Tutorial 2g.05

Bearing capacity of a shallow foundation

Ref: CESAR-TUT(2g.05)-v2021.0.1-EN

1. PREVIEW

The maximum load that can be sustained by shallow foundation elements, known as the bearing capacity, is a function of the cohesion and friction angle of soils as well as the width *B* and the shape of the foundation. In this tutorial, the foundation is a rigid circular footing (diameter *B* = 1 m) resting on an undrained homogeneous soil modelled using a Tresca criterion (undrained cohesion c_U).

CESAR helps the design of shallow foundation. With the Finite Element method, we will search the limit equilibrium of the system.

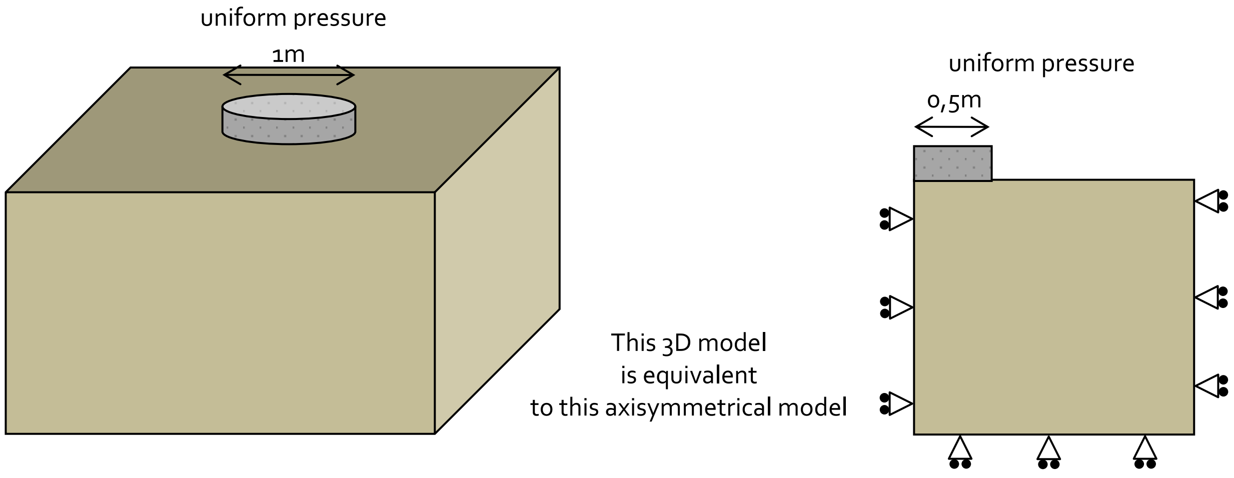
1.1. *Tutorial objectives*

- Learn how to design a mesh.
- Get familiar with the incremental-iterative process for non-linear analysis.
- Determine the safety factor on forces.
- Handle post-processing tools.

1.2. *Problem Specifications*

General assumptions

- Static analysis,
- Non-linear behaviour of the soil,
- Axisymmetrical problem, i.e. only a part of the foundation is modelled and analysed.



Material properties

The referenced article assumes the soil to behave in a rigid-perfectly plastic manner following the Tresca criterion.

	γ_h (kN/m³)	E_U (MPa)	ν	c_U (kPa)	φ_U (°)	K_0
Soil mass	20	50	0.45	20	0	0.5

2. GEOMETRY AND MESH

2.1. *General settings*

1. Run CESAR 2D.
2. Set the units **Units**.
   - In the tree, select the leaf **General/Length** and set the unit **m** in the bottom left combo box.
   - In the tree, select the leaf **Mechanic/Force** and set the unit **kN** in the bottom left combo box.
   - In the tree, select the leaf **Mechanic/Displacement** and set the unit **mm**.
   - Click on ***Apply*** to close.
3. In ***Working plane***, set the visible grid to 1m (dX = dY = 1m)

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

2.2. *Geometry*

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

Drawing of the geometry:

1. Click on . The ***Points*** dialog box is displayed.
2. Enter **'0, 0'** as X and Y, and press ***Apply*** button.
3. Tick "Linked points" to generate automatically a segment between two points.
4. Enter **'0.5 , 0'**, and press Apply button. Segment A is created.
5. Enter **'6, 0'**, and press Apply button. Segment B is created.
6. Enter **'6, -10'**, and press Apply button. Segment C is created.
7. Enter **'0, -10'**, and press Apply button. Segment D is created.
8. Enter **'0, 0'**, and press Apply button. Segment E is created.

Other methods could be used:

1. Define a grid of 1m x 1m and draw the lines with the mouse.
2. Use of "right click" to get information on an existing node and modify coordinates.

2.3. 2D Meshing

Mesh density

Define dense divisions in the area of high strains, i.e. below the loaded foundation. Use a progressive density definition to generate a progressive evolution from small segments to large segments on the boundary edges.

