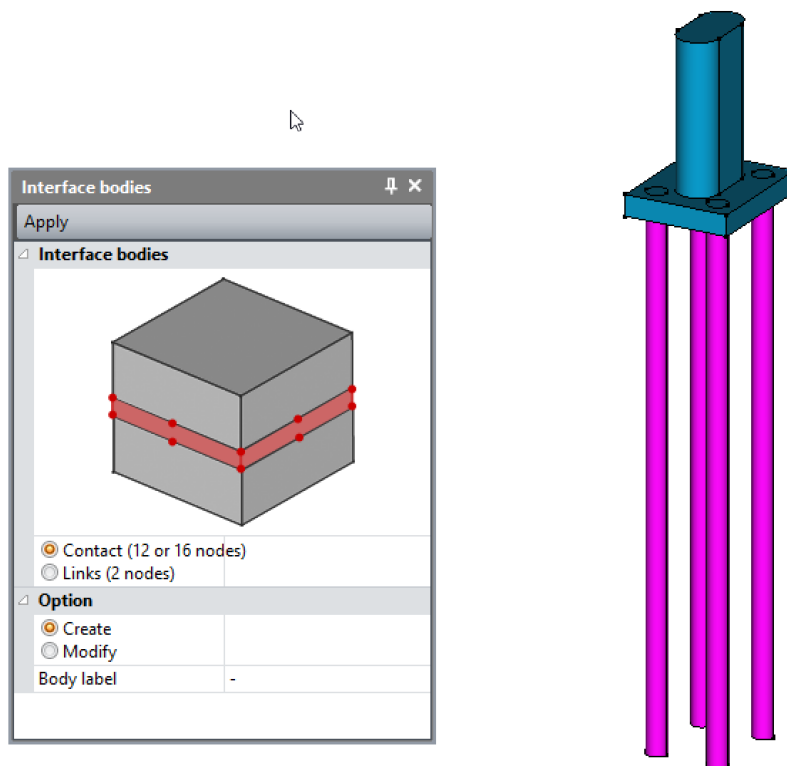

Figure 4: View of the selected faces for interface bodies generation

We repeat previous operations with the interface between the pile cap and the soil.

2.3. 3D meshing

Mesh refinement is important nearby the areas where high level of strains is expected. This will lead to more accurate results.

⚠ 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 (nearby the piles) to large segments on the boundary edges.

Mesh density

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 soil volume. Click on *Fixed length density* to divide these segments with a fixed length. Enter **10 m.** in the dialog box. *Apply*.

3.   Select all the edges of the foundation (cap, column and piles). Click on *Fixed length density* to divide these segments with a fixed length. Set it to **1m**. *Apply*.

4.   Select the edges of tip and top of the piles. Click on *Constant density* and set the number of divisions to 3. *Apply*.

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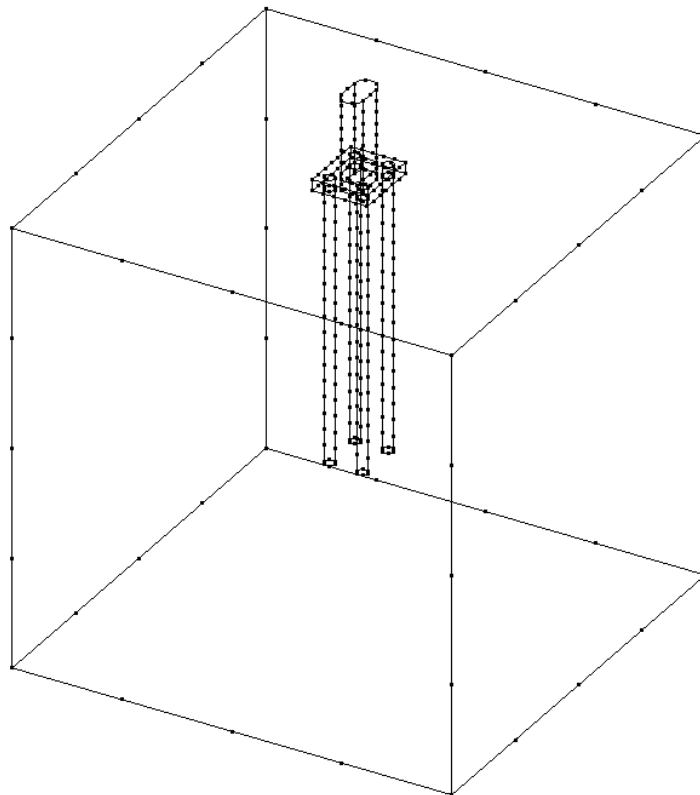

