Tutorial 2g.14

Stability analysis of a slope with water table

Ref: CESAR-TUT(2g.14)-v2025.0.1-EN

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

1. PREVIEW

This model is taken from Arai and Tagyo (1985), example#3, and it consists of a simple slope of homogeneous soil with pore pressure.

This tutorial presents two important aspects for geotechnical modelling:

- Modelling of a water table,

- Calculation of a safety factor on the global stability (so-called "c-phi reduction" procedure).

The distribution of the pore pressure is calculated with 2 different methods. The first method is based on the definition of a water table level and assumes that the pore pressure at a given point below the water table is calculated by measuring the vertical distance to the water table and multiplying it by the unit weight of the water (this is an approximation that assumes that the flow is horizontal throughout the soil mass). In the second method, the pressure distribution is calculated by using a steady-state or transient flow analysis.

Regarding the slope 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.

2. PROBLEM SPECIFICATIONS

General assumptions

- Static analysis under plain-strain conditions,
- Steady-state water table,
- Non-linear behaviour of the soil.

	ρ (kg/m³)	E (MPa)	ν	c (kPa)	φ (°)	ψ (°)
Slope	1882	100	0,3	42	15	0

Geometry

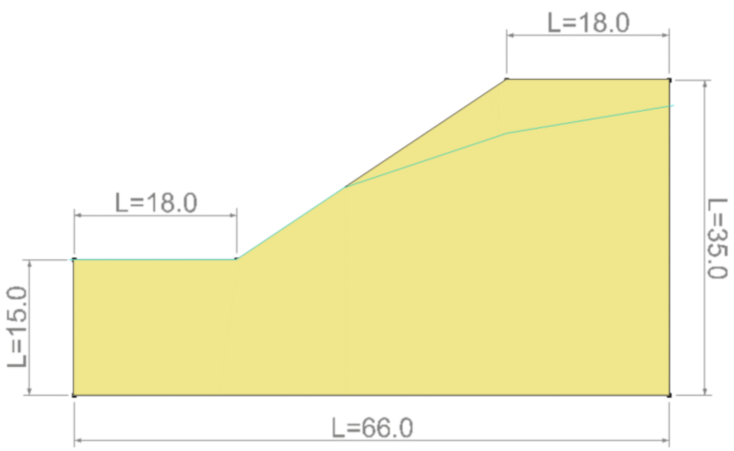


Figure 1: Situation of the problem

3. 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.

4. EDITION OF THE GEOMETRY AND THE MESH

4.1. *Geometry input*

A new project always 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 **(66; 0)**, and press ***Apply***. Segment A is created.
5. Enter **(60; 35)**, and press ***Apply***. Segment B is created.
6. Enter **(48; 35)**, and press ***Apply***. Segment C is created.
7. Enter **(18; 15)**, and press ***Apply***. Segment D is created.
8. Enter **(0; 15)**, and press ***Apply***. Segment E is created.
9. Enter **(0; 0)**, and press ***Apply***. Segment F is created.

Other methods could be used:

1. Define a grid of 1 m x 1 m and draw the lines with the mouse.

2. Use of "right click" to get information on an existing node and modify coordinates.

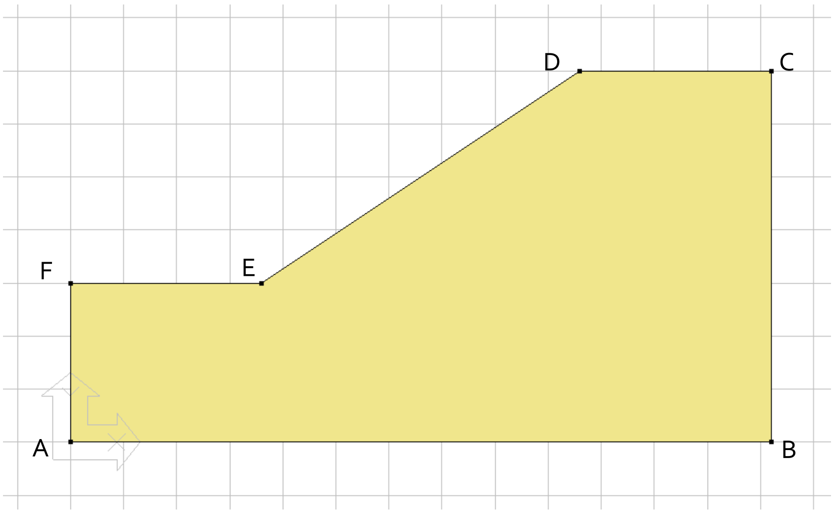


Figure 2: Geometry construction

Bodies definition:

This step is facultative but it eases the recognition of bodies if more than one has been generated.

1. Click on **_Body properties_**.
2. Right click on the area corresponding to the slope mass. Enter **Soil** as a name. **Apply**.

