Tutorial 2g.06

Slope stability analysis

Ref: CESAR-TUT(2g-06)-v2025.0.1-EN

itech – 8 quai de Bir Hakeim – 94410 Saint-Maurice - France
www.cesar-lcpc.com
Tél. : +33 1 49 76 12 59 – Fax : +33 1 42 83 33 84
cesar-lcpc@itech-soft.com

1. Introduction

Version: v2025.0.

1.1. Preview

This model is issued from the article "Slope stability analysis by finite elements", by Griffiths and Lane (in Geotechnique 49, n°3, pp. 387-403, 1999). This paper describes a procedure to reduce shear strength parameters of soils in a slope, also called "c-phi reduction" procedure.

An undrained clay slope with a thin weak layer is analysed. This thin layer of weak material runs a specified way inside the slope (see figure below). In this tutorial we will study the stability of this slope under self-weight. A factor of safety on its strength parameters will be calculated.

The safety factor is defined as follows:

$$c' = c/SRF \qquad\qquad \varphi' = Arctan(tan\varphi/SRF)$$

This factor is obtained when the finite element calculation reaches the latest converged solution, just before slope failure.

Thus, this so called "c-phi reduction" procedure is well suited for the ULS required by many design codes.

1.2. Problem Specifications

General assumptions

- Plain-strain problem,
- Static analysis,
- Non linear behaviour of the soil materials.

Geometry

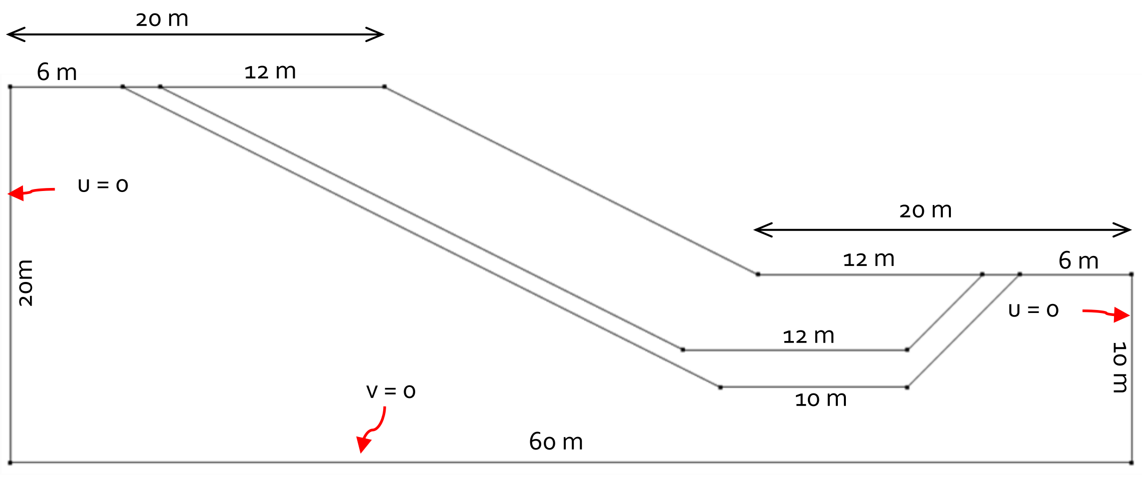


Figure 1: Situation of the problem

Material properties

The Tresca constitutive model will be used as plasticity criterion ($\varphi_u = \psi_u = 0$).

	ρ (kg/m^3)	E_u (MPa)	ν	c_u (kPa)	φ_u (°)
Slope	2000	100	0,3	50	0
Weak layer	2000	10	0,3	10	0

2. INPUTS FOR C/PHI REDUCTION ANALYSIS

2.1. *General settings*

1. Run CLEO2D.
2. Set the units in the menu **Preferences > Units**.
3. In the tree, select the leaf **General/Length** and set the unit **m** in the bottom left combo box.
4. In the tree, select the leaf **Mechanic/Force** and set the unit **kN** in the bottom left combo box.
5. In the tree, select the leaf **Mechanic/Displacement** and set the unit **mm**.
6. Click on ***Apply*** to close.
7. In ***Working plane***, set the visible grid to 1m (dX = dY = 2m)

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

2.2. *Geometry input*

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

Drawing of the geometry:

We first start with the drawing of the external limits of the slope.