Figure 5: View of the segments and the defined density

3D meshing:

💡 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 "Quadratic interpolation" as **Interpolation type**.
   - Set "Tetrahedron mesh" as **Mesh type**.
   - Select the default Tetmesh generator and set the **Density factor** to 0.85.
   - Mesh creating function is **Quadratic**. It was set in initial preferences.
   - *Apply*.

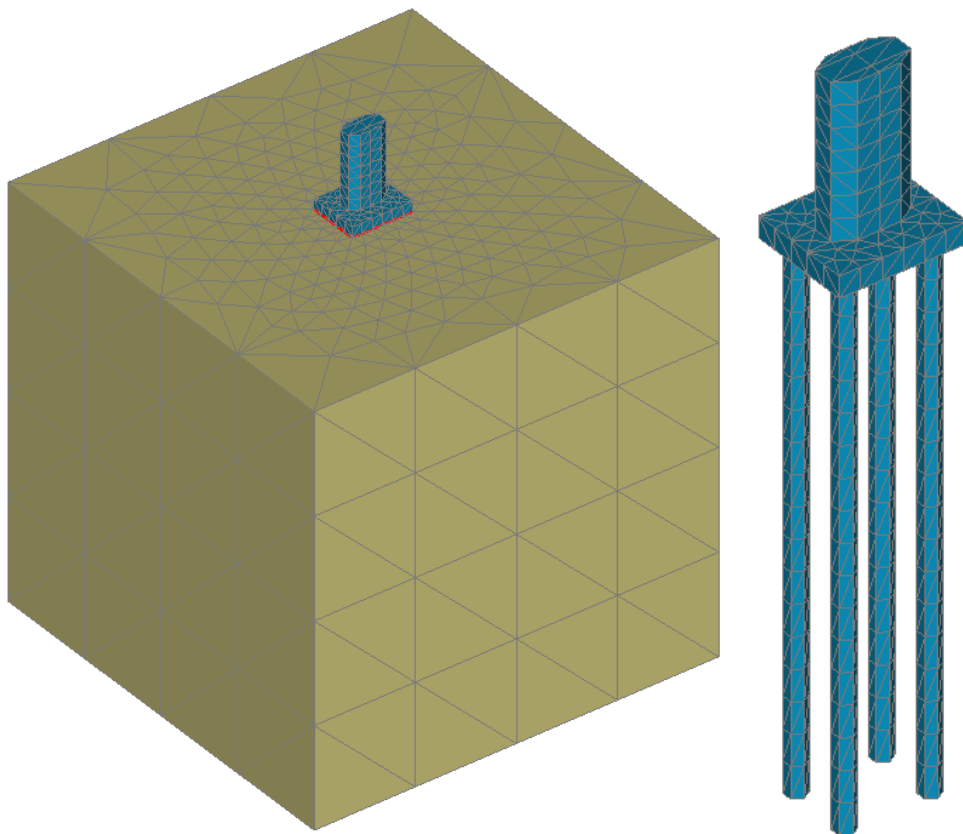💡 Note that the density factor has a high influence on the number of nodes generated. Cf. §6.

Figure 6: Mesh views

Mesh properties		⊨ ×
Update Elements quality check		
⊿ **Modelled bodies**		
Number of bodies	4	
⊿ **Mesh size**		
Number of groups	12	
Total number of nodes	35656	
Total number of elements	26318	
1D-elements	0 (-)	
Surface elements	0 (-)	
Volume elements	26318 (Q)	
⊿ **Mesh properties**		
Length [m]	0.000	
Area [m2]	0.000e+00	
Volume [m3]	0.000	
⊿ **Elements quality check**		
Number of elements entirely with .	0	
Number of elements partially with	0	
Number of very distorted element:	0	
Number of distorted surface eleme	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**.