1. Go to the tab ***MESH*** on the project flow bar to start the definition of divisions along lines.
2. Select edge A. Click on ***Constant density*** to divide this segment in a fixed number. Enter **5** on the number of divisions. Click on ***Apply***.
3. Click on ***Variable density*** to divide the segment with a variation of lengths. Tick **First division** and **Last division** to define the method. Enter **0.1 m** as First division and **1 m** as **Last division**. Click on edge B.
4. Select edge C. Click on ***Fixed length density*** to divide this segment with a fixed length. Enter **1 m** in the dialog box. Click on ***Apply***.
5. Select edge D. Click on to impose a length for the divisions of this segment. Enter **1 m** in the ***Fixed length density*** dialog box. Click on ***Apply***.
6. Click on to divide the segment with a variation of lengths. Tick **First division** and **Last division** to define the method in the ***Variable density*** dialog box. Enter **0.1 m** as First division and **1 m** as **Last division**. Click on edge E. Click on ***Apply***.

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

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

Meshing:

1. Click on the ***Surface meshing*** tool .
   - Choose **Quadratic** as interpolation type.
   - Choose **Triangle** as element shape.
2. Click on ***Apply*** to generate the mesh.

CESAR-LCPC proposes 3 levels for the surface meshing procedure, giving the possibility to generate a coarse or dense mesh. Adjust it in **Preferences>Program settings**: linear interpolation = coarse, cubic interpolation = dense.