User can display the body names on the geometry using feature **_Name of the body_** on the toolbar of the workspace.

4.2. 2D Meshing

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 2 **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.

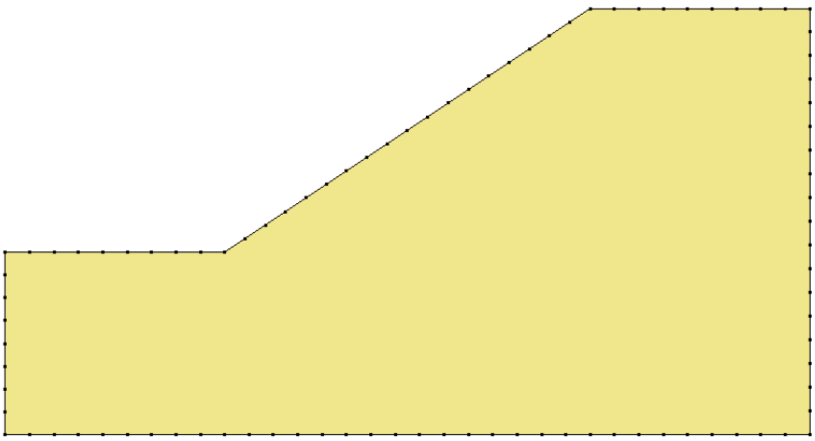


Figure 3: Example of mesh density

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.

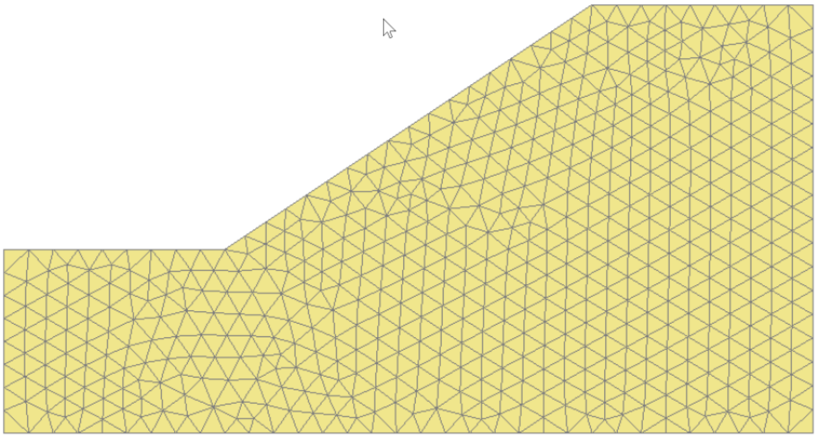


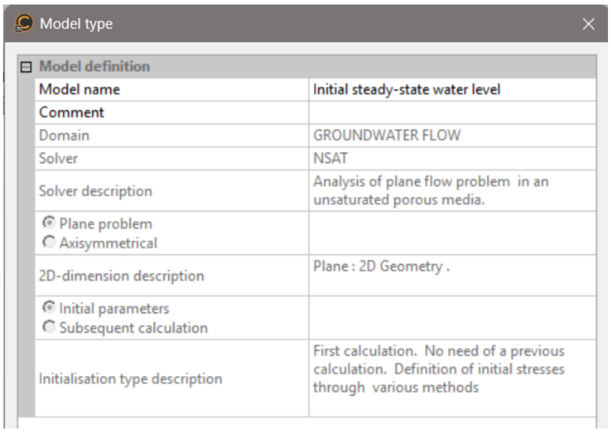
Figure 4: Example of mesh

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 interpolation = coarse, cubic interpolation = dense.

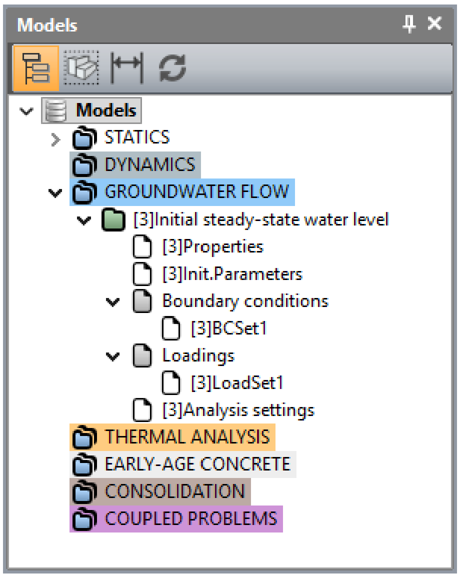
5. CALCULATION PROPERTIES FOR THE STEADY-STATE ANALYSIS

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 GROUNDWATER FLOW. Click on ***Add a model***. A new toolbox is open for definition of the Model.
3. Enter **Initial water level** as "Model name".
4. Select **NSAT** as "Solver".
5. Tick **Plane problem** as model configuration.
6. Click on ***Validate***.