3. CALCULATION SETTINGS

The calculation will consist of applying service loads on the column. A first stage will initialise the geostatic stresses and apply the dead load, self-weight of the pile foundation.

3.1. First stage 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. Set *Initialisation* as name of the model.
3. Select **MCNL** as Solver.
4. Tick **Plane strain** as model configuration, with **Staged construction**.
5. Tick **Geostatic stresses** as initialization type.
6. Click on *Validate*.
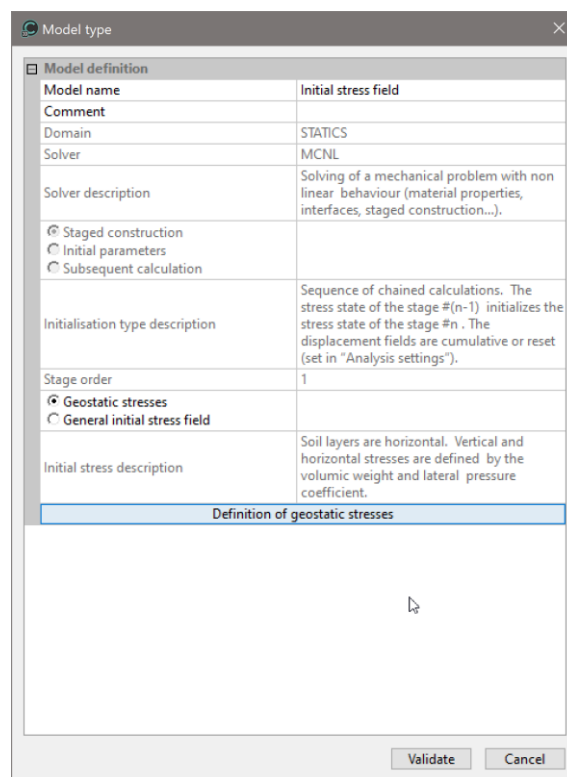
Now define the geostatic stresses as initial stress field:
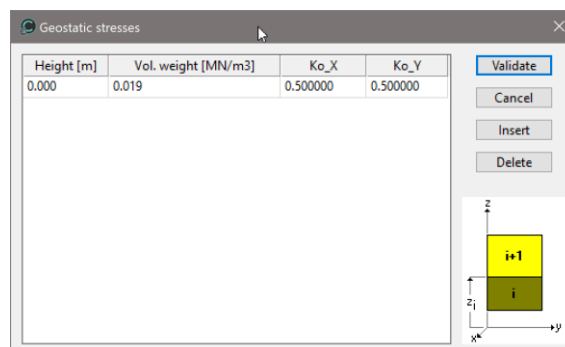
The model tree is now as illustrated below.
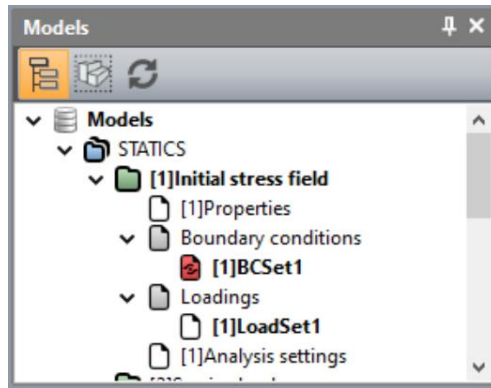
3.1.2 Material database

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, *Concrete*.

4. In **Elasticity parameters**, select "Isotropic linear elasticity" and define ρ, E and ν.

5. Click on **Validate**.

	ρ (kg/m³)	E (MN/m²)	ν
Concrete	2500	35000	0.2

6. Click and add a property set for the *Soil mass*.

7. In **Elasticity parameters**, select "Isotropic linear elasticity" and define ρ, E and ν.

8. In **Plasticity parameters**, select "Mohr-Coulomb without hardening" and define c, φ and ψ.

9. Click on **Validate** and **Close**.

	ρ (kg/m3)	E (MN/m²)	ν	c (MN/m²)	φ (°)	ψ (°)
Soil mass	1900	100	0.3	0.02	35	5

Material properties for the interface elements:

Properties of interfaces in CESAR-LCPC can be adherent, slipping or Coulomb's friction. In this case we select the Coulomb's friction with a friction angle equals to 2/3 of the soil one.

