Tutorial 2g.05

Bearing capacity of a shallow foundation

Ref: CESAR-TUT(2g.05)-v2025.0.1-EN

1. INTRODUCTION

Version supported: v2025.0.

1.1. *Overview*

This tutorial is based on an academic example of bearing capacity analysis for footings, detailed in "Undrained bearing capacity factors for conical footings on clay", G. T. Houlsby and C. M. Martin, Geotechnique 53, No. 5, 513–520, 2003

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 for the design of shallow foundations. With the FEM, common results are settlements (SLS analysis) ; with an appropriate method, bearing capacity can be achieved and thus allow for ULS analysis.

1.2. *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.
- Generate a load control or displacement control analysis.
- Handle post-processing tools.

1.3. *Problem Specifications*

<u>General assumptions</u>

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

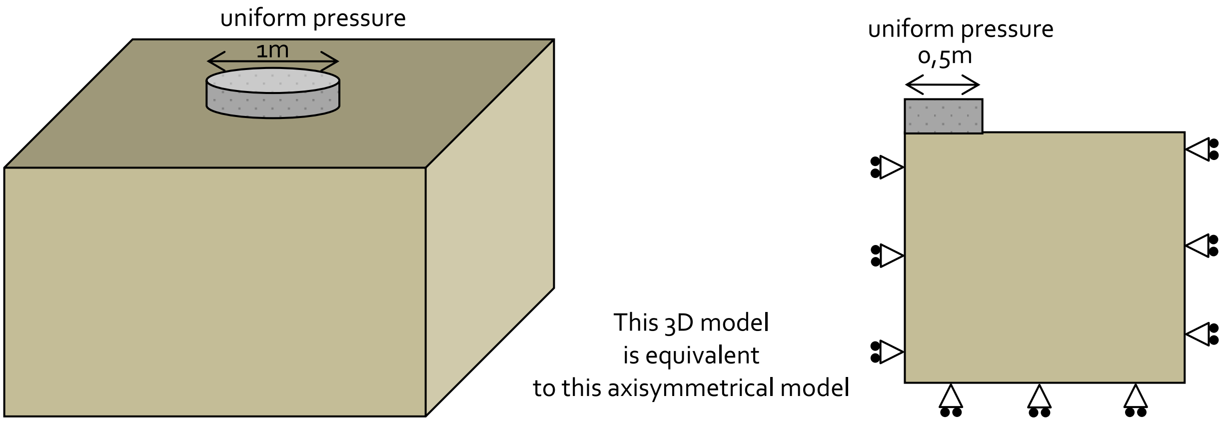


Figure 1: Context

Material properties

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

	γ_h (kN/m^3)	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.