1. Click on . The ***Points*** dialog box is displayed.
2. Enter **(0 ; 0)** as X and Y, and press ***Apply***.
3. Tick "Linked points" to generate the segments between the points.
4. Enter **(60 ; 0)**, and press ***Apply***. Segment A is created.
5. Enter **(60 ; 10)**, and press ***Apply***. Segment B is created.
6. Enter **(40 ; 10)**, and press ***Apply***. Segment C is created.
7. Enter **(20; 20)**, and press ***Apply***. Segment D is created.
8. Enter **(0 ; 20)**, and press ***Apply***. Segment E is created.
9. Enter **(0 ; 0)**, and press ***Apply***. Segment F is created.

Now we draw the inferior limit of the weak layer.

1. Click on . The ***Points*** dialog box is displayed.
2. Enter **(6 ; 20)** as X and Y, and press ***Apply***.
3. Tick "Linked points" to generate the segments between the points.
4. Enter **(38 ; 4)**, and press ***Apply***. Segment G is created.
5. Enter **(48 ; 4)**, and press ***Apply***. Segment H is created.
6. Enter **(54 ; 10)**, and press ***Apply***. Segment I is created.

Finally, we draw the superior limit of the weak layer.

1. Click on . The ***Points*** dialog box is displayed.
2. Enter **(8 ; 20)** as X and Y, and press ***Apply***.
3. Tick "Linked points" to generate the segments between the points.
4. Enter **(36 ; 6)**, and press ***Apply***. Segment J is created.
5. Enter **(48 ; 6)**, and press ***Apply***. Segment K is created.
6. Enter **(52 ; 10)**, and press ***Apply***. Segment L is created.

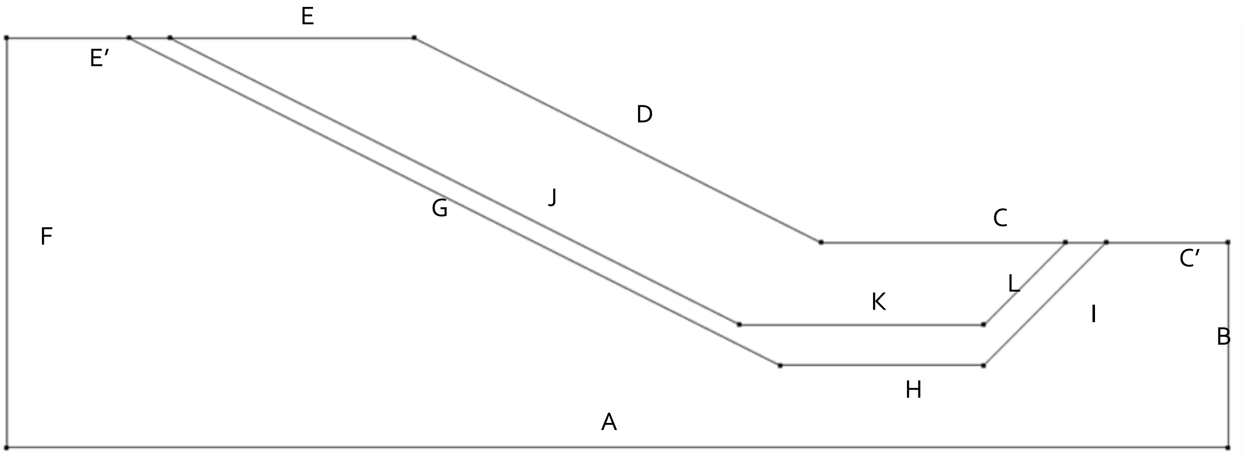


Figure 2: Geometry construction

Other methods could be used: define a grid of 2m x 2m and draw the lines with the mouse.

<u>Bodies definition:</u>

This step is facultative but eases the recognition of bodies when several have been generated.

1. Click on ***Body properties***.
2. Right click on the area corresponding to the slope mass. Enter **Soil** as a name. **Apply**.
3. Right click on the area corresponding to the weak layer. Enter **Weak layer** as a name. **Apply**.