The model tree is now as below:



Material properties for the surface bodies:

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 **Flow parameters**, chose "Flow in porous media" and define K_x, K_y, K_{xy} and Ce. Note here that Ce (accumulation coefficient) is set to zero when steady-state condition analysis.
4. Click on ***Validate*** and ***Close***.

	Kx (m/sec)	Kx (m/sec)	Kx (m/sec)	Ce (1/m)
Slope	10^{-5}	10^{-5}	0	0

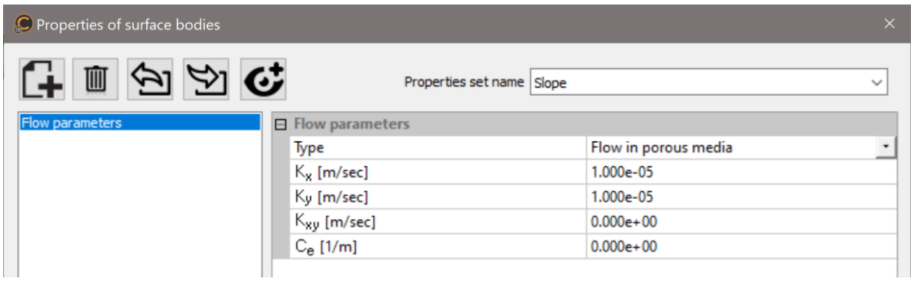


Figure 5: Toolbox for material library edition

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 for surface bodies***.
3. Select the body of the slope on the model window and a set of parameters in the list.
4. ***Apply***.

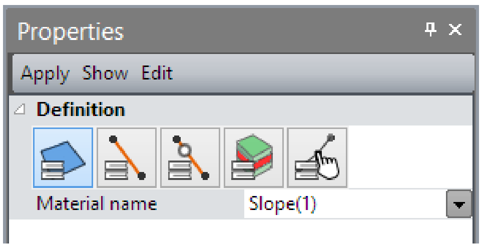


Figure 6: Toolbox for properties assignment

Boundary conditions:

1. Go to the ***BOUNDARY CONDITIONS*** tab.
2. Activate $\bar{h}$ Constant water head. Set a value of 32 m on the left of the slope and a value of 15 m at the bottom of the slope (as illustrated in the figure below).
3. ***Apply*** to validate each constant water head.

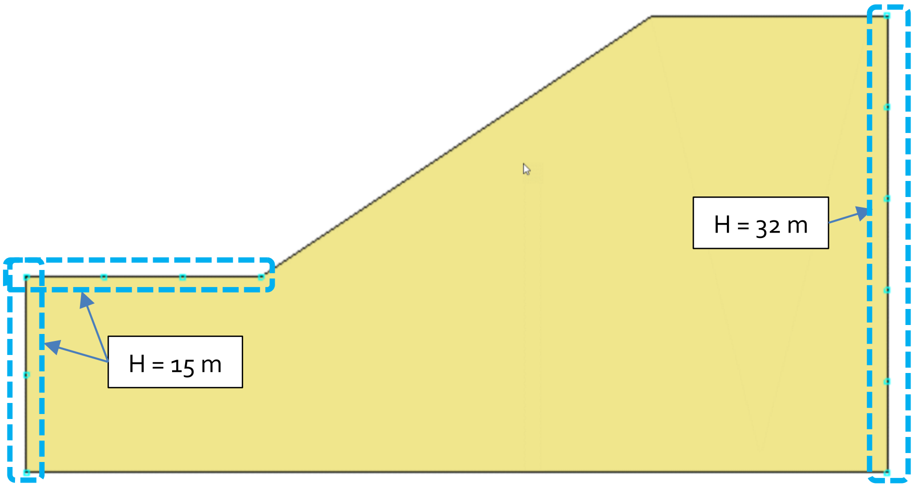


Figure 7: Applied water heads

Facultative. It is possible to modify the default name assigned to the boundary conditions set. Press [F2]; enter **Water heads** for example.

Analysis settings:

1. In **ANALYSIS** tab, activate **Analysis settings** .
2. In the **Time steps** section, select the "Steady state".
3. Other settings are set with their default values.
4. Click ***Validate*** to close.