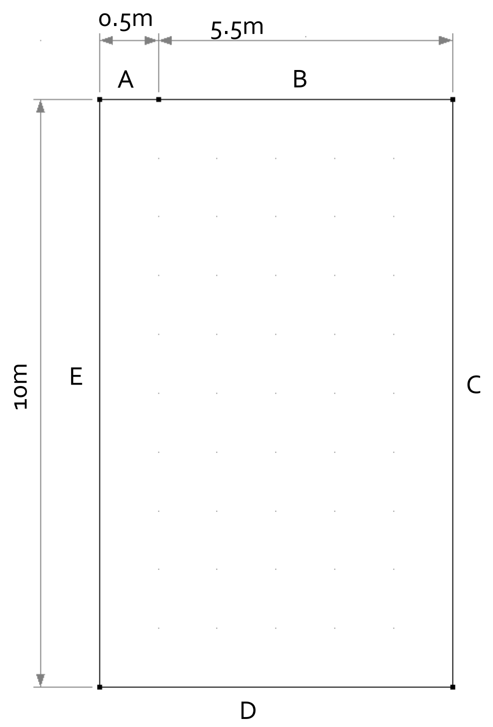


Figure 2: Geometry

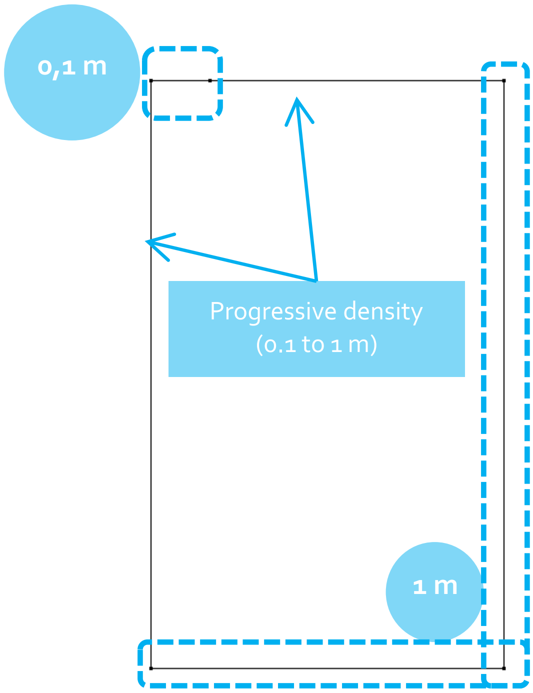


Figure 3: Density of meshing

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 ***Fixed length density*** to divide this segment with a fixed length. Enter **0,1 m** in the dialog box. Click on ***Apply***.

3. Select edges C and D. Click on ***Fixed length density*** to divide these segments with a fixed length. Enter **1 m** in the dialog box. Click on ***Apply***.

4. 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 near the foundation. As the division is interactive, this first click defines the position of the first division.
   - Repeat previous operation by click on edge E near the foundation.

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.

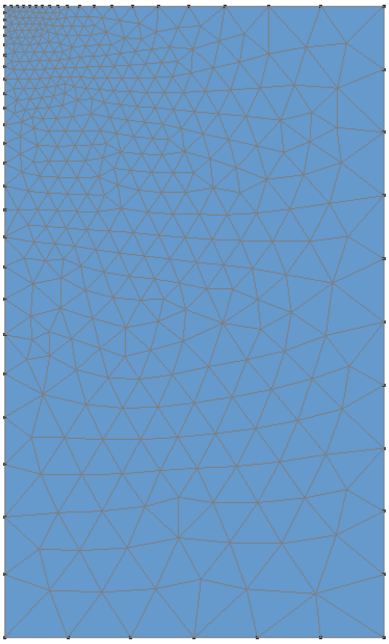


Figure 4: Example of mesh

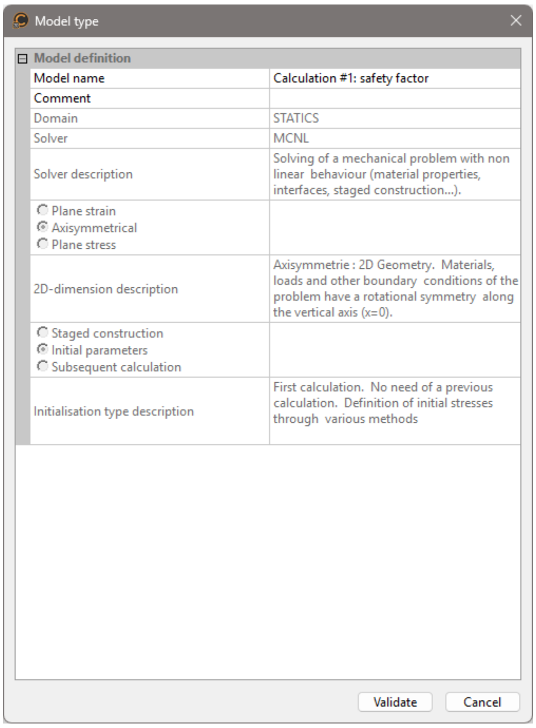
3. LIMIT PRESSURE ANALYSIS

3.1. *Model settings*

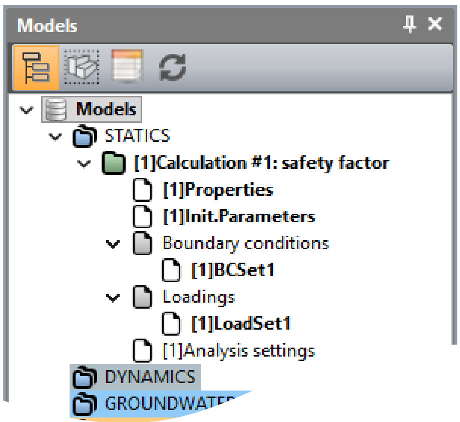
Model definition:

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 - Safety Factor 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"



The date tree is now as illustrated below.



Material properties for the solid elements:

We define the material library of the study.

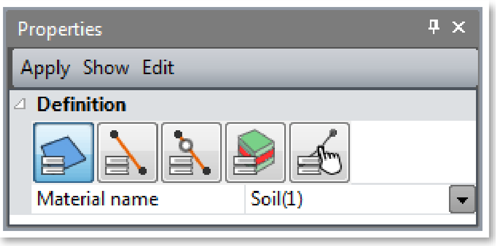
1. Click on Properties of surface bodies.
2. Give a name for the properties set name ("Soil" for example).
   - In **Elasticity parameters**, choose "Isotropic linear elasticity" and define ρ, E and ν.
   - In **Plasticity parameters**, choose "Mohr-Coulomb without hardening" and define c, φ and ψ.
3.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

	ρ (kg/m^3)	E (MN/m^2)	ν	c (MN/m^2)	φ (°)	ψ (°)
Soil mass	2000	50	0,45	0,02	0	0

Assignment of data 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 window is displayed. Click on ***Properties of surface bodies***.
3. Select the body of the soil mass on the model window and the setof parameter in the list.
4. Apply.



Initial stress field:

The initial stress field is initialized as an existing geostatic stress field; vertical stress is linked to the horizontal stress by K_0 value.

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

Height (m)	Volumic weight (MN/m^3)	K_{0_x}
0.	0,02	0,5

5. Validate.

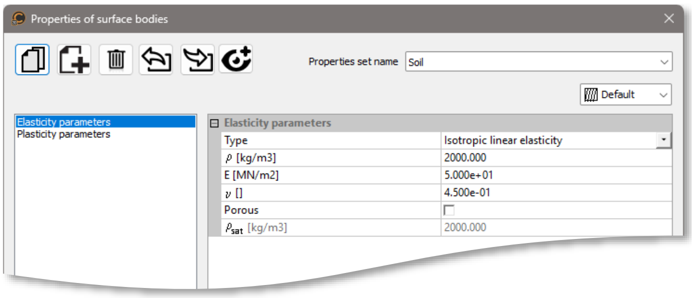


Figure 5: Toolbox for properties assignment

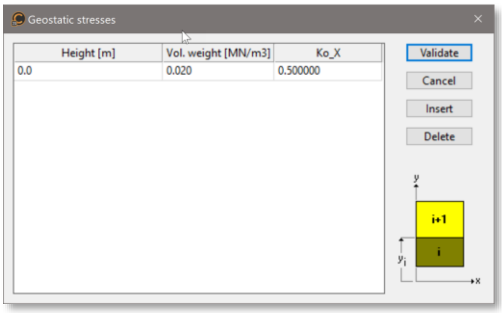
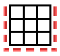


Figure 6: Toolbox for initial geostatic stresses input

<u>Boundary conditions:</u>

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.

<u>Loading:</u>

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.