User can display the body names on the geometry. Activate the option ***Body label*** on the selection toolbar. This can also be set in Preferences > Geometry Display, tick "Body label".

2.3. 2D Meshing

User will notice that this step is necessary but can be done at any moment in the construction process before the calculation, if users do not use the mesh as support of boundary conditions and loads (M-mode).

Density definition:

1. Go to the ***MESH*** tab on the project flow bar to start the definition of divisions along lines.
2. Select all edges. Click on ***Fixed length density***. to divide these segments with a fixed length. Enter **1 m.** in the dialog box. Click on ***Apply***.

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

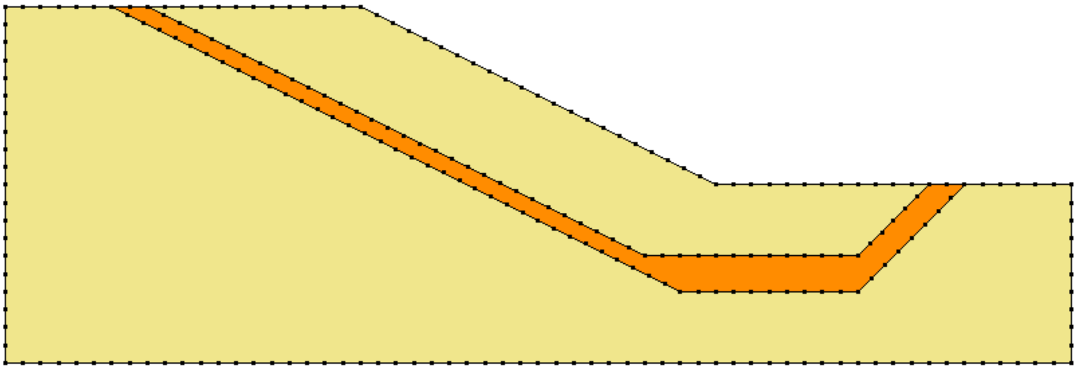


Figure 3: Example of mesh density

Meshing:

1. Select the areas corresponding to the layers.
2. Click on the ***Surface meshing*** tool . Chose **Quadratic** as interpolation type. Chose **Triangle** as element shape.
3. 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. The choice is made in *Preferences>Program settings*: linear = coarse, cubic = dense.

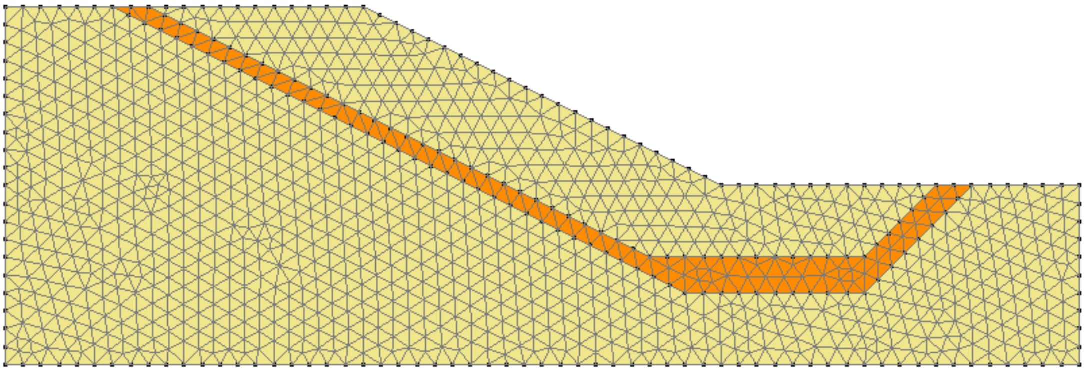
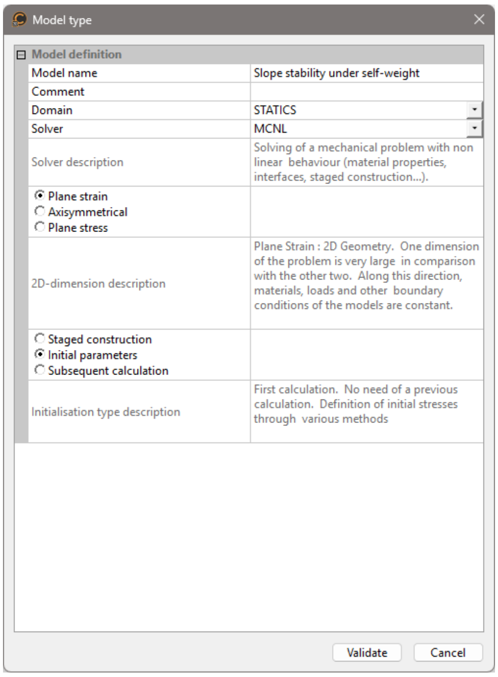