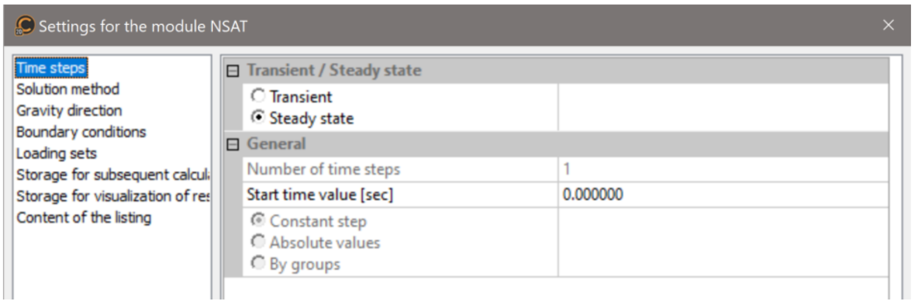


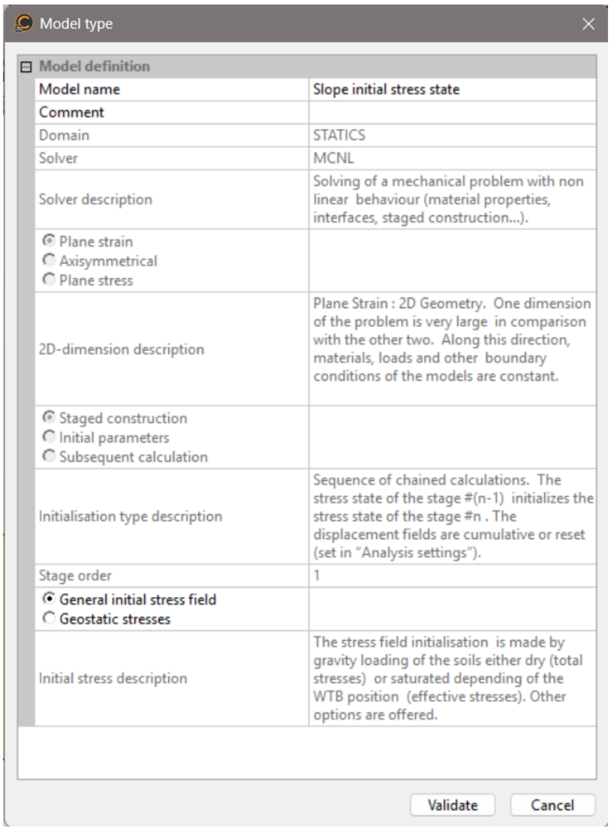
Figure 8: Toolbox for input of calculation parameters

6. Calculation properties for the slope stability analysis

6.1. *Slope initialisation*

Model definition:

1. On the right side of the working window, the "Tree view" window 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 **Slope initial stress state** as "Model name".
3. Select **MCNL** as "Solver".
4. Tick **Plane strain** as model configuration.
5. Tick **Stage construction**. This is mandatory for the definition of the sub-stage for the c-phi reduction analysis.
6. Tick **General initial stress field** as we define the water-table with the option WTB.
7. Click on ***Validate***.



Material properties for the solid bodies:

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

1. Click on  ***Properties for surface bodies***.
2. Give a name for the properties set name ("Slope" for example).
3. In **Elasticity parameters**, select "Isotropic linear elasticity"
   - input ρ, E and ν.
   - activate "Porous" and input ρ_{sat}
4. In **Plasticity parameters**, select "Mohr-Coulomb without hardening" and define c, φ and ψ.
5. Click on ***Validate*** and ***Close***.

	ρ (kg/m³)	E (kN/m²)	ν	c (kN/m²)	φ (°)	ψ (°)
Slope	1882	10 000	0,3	42	15	0

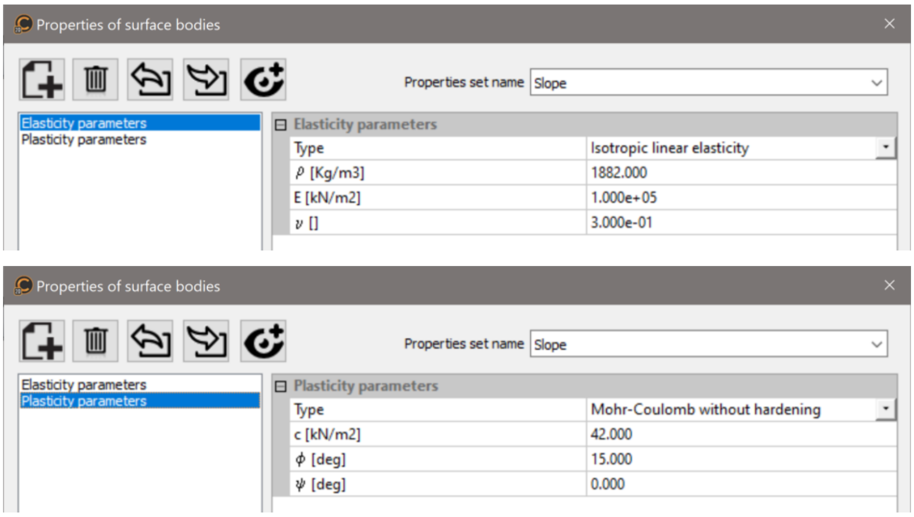


Figure 9: Toolbox for material library definition

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 the body on the model window and a set of parameter in the list.
4. ***Apply***.