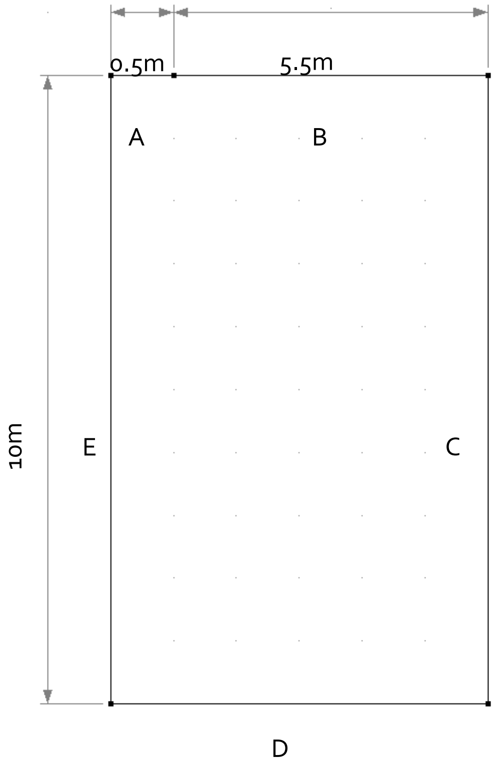


Figure 1: Geometry

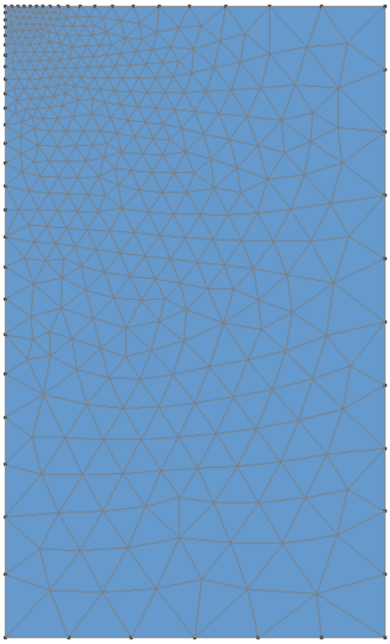


Figure 2: Example of mesh

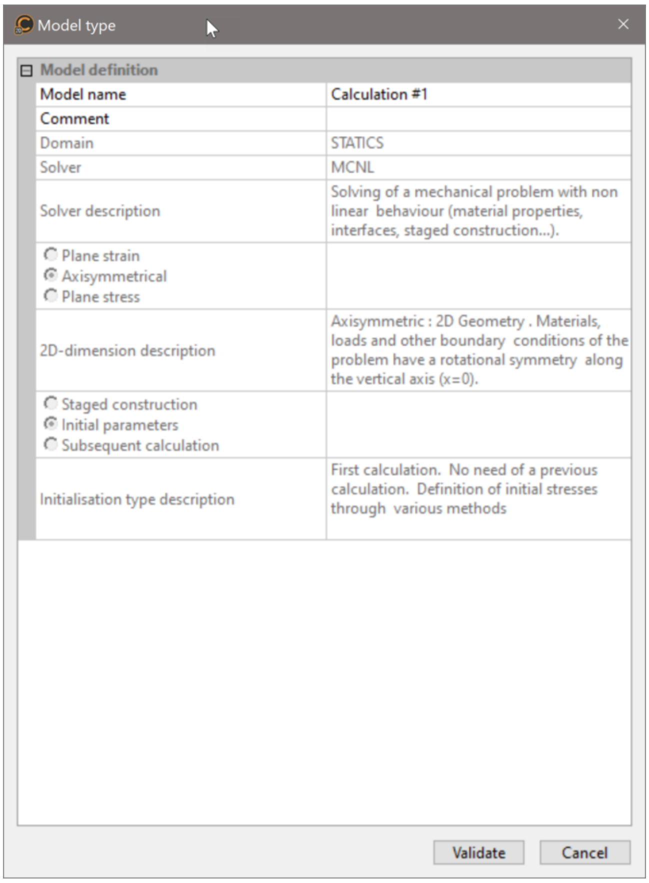
3. LIMIT PRESSURE ANALYSIS

3.1. *Model settings*

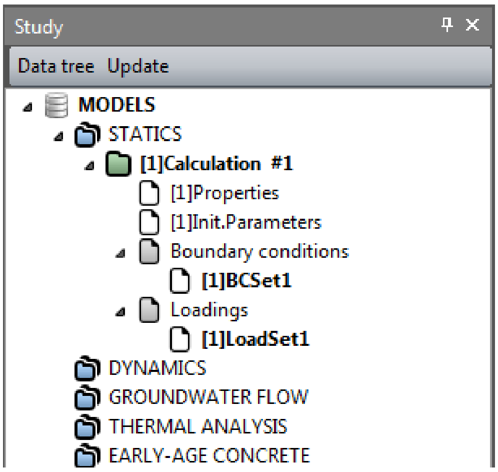
Model definition:

1. On the right side of the working window, the Tree view window displays the list of physical domains.
2. Right click on STATICS. Click on ***Add a model***. A new toolbox is open for definition of the Model.
3. Enter **Calculation #1** as "Model name".
4. Select **MCNL** as "Solver".
5. Tick **Axisymmetrical** as model configuration, with **Initial parameters**.
6. Click on ***Validate***.

See chapter "Initial stress field" in the document "Get Started with CESAR-LCPC v5"



The date tree is now as illustrated below.



<u>Material properties for the solid elements:</u>

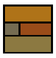
We initially define the material library of the study.

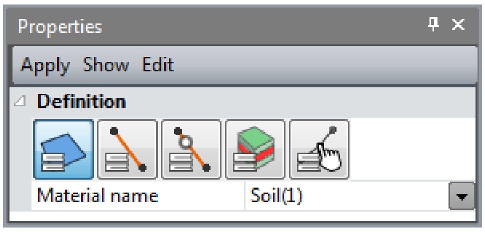
1. Click on ***Properties for 2D elements***.
2. Give a name for the properties set name ("Soil" for example).
3. In **Elasticity parameters**, choose "Isotropic linear elasticity" and define ρ, E and ν.
4. In **Plasticity parameters**, choose "Mohr-Coulomb without hardening" and define c, φ and ψ.
5. Click on ***Validate*** and ***Close***.

	ρ (kg/m³)	E (MN/m²)	ν	c (MN/m²)	φ (°)	ψ (°)
Soil mass	2000	50	0.45	0.02	0	0

<u>Assignment of data sets:</u>

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 window is displayed. Click on  ***Properties for 2D elements***.
3. Select the body of the soil mass on the model window and the setof parameter in the list.
4. ***Apply***.



<u>Initial stress field:</u>

The initial stress field is initialized as an existing geostatic stress field; vertical stress is linked to the horizontal stress by Ko value.

1. Go to the ***INITIAL PARAMETERS*** tab.
2. Select  ***Geostatic stresses***.
3. Click on ***Insert*** to define a new layer.
4. Enter the following values:

Height (m)	Volumic weight (MN/m³)	Ko_x
0	0.02	0.5

5. ***Validate***.

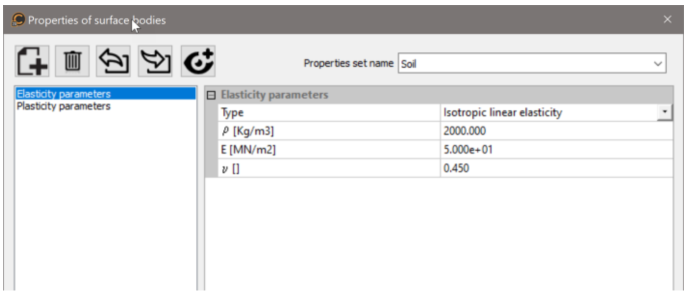


Figure 3: Toolbox for properties assignment

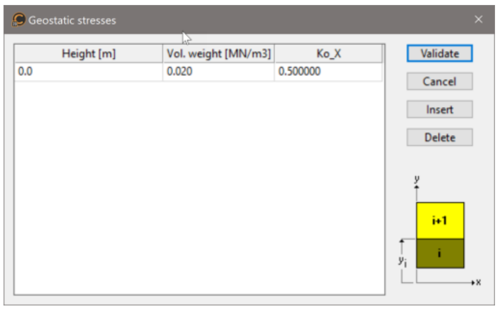


Figure 4: Toolbox for initial geostatic stresses input

Boundary conditions:

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

Default name of the boundary condition, BCSet1, set can be edited using the [F2] key.