1. Click on **Properties for interface bodies**.
2. Give the name *Interface soil-concrete* for the properties set name.
3. Select the **Joint** type
4. Select **Coulomb's friction** as material model. Enter the properties of the contact between the wall and the soil (see table below).
5. Select the interface between the wall and the clay.
6. Click on **Apply** and **Close**.

Name of the group	Ei (MPa)	Rt (MPa)	c (MPa)	φ (°)	ψ (°)
Contact Concrete/Soil	10 000	0.1	0	24	0

⚠ Recommendations for the properties of interface elements:
- Contact modulus, Ei, is the value x 100 of the less stiff of the two materials in contact;
- Tensile strength, Rt, is the value of the tensile stress normal to the interface necessary to cause debonding. A value of 10000 MPa can be adopted as input of a high value of Rt would prevent the debonding.

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**.
3. Select the *Column*, *Piles* and *Cap* bodies and the set of parameter *Concrete* in the list. **Apply**.
4. Select the *Soil mass* body and the set of parameter *Soil mass* in the list. **Apply**.

5. On the left side, click on **Properties for interface bodies**.
6. Select all the interface bodies of the model and the set of parameter *Interface soil-concrete* 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.

<u>Loading set:</u>

1. Go to the tab *LOADS*.

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

3. Select the *Column*, *Piles* and *Cap* 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
   - Algorithm type: Pardiso
   - Storage:
     Storage of total strains: ✓
     Storage of plastic strains: ✓

3. ***Validate***.

💡 Use "Mulitfrontal" as algorithm type if you think your computer may be undersized for the calculation. 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. Second stage definition:

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 *Service loads* 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 *Initialisation*.
2. By drag and drop, place it on the new Properties set of model *Service loads*.
3. A dialog box proposes to copy or share these parameters. Select ***Copy***.
4. Repeat the operations 1 to 3 for the Boundary conditions set. Select ***Share***.
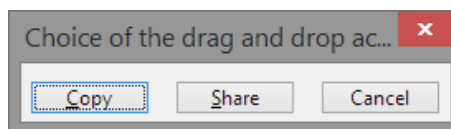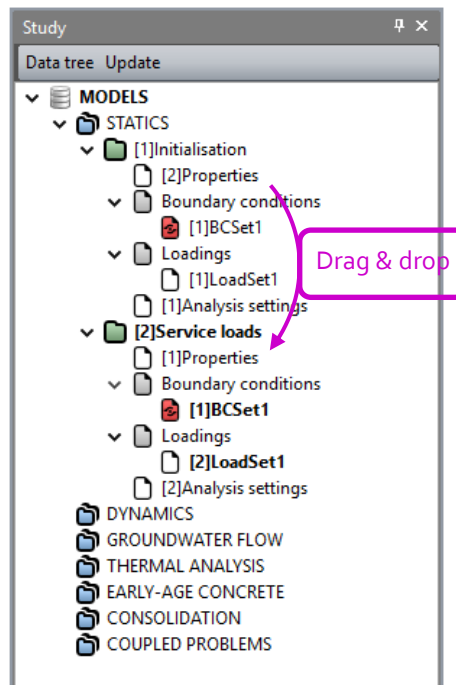
Assignment of properties sets:

No changes.

Boundary conditions:

No changes.

Loading sets:

We define 2 load sets: horizontal load, vertical load. The fact that these load sets are independent will ease