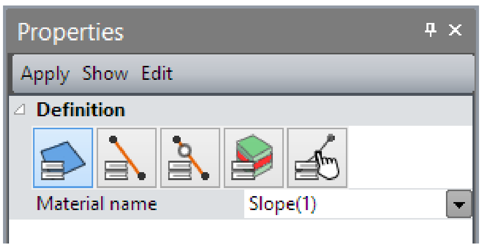


Figure 10: Toolbox for properties assignment

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.

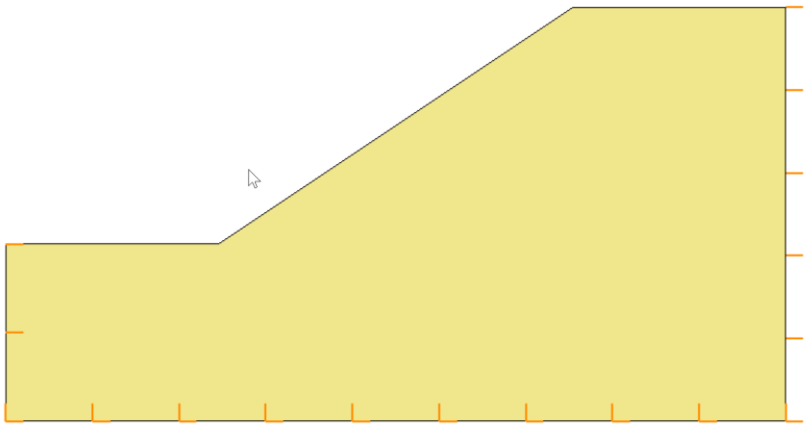


Figure 11: display of the boundary conditions

Load case:

We define here the hydrostatic field using the water head field calculated in the steady-state flow analysis.

1. Go to the ***LOADS*** tab.
2. Activate ***WTB – Hydraulic head variation***.
   - Select "Model" as **Initial hydrostatic field**.
   - Select the previously defined model "Initial steady-state water level".
   - ***Validate***. The activation of the loading is identifiable by the mention "WTB – Hydraulic head variation - Ha (Model: Initial steady-state water level, Step: 1)" displayed at the bottom left of the workspace.

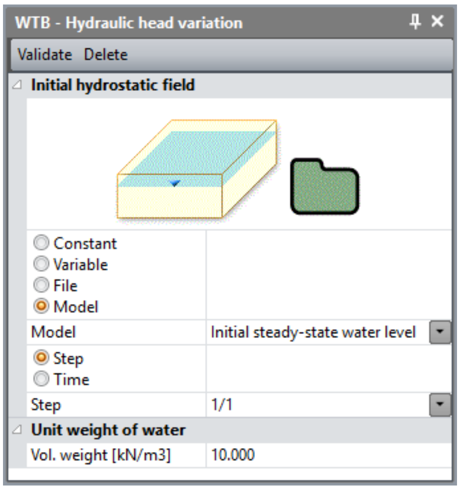


Figure 12: Water-table definition using the steady-state water head field

Alternative for defining the WTB position and resulting stress field:

When the free surface line is known (height were p =0), user can input it.

1. Go to the ***LOADS*** tab.
2. Activate ***WTB – Hydraulic head variation***.
   - Select "Variable" as **Water head input**.
   - The water table is defined by 5 points: (0; 15), (18; 15), (30; 23), (48; 29), (66; 32). These points may be edited in the grid or defined by direct click on the workspace using the option [P*].
   - ***Validate***.

User may also define bodies below the water table that are dry. Activate the option ***WTB – Loaded bodies***., select the dry bodies and declare them as inactivated.