Several boundary conditions sets are possible. Right click on **Boundary conditions**, in the data tree, to generate another one.

Loading:

1. Go to the ***LOADS*** tab.
2. On the toolbar, activate ***Linearly distributed pressure***.
    - Tick the box "Uniform pressure".
    - Enter the value of 0.120 MN/m.
3. Select the segment A.
4. ***Apply***.

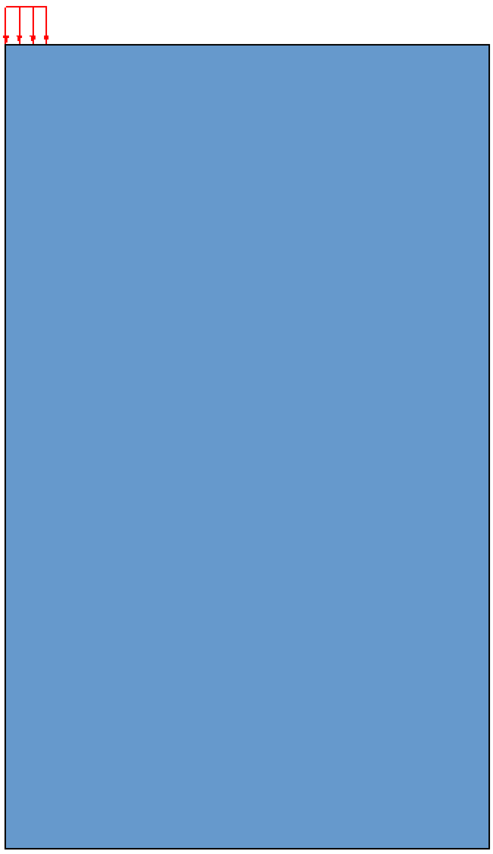
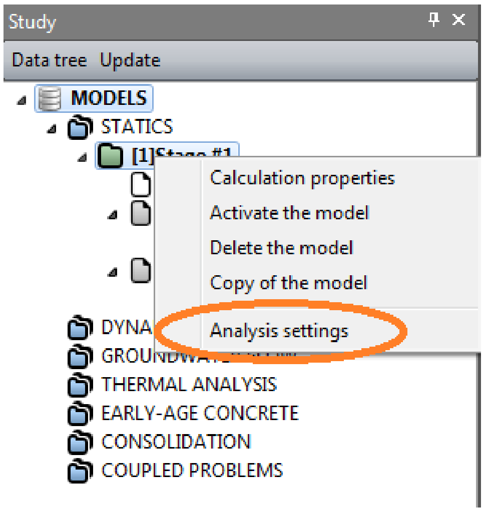
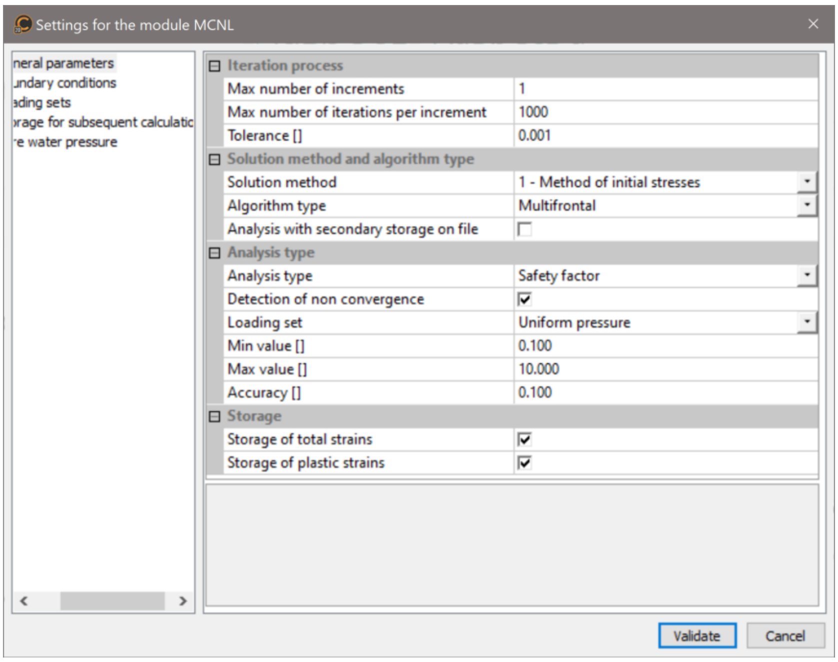


Figure 5: Display of the uniform pressure

Analysis settings :



1. In the data tree, right click on the calculation title. In the displayed list, select **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: 1000
     Tolerance: 0.001
   - Solution method: 1- Method of initial stresses
   - Algorithm type: Pardiso
   - Analysis type: Safety factor
     Tick "Non-convergence detection"
     Set "Minimum Value" to 0.1
     Set "Maximum Value" to 10
     Set accuracy to 0.1
   - Tick "Storage of plastic strains".
   - Tick "Storage of total strains".
3. Validate and close with ***OK***.