1. In the **Model tree**, click on *LoadSet1*. Press [F2] ; enter **Vertical load** as name.
2. Select the face at the top of the column.
3. Click on *Surface forces*. Enter Pz = -0,2 MN/m². *Validate*.
4. Right-click on *Loadings*. Select *Add loading set*.
5. Enter **Horizontal load** as name. *Validate*.
6. Select the face at the top of the column.
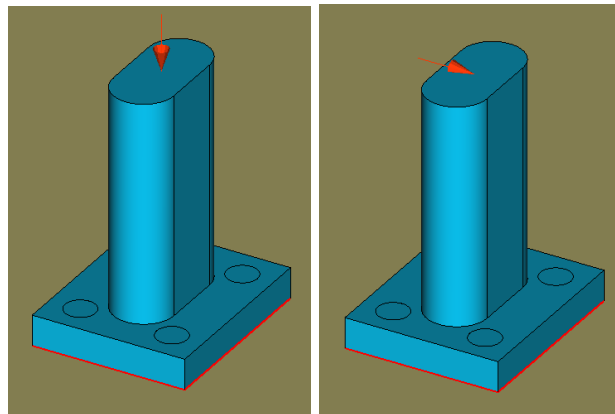7. Click on *Surface forces*. Enter Py = 0,1 MN/m². *Validate*.

Figure 7: Views of the vertical and horizontal load

Calculation parameters:

1. In *ANALYSIS*, activate **Analysis settings**.
2. In the **General parameter** section, enter the following values:
    -   Iteration process:
        Max number of increments: 4
        Max number of iterations per increment: 500
        Tolerance: 0,01
    -   Method of resolution: 1- initial stresses
    -   Solver type: Multi frontal
3. In the **Loading sets** section:
    -   Select **Load Set** *Vertical load*. Input values 0.25, 0.5, 0.75, 1 as program of loading.
    -   Select **Load Set** *Horizontal load*. Input values 0.,0., 0.5, 1 as program of loading.
4. Click on **Validate**.

⚠ Note that the load control program has been designed and simplified for a quick resolution of this highly non-linear problem. It has an evident influence on the results.

⚠ User can use the load control program to assign a specific program of loading/unloading of the structure. It is also useful for pseudo-dynamic analysis.
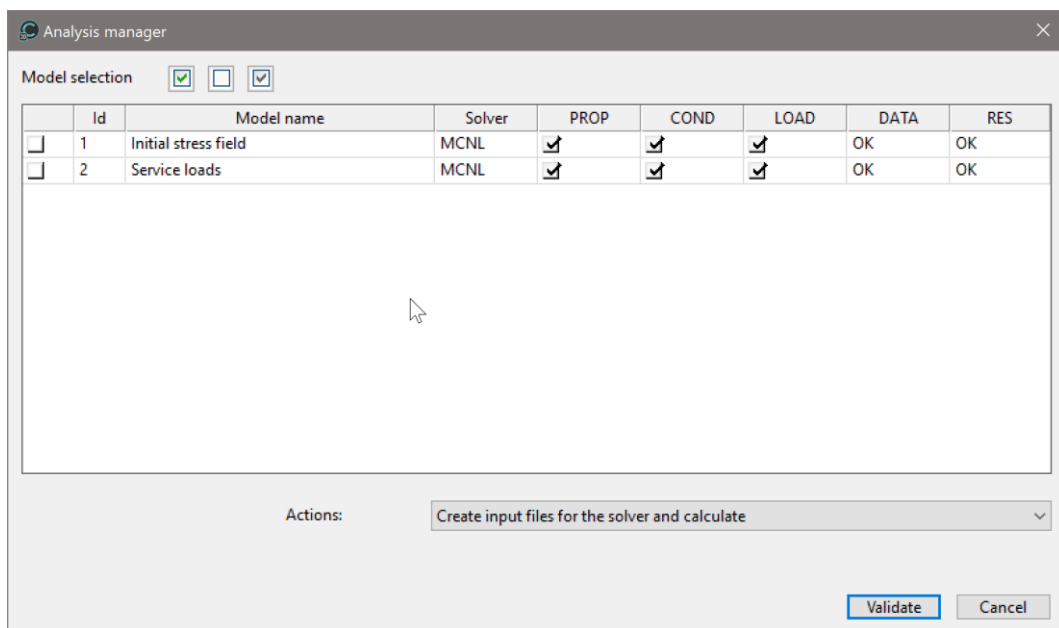
4. SOLVE

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. Tick the *Model1*.
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
☐	1	Initial stress field	MCNL	✔	✔	✔	OK	OK
☐	2	Service loads	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 total displacements inside the model with the deformed mesh.