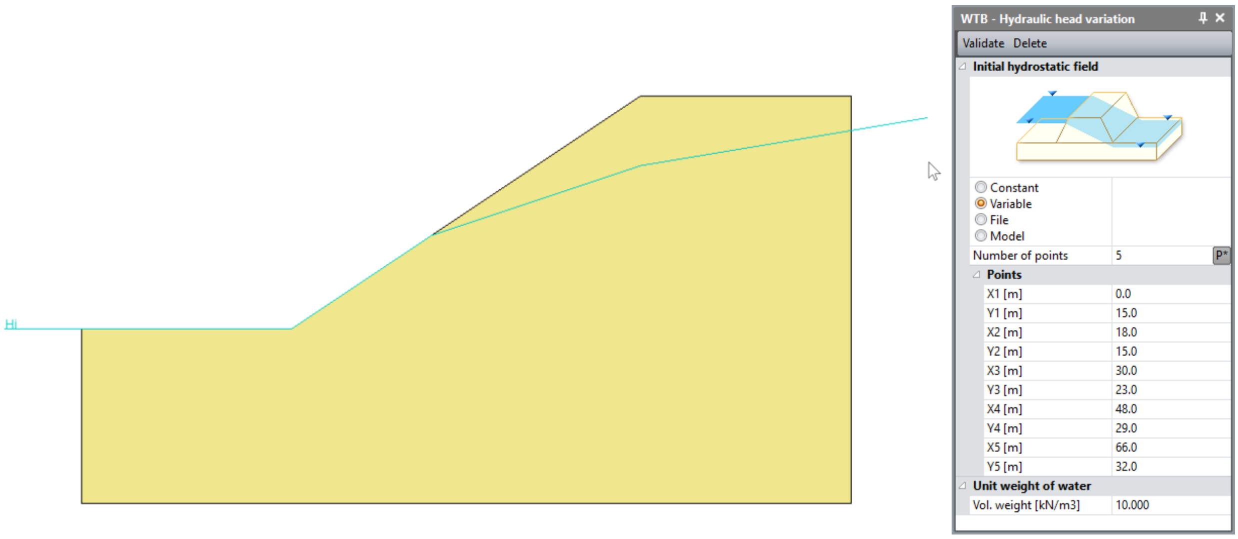


Figure 13: Alternative for the WTB definition

Analysis settings:

1. In **ANALYSIS** tab, 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: 1000
     Tolerance: 0,001
   - Method of resolution: 1- initial stresses
   - Solver type: Pardiso
   - Storage
     Storage of total strains: ☑
     Storage of total strains: ☑
3. Click ***Validate*** to close.

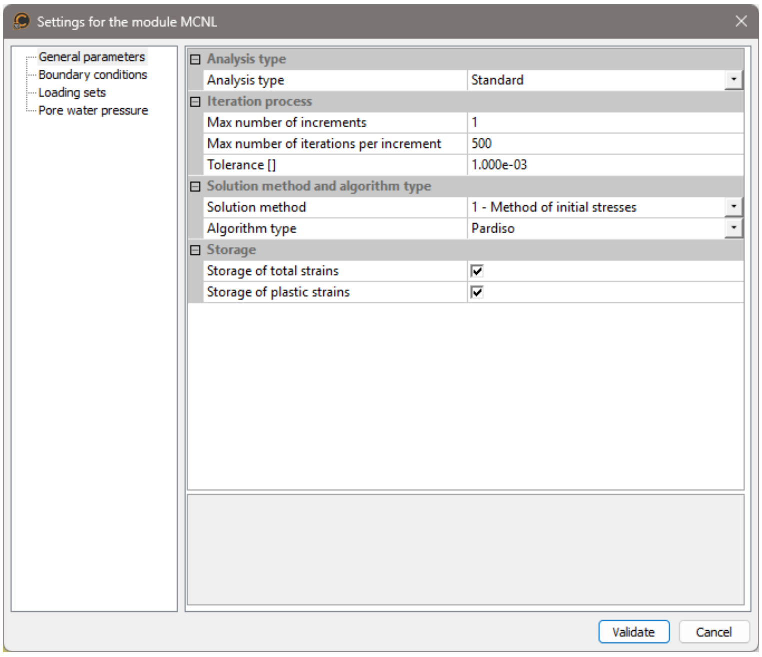


Figure 14: Toolbox for input of calculation parameters

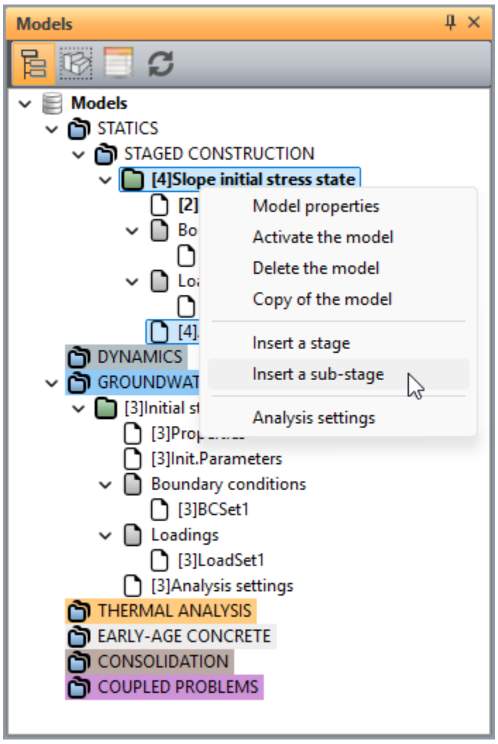
6.2. C-phi reduction process

The c-phi reduction starts from the existing stress field state.

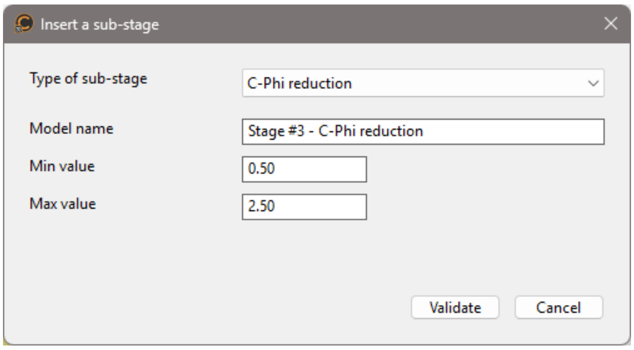
The edition of this analysis is resumed by the generation of a sub-staged, with copy of the same properties and boundary conditions (no unbalancing loads).