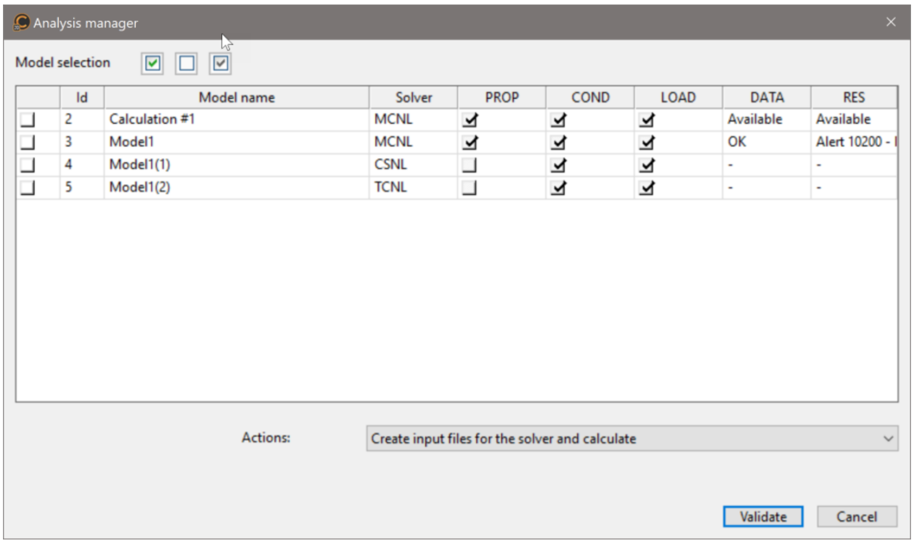


3.2. Solve

1. Go to the ***ANALYSIS*** tab.
2. Click on ***Analysis manager***.
3. Select the model
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".

The computation will take few minutes depending on the computer configuration.

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).

3.3. *Results*

The result is the safety factor. It is displayed:

- In the **Project window**,
- In the listing file.

According to "Undrained bearing capacity factors for conical footings on clay", G. T. Houlsby and C. M. Martin, Geotechnique 53, No. 5, 513–520, 2003; the ultimate bearing capacity obtained for the considered footing problem is:

$$p = 5.69 \times c_u = 113.8 \text{ kPa}$$

The calculated safety factor is 0.99. Thus, the limit pressure is 118.8 kPa, which fits to the theoretical value of 113.8 kPa.

<u>Display of the scalar plot of plastic strains</u>

1. Click on ***RESULTS***
2. Click on ***Result Types***
   - Select **Deformed** as Mesh,
   - Tick **the box** "Active" and select **Norm of plastic strain in** *Contour plots* section,
   - Click on ***Apply***.
3. Click on ***Contour plot settings***.
   - Tick the box "Activate" and select **Areas** as *plot style*,
   - Tick the box "Contour lines"and select **Grey** as *color set*,
   - ***Apply***.
4. Click on ***Displacement settings***.
   - Select **Manual** as scalar type,
   - Set that a value of **100 mm** is represented by **2 m**,
   - ***Validate***.
5. Click on ***Legend***.
   - Select **Contour plot** as *Legend type*,
   - ***Apply***.

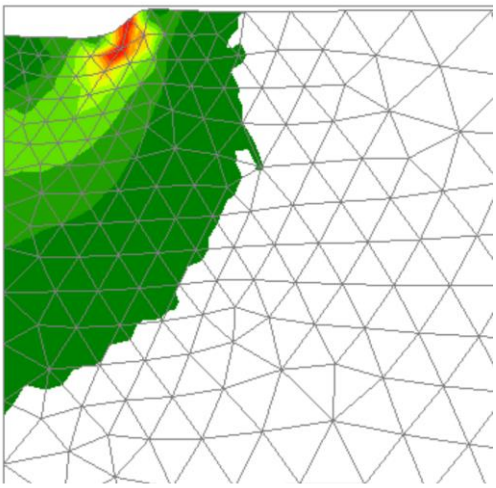


Figure 6: View of the field of plastic strains

4. Other types of calculations for determining the limit pressure

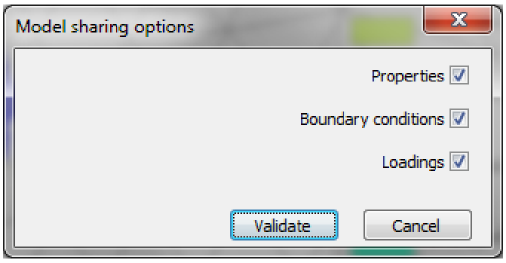
4.1. *Load controlled analysis*

This analysis is similar to the previous one, with the difference that the process of loading is not automatic but user-defined.