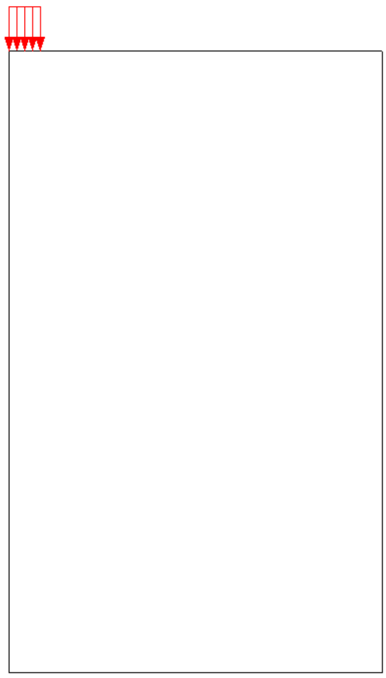
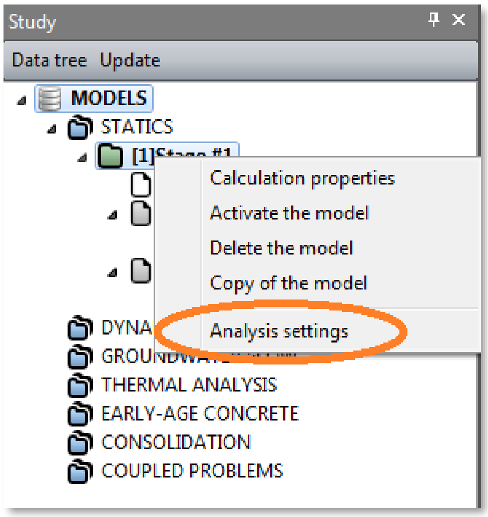
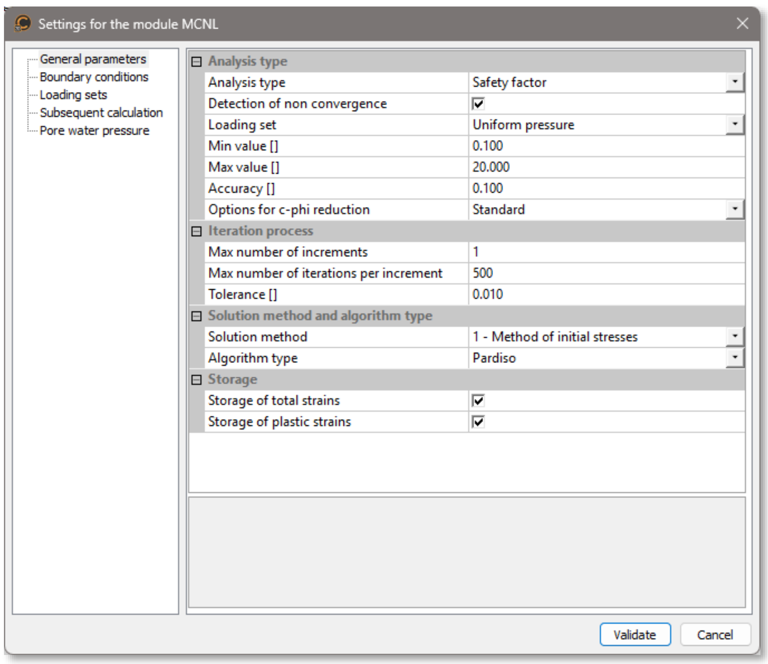


Figure 7: Display of the uniform pressure

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:
   - Analysis type: Safety factor
     Tick "Non-convergence detection"
     Set "Minimum Value" to 0.1
     Set "Maximum Value" to 10
     Set accuracy to 0.1
   - Iteration process:
     Max number of increments: 1
     Max number of iterations per increment: 500
     Tolerance: 0.01
   - Solution method: 1- Method of initial stresses
   - Algorithm type: Pardiso
3. Validate and close with ***OK***.