Model definition:

1. In the study tree, right click on the previous model. In the displayed list, select ***Insert a sub-stage***.



2. A toolbox is displayed for edition:
   - The type of the sub-stage: select c-phi reduction;
   - The name of the model;
   - The minimal and maximal values of the search interval of the safety factor;
   - ***Validate***.



Properties, boundary conditions and loadings

As default behaviour; properties and boundary conditions are copied and shared.

There is no additional load to be applied as we are looking for the ultimate stability state of the previous model by reduction of the shear properties of the soil.

Analysis settings:

The Les propriétés de calcul sont automatiquement adaptées, mais l'utilisateur peut venir modifier la méthode de calcul proposée :

- Méthode 1 (standard, par défaut) :
    - o Méthode de résolution : contraintes initiales
    - o Nombre d'itérations : 500
    - o Précision : 0,001
- Méthode 2 (accélérée) :
    - o Méthode de résolution : contraintes initiales + méthode sécante
    - o Nombre d'itérations : 200
    - o Précision : 0,001
- Méthode 3 (personnalisée) :
    - o Méthode de résolution : « libre »
    - o Nombre d'itérations : « libre »
    - o Précision : « libre »

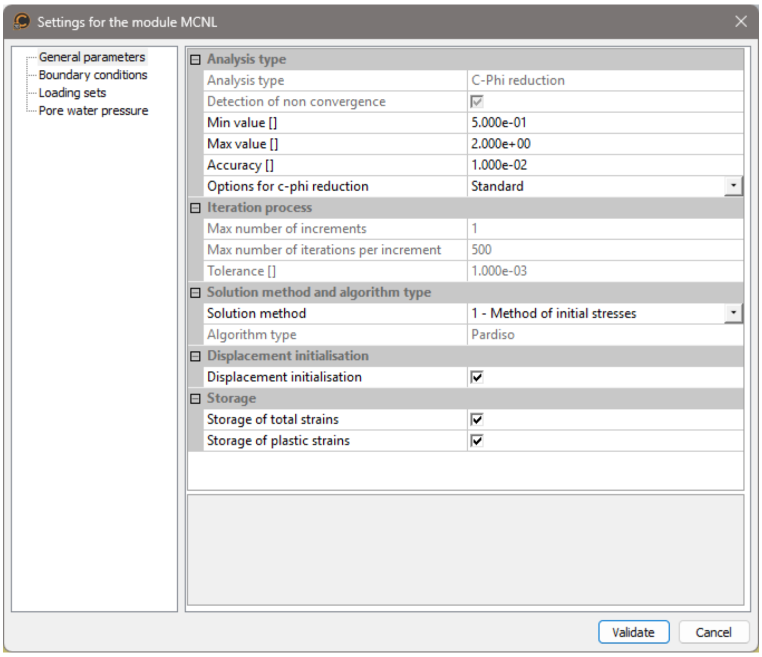


Figure 15: Example of calculation parameters with « standard » settings

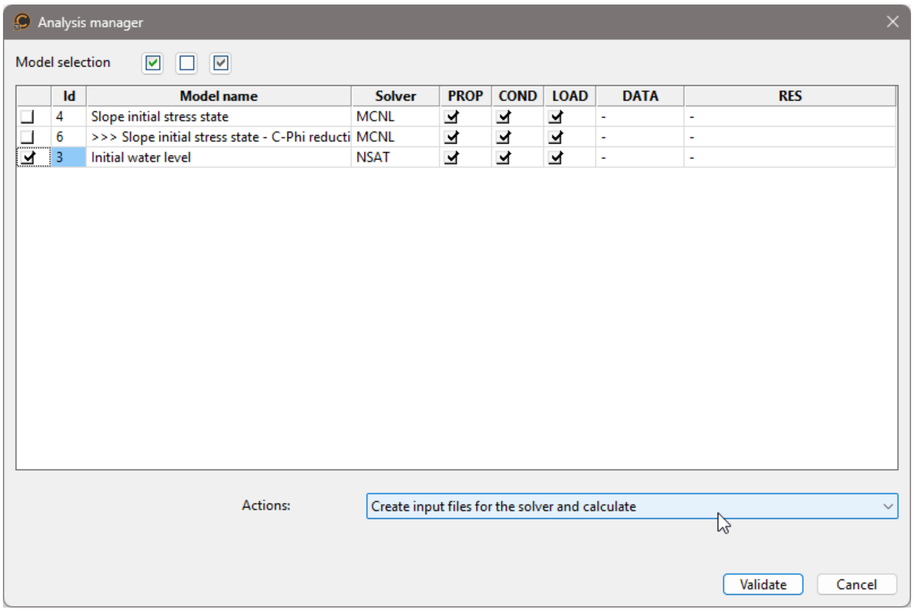
7. Solve

The mechanical calculations depend on the result of the flow rate calculation as the water level was defined using the "Initial water level" model.