Figure 4: Example of mesh

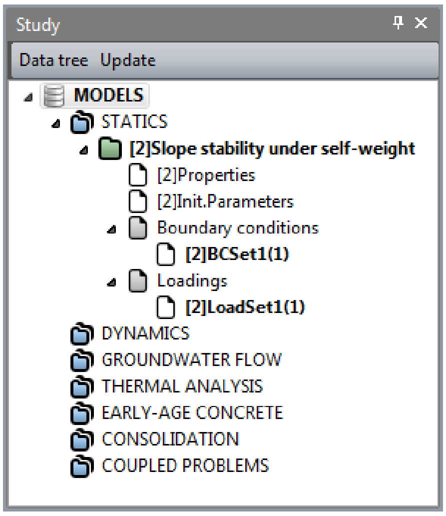
2.4. *Calculation properties for c/phi reduction analysis*

<u>Model definition:</u>

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 **Slope stability under self-weight** as "Model name".
4. Select **MCNL** as "Solver".
5. Tick **Plane strain** as model configuration, with **Initial parameters**.
6. Click on **Validate**.



The date tree is now as illustrated below.



Material properties for the solid elements:

We initially define the material library of the study in the ***PROPERTIES*** tab.

1. Click on ***Properties of surface bodies***.
2. Give a name for the properties set name ("Slope" for example).
3. In **Elasticity parameters**, chose "Isotropic linear elasticity" and define ρ, E and ν.
4. In **Plasticity parameters**, chose "Mohr-Coulomb without hardening" and define c, φ and ψ.
5. Click on to copy the previous properties set. Repeat operations 2 to 4 for the definition of the weak layer.
6. Click on ***Validate*** and ***Close***.

	ρ (kg/m³)	E (MN/m²)	ν	c (MN/m²)	φ (°)	ψ (°)
Slope	2000	100	0,3	0,05	0	0
Weak layer	2000	10	0,3	0,01	0	0

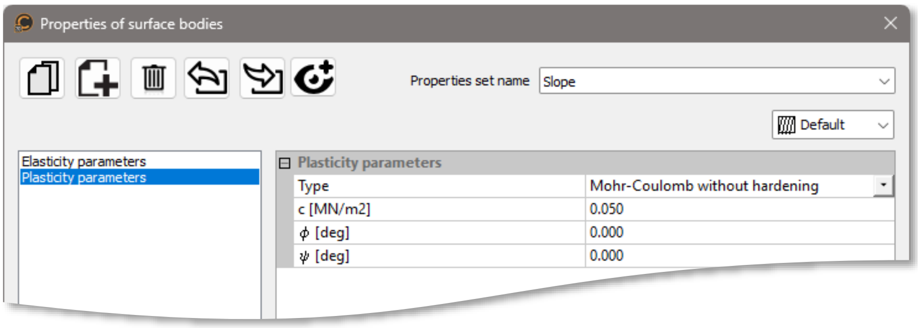


Figure 5: Toolbox for material library definition

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 a body on the model window and a setof parameter in the list.
4. ***Apply***.
5. Repeat the operations for all the bodies of the model.