Model definition:

1. In the ***CALCULATION TREE***, right-click on **Calculation #1** and **Copy of the model**.
2. Define the model type:
   - Enter **Calculation #2** as name.
   - Select **Initial parameters** as calculation type.
   - ***Validate***.
3. Share the **Properties**, **Boundary conditions** and **Loads**. **Validate**.

By sharing, any modification in one of the model will automatically be updated in the other one.



Material properties:

No changes.

Initial stress field:

No changes.

Boundary conditions:

No changes.

Load Case

No changes.

Calculation parameters:

1. Right-click on the model **Calculation #2** in the ***CALCULATION TREE*** and activate ***Analysis settings***:
2. In the **General parameters** tab:
   - Iteration process:
     Max number of increments: 12
     Max number of iterations per increment: 1000
     Tolerance: 0,001

- Solution method: 1- initial stresses
- Algorithm type: Pardiso
- Analysis type: Standard

3. Validate using ***OK***.

In the **Loading sets** tab, the program automatically sets the factors to 1 (default value), divided by 12 (the number of increments). Therefore, one gets 12 fractions of 120kPa as loading program.

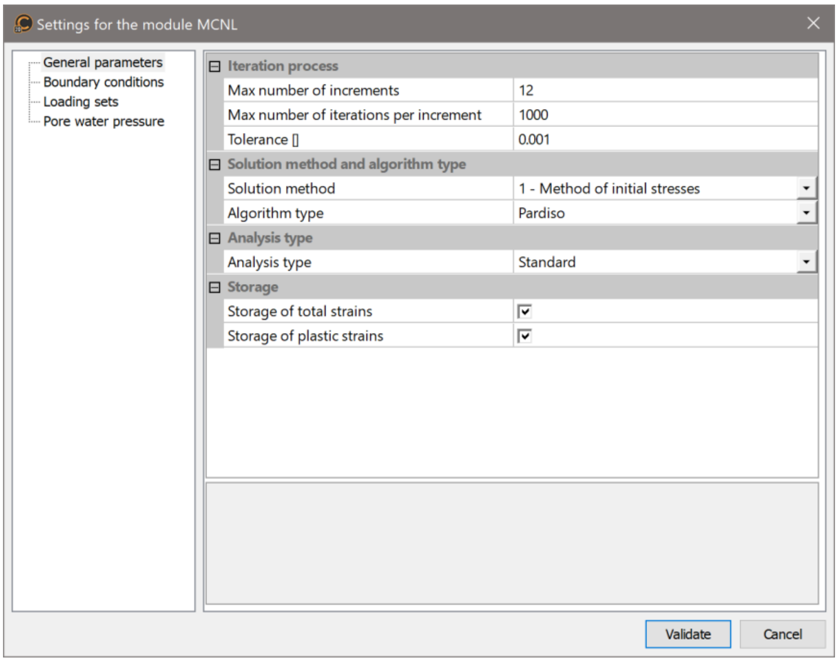


Figure 7: Toolbox for the setting of the calculation parameters

Solve:

Now that all data are input, go to ***ANALYSIS***.

1. Click on ***Analysis manager***.
2. Select the model **Calculation #2**.
3. Select **Create input files for the solver and calculate**.
4. **Validate**.

All the messages during the analysis will be shown in the project 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).

Results

The calculation process is displayed:

- In the **Project window**,
- In the listing file.

We read that the calculation stopped during increment #12 because the criteria of convergence are not verified until the maximum number of iteration (1000). This means that the equilibrium of the soil is not reached under the 12^{th} increment of load.

The last value of load applied with an equilibrium is 11/12 x 120kPa = 110 kPa.

The limit pressure value is between 110 kPa and 120 kPa.

```
Iteration number 992   Convergence index (residual) =  0.11032E+00
Iteration number 993   Convergence index (residual) =  0.11029E+00
Iteration number 994   Convergence index (residual) =  0.11027E+00
Iteration number 995   Convergence index (residual) =  0.11025E+00
Iteration number 996   Convergence index (residual) =  0.11023E+00
Iteration number 997   Convergence index (residual) =  0.11021E+00
Iteration number 998   Convergence index (residual) =  0.11019E+00
Iteration number 999   Convergence index (residual) =  0.11017E+00
Iteration number 1000  Convergence index (residual) =  0.11015E+00

CPU time for iteration 12  ( 1001 iter.)           :     10.68 seconds
CPU time for output processing                     :      0.02 seconds



**** STOP because of NO CONVERGENCE ****

STOP in EXMCNL, IERRCS = 200

         ----------------------------
         END of analysis in EXEC mode
         ----------------------------
```

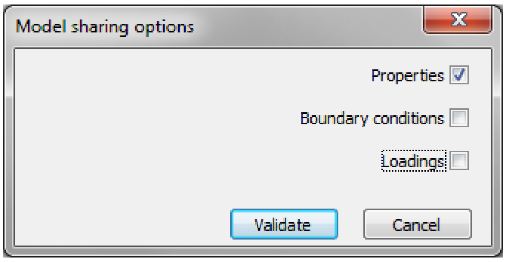
Figure 8: Extract of the listing at the end of the calculation

4.2. Displacement controlled analysis

This calculation is similar to the previous one. Only loads and boundary conditions are modified.