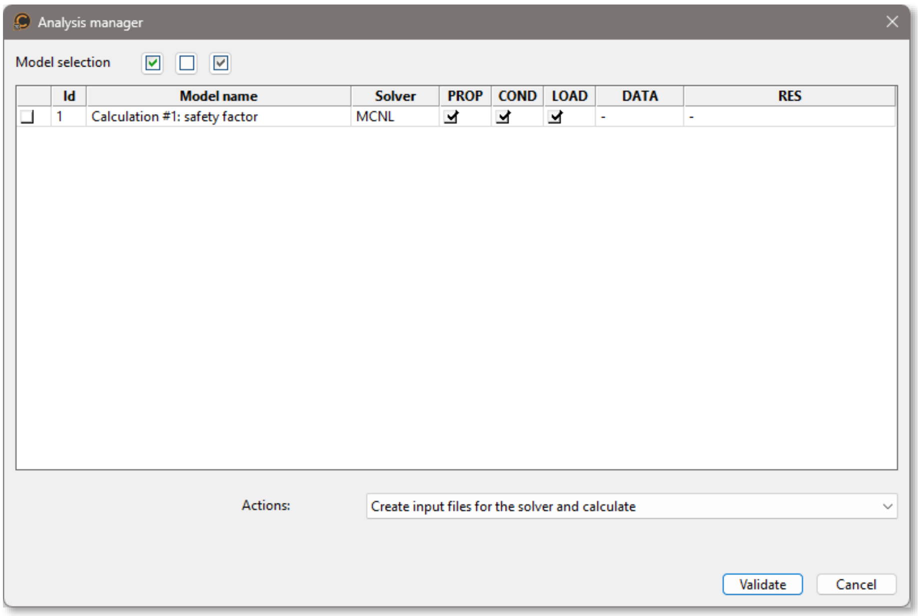


3.2. Solve

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 to activate the ***Deformed model*** and on to activate the ***Contour plot*** display.
3. Click on ***Type of results to display***.
   - In **Contour plots**, select |ε_p|, **Norm of plastic strain**,
   - Click on ***Apply***.
4. Click on ***Displacement settings***.
   - Select **Manual** as type of scale,
   - Set that a value of **100 mm** is represented by **2 m**,
   - ***Apply***.
5. 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***.

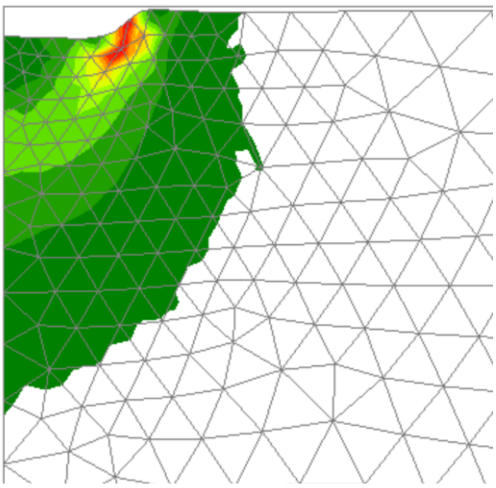


Figure 8: View of the field of plastic strains

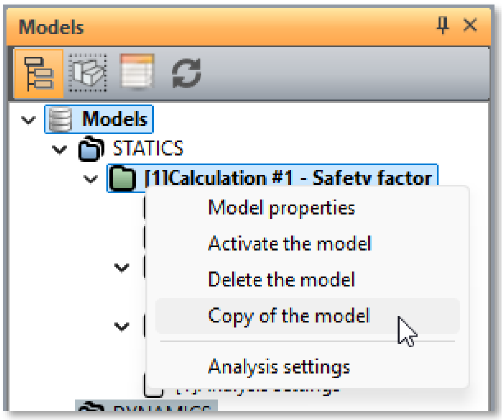
4. OTHER TYPES OF CALCULATIONS FOR DETERMINING THE LIMIT PRESSURE

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.