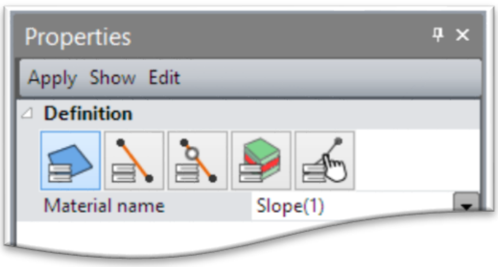


Figure 6: Toolbox for properties assignment

Boundary conditions:

1. Go to the ***BOUNDARY CONDITIONS*** tab.
2. On the toolbar, activate to define side and bottom supports. Supports are automatically affected to the limits of the mesh.
3. Facultative. It is possible to modify the default name assigned to the boundary conditions set. Press [F2]; enter **Standard supports** for example.

Load case:

1. Go to the ***LOADS*** tab.
2. On the toolbar, activate ***Gravity forces*** .
3. Select all the bodies.
4. Click on ***Apply***.
5. Facultative. It is possible to modify the default name assigned to the loading set. Press [F2] ; enter **Self-weight** for example.

Analysis settings:

When selecting "c-phi reduction" as **Analysis type**, user needs to select options between "Accelerated", Standard" and "Custom settings".

- Accelerated: the result will be calculated quicker but the failure mechanism will be slightly of lower quality;
- Standard: the result will be reached with a longer time but failure mechanism will be of better quality.

For the purpose of this tutorial, we select "accelerated".

1. In **ANALYSIS** tab, activate **Analysis settings** .
2. In the **General parameter** section, enter the following values (several values of the parameters are not accessible as depending of choices):
   - Analysis type: *c/phi reduction*
     Non-convergence detection ☑
     Minimum Value 0,1
     Maximum Value 5
     Precision 0,1
     Options for c-phi reduction: Accelerated
   - Iteration process:
     Max number of increments: 1
     Max number of iterations per increment: 200
     Tolerance: 0,01
   - Method of resolution: 12- initial stresses
   - Solver type: Pardiso
   - Storage
     Storage of total strains: ☑
     Storage of total strains: ☑
3. Close, ***Validate***.

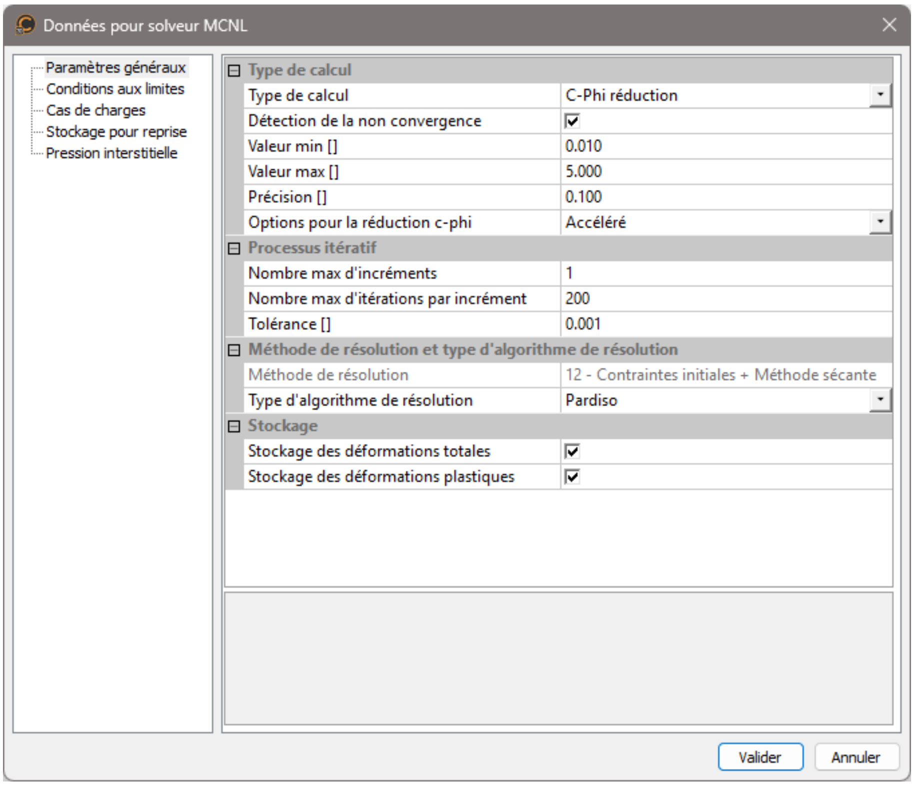
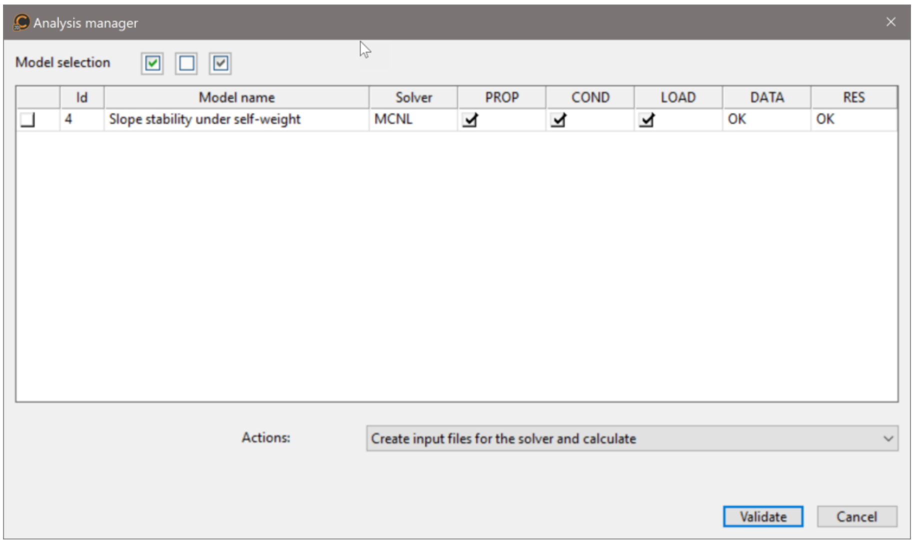