1. Activate the tab *RESULTS*

2. Select model *Service loads* and Increment #4.

3. Click on 👁 *Type of results to display*.
   - Select **Total displacement** in Contour option,
   - Activate Contact Status.
   - *Apply*.

4. Click on 🗹 *Scalar settings*.
   - Select **Areas** as Style of contour plots.
   - Activate **Contour lines**, and select **Grey** as color set,
   - Select **Manual** as Scale and set the **Manual sup. value** to 3.4 mm.
   - *Apply*.

5. Click on 🗹 *Displacements settings*.
   - Select **Manual** as scale of **Deformed Mesh**.
   - Set Manual scale with 10 mm being represented by 4 m.
   - *Apply*.

6. Click on *Legend*.
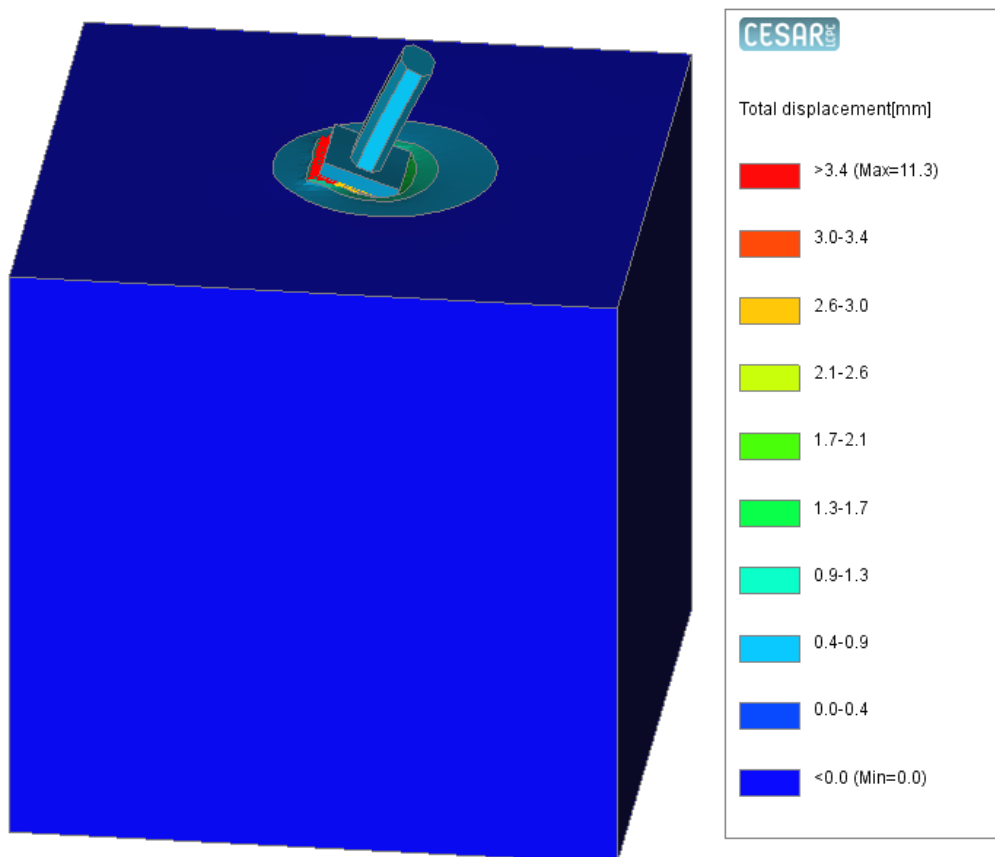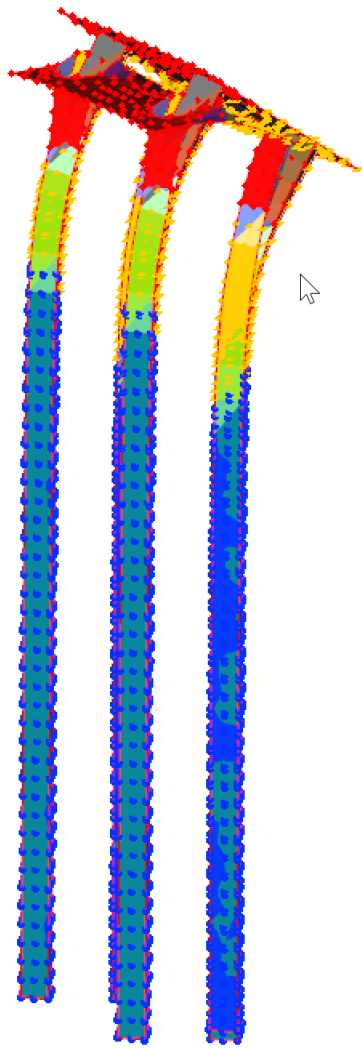   - Select **Contour plot** as legend type,
   - *Apply*.