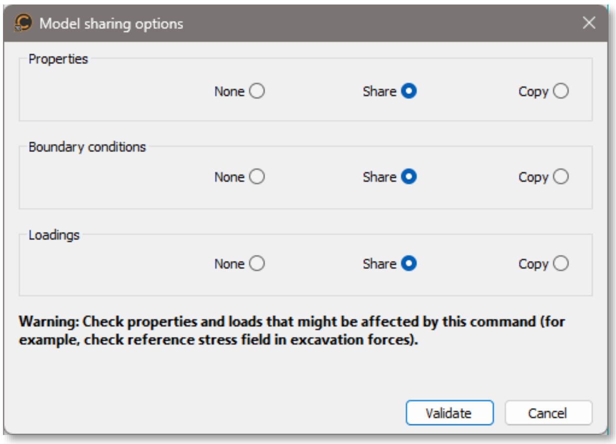
Model definition:

1. In the *CALCULATION TREE*, right-click on Calculation #1 and Copy of the model.



2. Define the model type:
   - Enter **Calculation #2 -Load Control** 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 items 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 Analysis settings in the CALCULATION TREE:
2. In the General parameters tab:
   - Analysis type: Standard
   - 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
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.

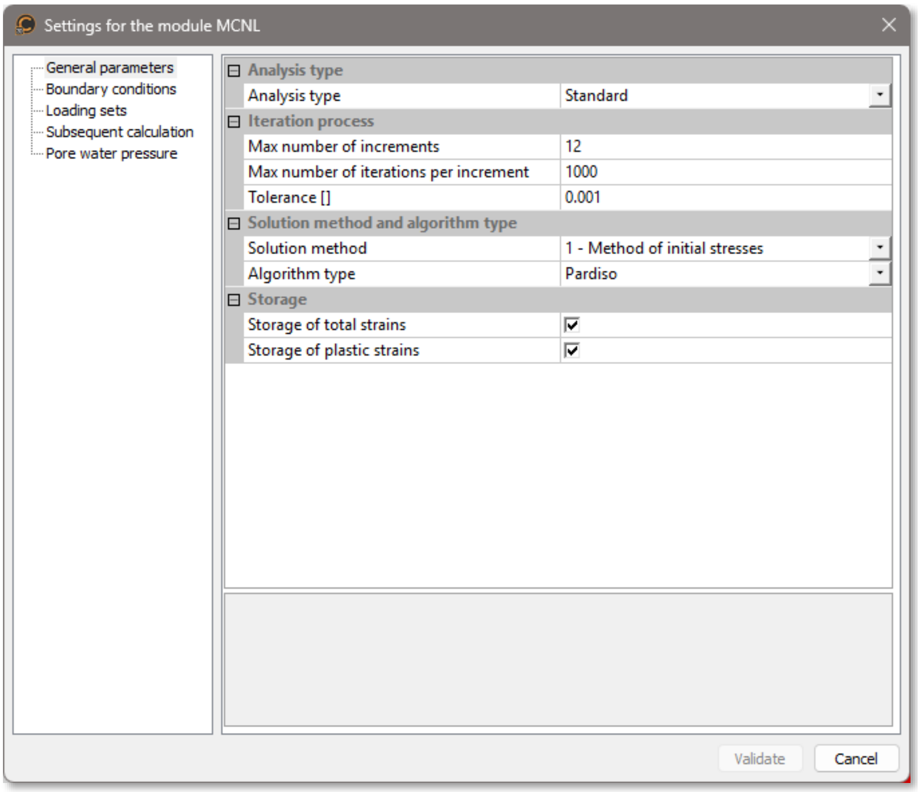


Figure 9: Toolbox for the setting of the calculation parameters

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 conclusion is that the limit pressure value is between 110 kPa and 120 kPa.

A higher precision on the value can be reached by defining a higher number of increments.

```
    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
         ----------------------------
```

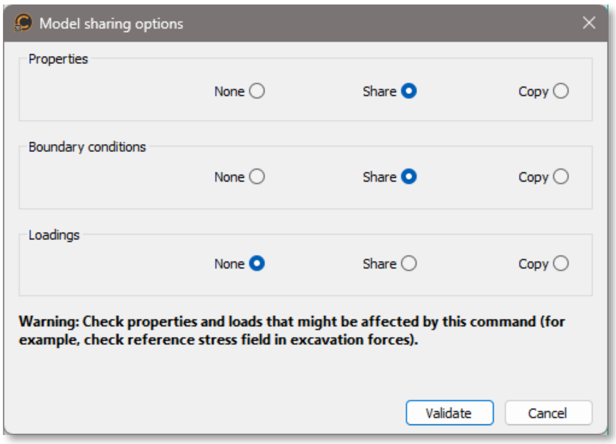
Figure 10: Extract of the listing at the end of the calculation

4.2. *Displacement controlled analysis*

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

Model definition:

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



Material properties:

No changes.