Figure 7: Toolbox for input of calculation parameters

3. SOLVE

1. Go to the **ANALYSIS** tab.
2. Click on ***Analysis manager***.
3. Select the model "Slope stability under self-weight".
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).

4. RESULTS

The result is the safety factor. It is displayed:

- In the **Output Window**
- In the listing file.

The calculated safety factor is 0,52.

1. Go to the **RESULTS** tab.
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_1,t,
   - Click on ***Apply***.
4. Click on ***Legend***.
   - In **Legend**, select **Contour plot**
   - Click on ***Apply***.

The figure shows that the failure mechanism is concentrated in the weak layer, as expected.

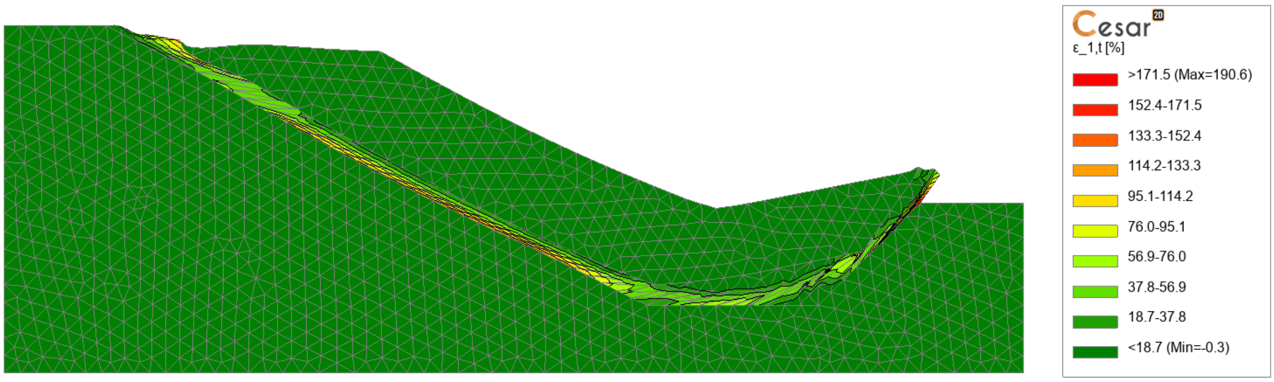


Figure 8: Plasticity and deformed mesh

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