<u>Model definition:</u>

1. In the ***CALCULATION TREE***, right-click on **Calculation #1** and **Copy calculation**.
2. Define the model type:
   - Enter **Calculation #3** as name.
   - Select **Initial parameters** as calculation type.
   - Click on ***Validate***.
3. Share the **Properties**. Unshared the **Boundary conditions** and **Loads** as they will be modified. **Validate**.



<u>Material properties:</u>

No changes.

<u>Initial stress field:</u>

No changes.

<u>Boundary conditions:</u>

The imposed displacements are considered as a new set of boundary conditions. There are no changes for standard supports at the boundaries.

1. In the ***CALCULATION TREE***, right-click on **Boundary conditions** and then click on **Add boundary conditions set**.
   - Name it "Imposed displacement"
   - **Validate**.
2. Go to the ***BOUNDARY CONDITIONS*** tab. Select "Imposed displacement" on *boundary condition sets* toolbox.
3. Click on ***General definition***.
   - Tick **v imposed**.
   - Enter the value of **-40 mm**.
   - Select the segment A.
   - ***Apply***.

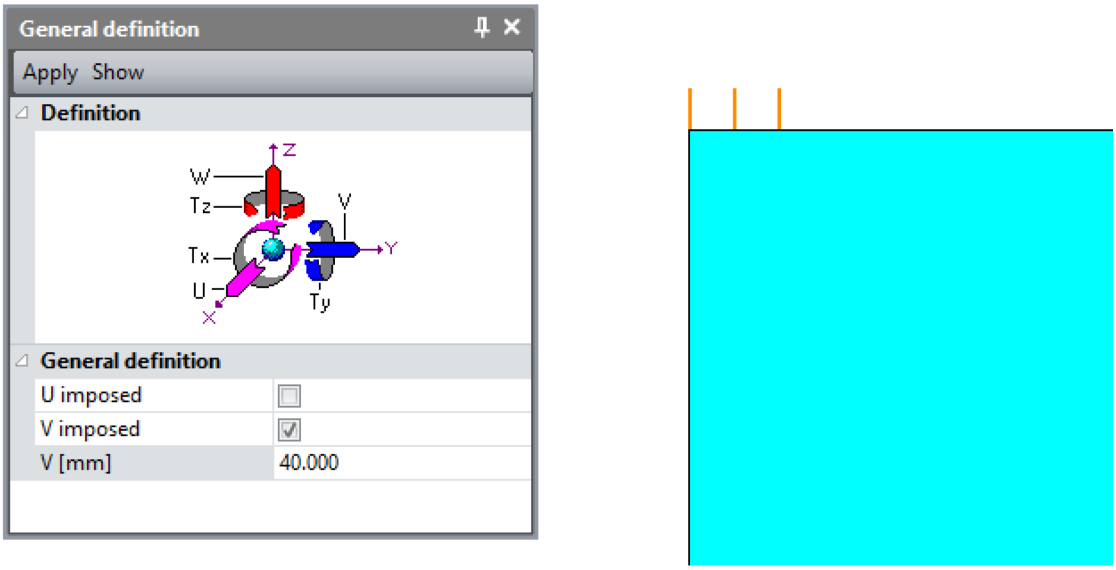


Figure 9: View of the software interface after application of the imposed displacement

<u>Load Case:</u>

As we impose a displacement, the uniform pressure is useless. However we made a copy from the previous model, therefore we must delete it.

1. Go to the ***LOADS tab***.
2. Select segment A.
3. Delete
4. With [F2], rename the loading set to "no load"

The active set cannot be deleted and the software imposes at least one active load set.

<u>Calculation parameters:</u>

1. Right-click on the model **Calculation #3** in the ***CALCULATION TREE*** and activate ***Analysis settings***:
2. In the **General parameters** tab:
   - Iteration process:
     Max number of increments: 20
     Max number of iterations per increment: 500
     Tolerance: 0,001
   - Solution method: 1- initial stresses
   - Algorithm type: Multi frontal
   - Analysis type: Standard
3. Validate using ***OK***.

In the **Boundary conditions** tab, the program automatically sets the factors to 1 (default value), divided by 20. Therefore, one gets 20 fractions of 40 mm as displacements steps.

Solve:

Now that all data are input, go to ***ANALYSIS***.

1. Click on ***Analysis manager***.
2. Select the model **Calculation #3**.
3. Select Create input files for the solver and calculate. Click on Validate.