Initial stress field:

No changes.

Boundary conditions:

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. The set "Imposed displacement" is active
3. Click on General definition.
   - Tick **v imposed**.
   - Enter the value of **-40 mm**.
   - Select the segment A.
   - ***Apply***.

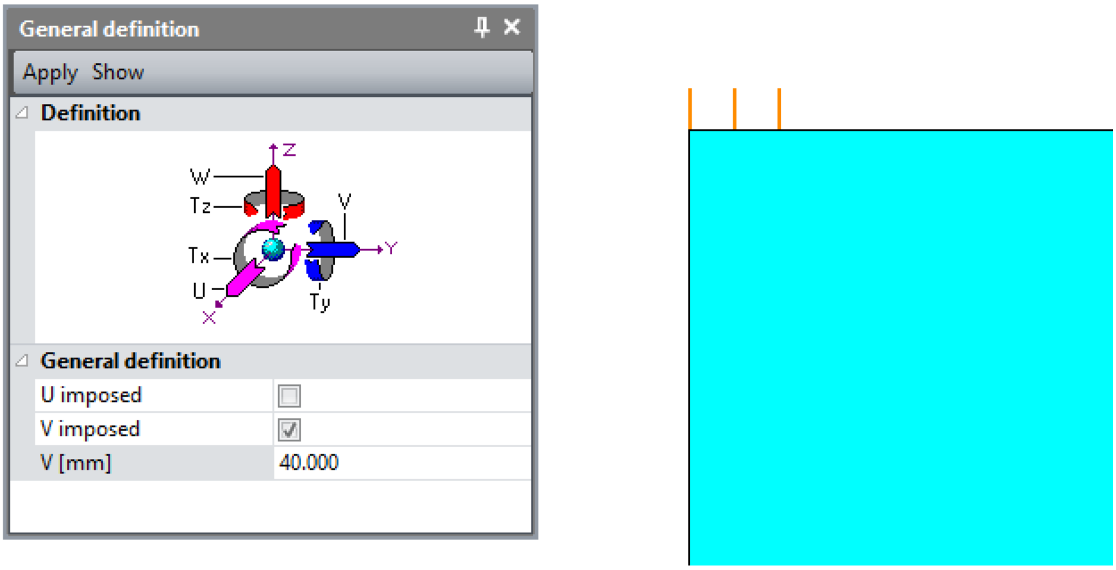


Figure 11: View of the software interface after application of the imposed displacement

<u>Load Case:</u>

No load set has to be defined here.

<u>Calculation parameters:</u>

1. Right-click Analysis settings in the CALCULATION TREE.
2. In the General parameters tab:
   - Analysis type: Standard
   - Iteration process:

Max number of increments:	20
Max number of iterations per increment:	1000
Tolerance:	0,001
   - Solution method: 1- initial stresses
   - Algorithm type: Pardiso
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. *Results*

As pointed earlier, the convergence is not achieved for the 12^{th} step. This indicates that the soil is ruptured. The limit pressure value is between 110 kPa and 120 kPa.

We can also plot the loading curve to illustrate the evolution of the displacement with the vertical stress applied.

1. Go to tab ***CHARTS***.
2. Activate ***Points***.
   - Select point with coordinates (0; 0) (the default one)
   - Give name "Centre"
   - ***Apply***:
3. Activate ***Set of points*** on the Entities for charts toolbar.
   - Select point "Center"
   - Enter the name "Group 1" in the *Set of points* box, then click on ***Add***.
4. Activate ***Parametric charts***.
   - Select **|u|** as *X-Axis*,
   - Select **σ_yy** as *Y-Axis*,
   - Click on ***Apply***.
   - Click to invert the orientation of the Y-axis.