Figure 8: Views of the deformed mesh and total displacements

We will now display main results along the pile shaft.

First, we display the piles alone.

1. Select the piles bodies,
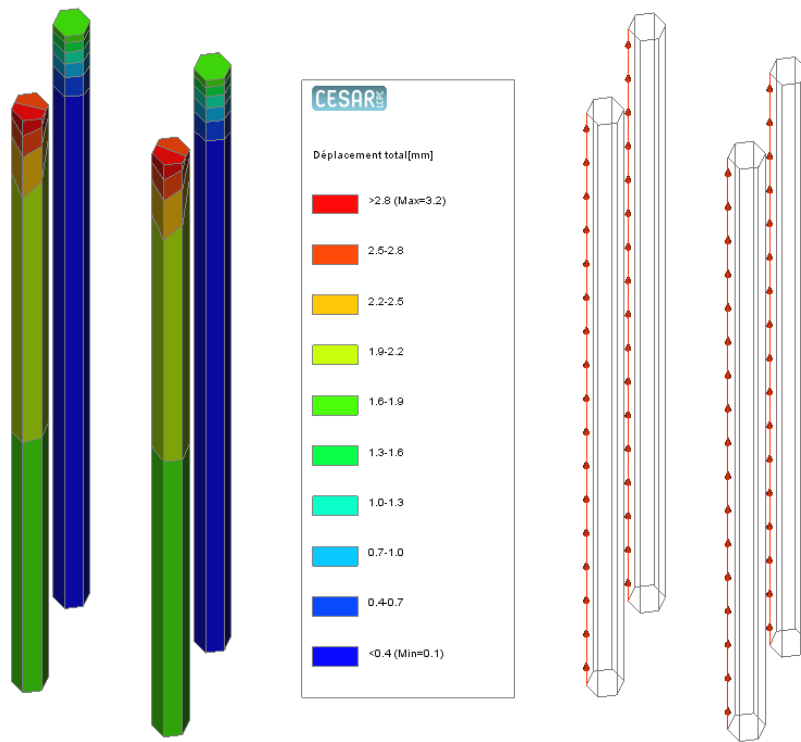
2. Use ![icon] to hide the other bodies.

Figure 9: Views of the deformed piles and the lines for graphs analysis

Now we can to define entities of results that will support our graphs.

1. Activate the **CHARTS** tab.

2. Select the segments along one pile shaft. For this selection, you can click on the segments ("Segments by angle" being activated) or make a box of selection using top-view.

3. Click on ⚬—⚬ .***Lines set***.

   - Give a name to this line set, **line p#1** for example.
   - ***Apply.***
   - The line set is created. It is displayed on the mesh with arrows indicating the orientation. This orientation can be modified using ***Inverse orientation***.

4. Repeat the previous operation for the 4 piles (see figure above).

We can now display graphs.