4.3. *Alternative analyses*

4.3.1 *Linear elastic drained analysis with variable Young's modulus*

A non-linear elastic analysis can be carried out to assess the settlement under the footing in drained conditions. The increase of the modulus with depth is an important factor to take into consideration. Divide the model in several layers (2 m, 3 m and 5 m for example) and assess the drained modulus at the centre of each layer using the Ohde-Janbu empirical equation:

$$E = E^{ref}\left(\frac{\sigma_v}{\sigma^{ref}}\right)^N$$

with the following soil properties:

	γ_h (kN/m³)	E (MPa)	N	ν
Soil mass	20	10 at σ^{ref} = 100 kPa	0.7	0.33

4.3.2 *Non-linear elastoplastic analysis with variable undrained cohesion*

The increase of the undrained cohesion with depth is an important factor to take into consideration. Divide the upper part of the model in several thinner layers and assess the undrained cohesion at the centre of each layer using the following relationship:

$$\varphi_u \approx 0$$

$$c_u = \frac{1}{2}\left(\sigma'^{ini}_v + \sigma'^{ini}_h\right)\sin\varphi + c\cos\varphi$$

with the following soil properties:

	γ_h (kN/m³)	E_u (MPa)	N	ν	c_u (kPa)	φ_u (°)	K_0
Soil mass	20	50 at σ^{ref} = 100 kPa	0.7	0.45	22	35	0.5

Note 1: The plasticity concentrating in the first meter of soil, the layers with varying cohesion must be thin enough to obtain the desired effect on the solution.

Note 2: Use the Ohde-Janbu formula to assess the undrained stiffness of each of the model layers.

4.4. *Results*

Load controlled analysis

The convergence is not achieved for the 12th step. This indicates that the soil is ruptured. The limit pressure value is between 110 kPa and 120 kPa.

Displacement controlled with linear elastic drained analysis and variable Young modulus

The stress-displacement curve obtained by finite element analysis is compared with the benchmark solution. The results for various load application assumptions at the centre of the footing are presented in the figure below.

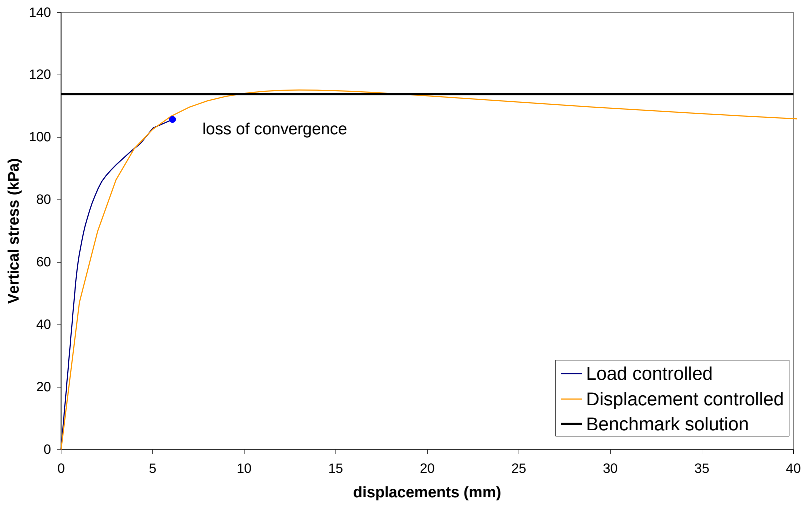


Figure 10: Stress-displacement curve at the centre point of the footing

Displacement controlled with non-linear elastoplastic analysis and variable undrained cohesion:

Results are presented below for a model where the upper 2 meters of soil, are divided in 3 equal sub-layers (each 667 mm thick). Figure 11 shows the obtained stress displacement results. Note that a discontinuity of the curve is observed at the intersection with the analytical bearing capacity corresponding to the cohesion of the first sub-layer.

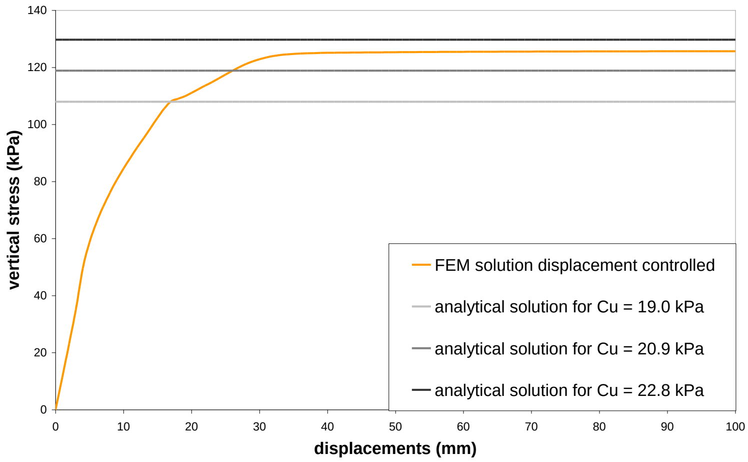


Figure 11: Stress-displacement curve at the centre point of the footing

Edited by :

itech

8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tél. : +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com