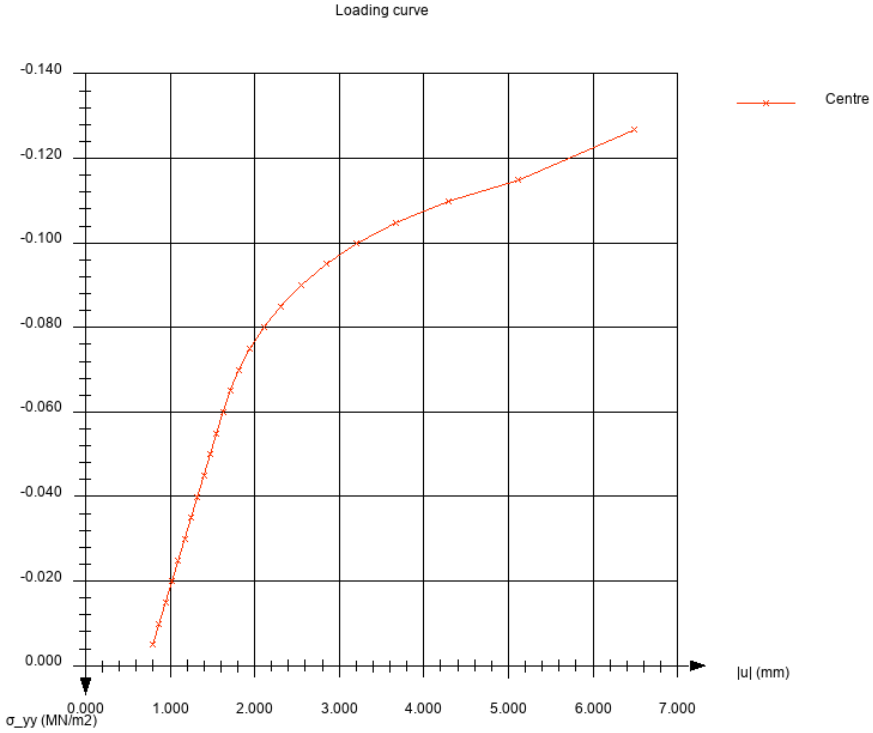


Figure 12: Loading curve in the case of the load-control analysis

Use the **Chart settings** to edit the titles, the axes and the colour of the curve.

The same curve is edited for the displacement control analysis.

The values plotted can be exported for external operations. Use the option « **_Export of data_**.

They are plotted on the same graph to compare them to the reference solution. This comparison is shown in the figure below.

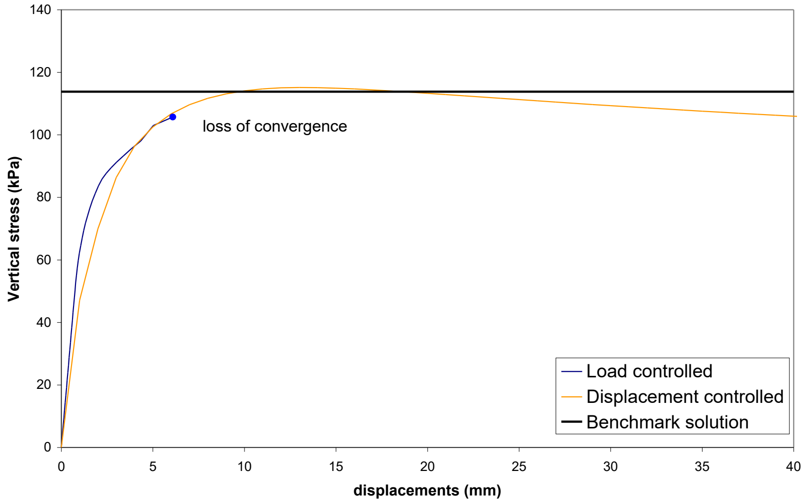


Figure 13: Stress-displacement curve at the centre point of the footing

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