1. Activate ***Curves for a line set***.

   - Select the parameter, **Szz** for example;
   - Select one of the line set, **line p#1** for example;
   - Select all the increments,
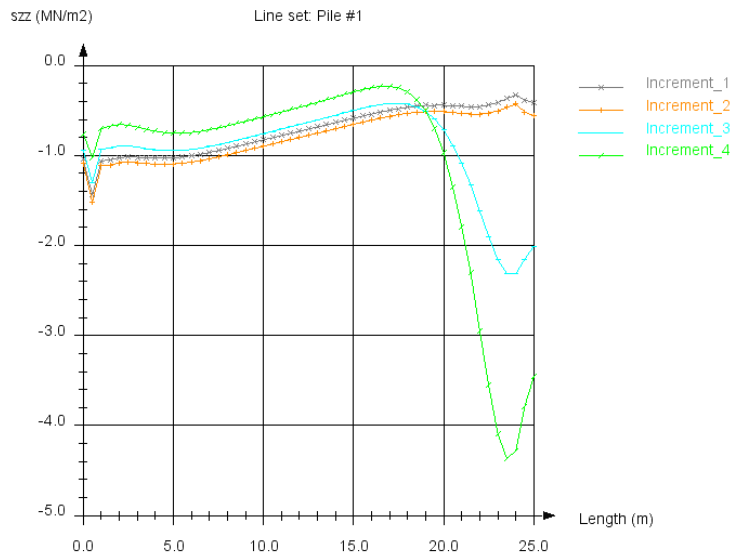   - ***Validate***.

2. The graph is displayed.

*Figure 10: Graph of Szz along pile shaft*

6. ADDENDUM: INFLUENCE OF THE DENSITY FACTOR ON THE MESH GENERATION

When choosing the TETGEN tetrahedron mesher, user sets the density factor. This factor has an influence on the density of elements that are generated between 2 neighbouring piles as shown below. The number of elements, and consequently the number of nodes, can be substantially reduced.
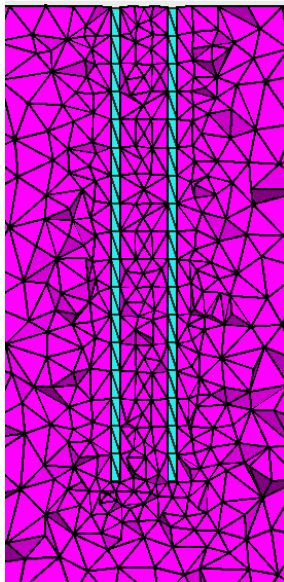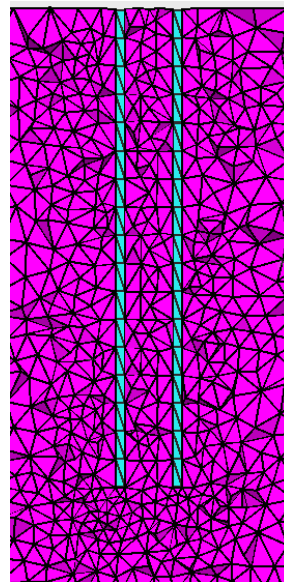
Density factor = 0.85
19 090 nodes
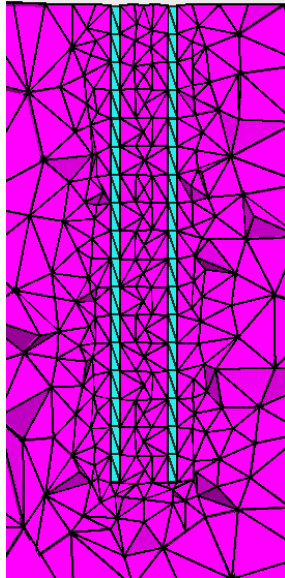
Density factor = 0.9
23 186 nodes
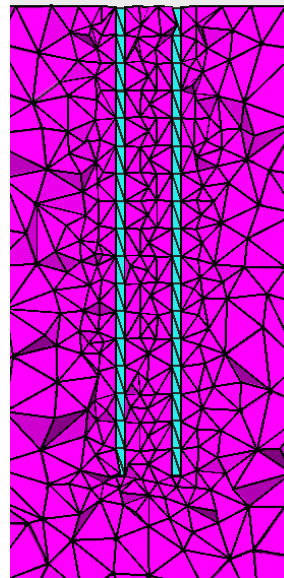
Density factor = 0.95
35 208 nodes

Density factor = 1.0
105 867 nodes

Density ratio	0.85	0.9	0.95	1.0
Duration of calculation #2	2'15''	2'41''	6'07''	39'47''
Maximal displacement on soil (mm)	3.0	3.0	3.1	3.1
Max stress in Cap (MPa)	2.156	2.229	2.221	2.224
	-1.054	-1.084	-1.081	-1.080

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