Therefore, this later must be calculated first, then the two mechanical models can be calculated.

The GROUND WATER FLOW models are identified in the tree by the blue colour.

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



6. Click on ***Analysis manager***.
7. Select the models "Slope initial stress state" and "C-phi reduction ».
8. Select **Create input files for the solver and calculate**. Click on **Validate**.
9. 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).

8. Results

8.1. Steady-state flow analysis

The main result is the water head field and the fere surface (where hydrostatic pressure = 0).

Display of the water head field.

1. Go to the **RESULTS** tab.
2. Click on ***Contour plot*** to activate the display of contour plots.
3. Click on ***Type of contour plot*** to select the result to be displayed.
   - Select **H**
   - Click on ***Apply***.
4. Click on ***Contour plot settings***.
   - In *Contour plot settings*
     - Activate
     - Style: Areas
   - In *Contour plot*
     - Number of contours: 10
     - Color scheme: Groundwater
   - Click on ***Apply***.
5. Click on ***Legend***.
   - In *Legend*, select **Contour plot**
   - Click on ***Apply***.

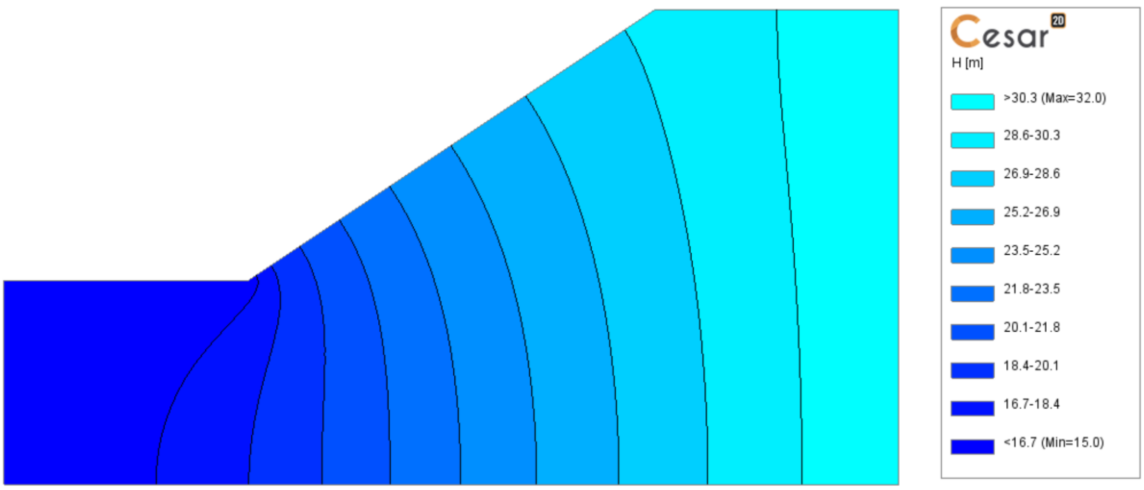


Figure 16: Display of the water head H

Display of the free surface (trick).

1. Go to the **RESULTS** tab.
2. Click on ***Contour plot*** to activate the display of contour plots.
3. Click on ***Type of contour plot*** to select the result to be displayed.
   - Select **P**
   - Click on ***Apply***.
4. Click on ***Contour plot settings***.
   - In *Contour plot settings*
     - Inactivate
     - Style: Areas
   - In *Contour edges*
     - Activate
     - Color set: Black
   - In *Scale*
     - Select "Manual"
     - Set "Min" to -0.001 MN/m²
     - Set "Max" to 0.001 MN/m²
   - In *Contour plot*
     - Number of contours: 10
     - Color scheme: Groundwater
   - Click on ***Apply***.
5. Click on ***Legend***.
   - In **Legend**, select **None**
   - Click on ***Apply***.

Figure 17: Display of the free surface (p = 0)

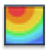
8.2. *Stability analysis*

The main result is the safety factor. It is displayed:

- In the **Output Window**
- In the listing file,
- At the top of the working space.

The calculated safety factor is 1,11.

Display of the rupture mechanism in the slope.

1. Go to the **RESULTS** tab.
2. Click on  ***Contour plot*** to activate the display of contour plots.
3. Click on  ***Type of contour plot*** to select the result to be displayed.
    - Select **ε_1,p**
    - Click on ***Apply***.
4. Click on  ***Contour plot settings***.
    - In *Contour plot settings*
        - Activate
        - Style: Areas
    - Click on ***Apply***.

The figure shows the classic "circular" rupture mechanism on the deformed mesh.

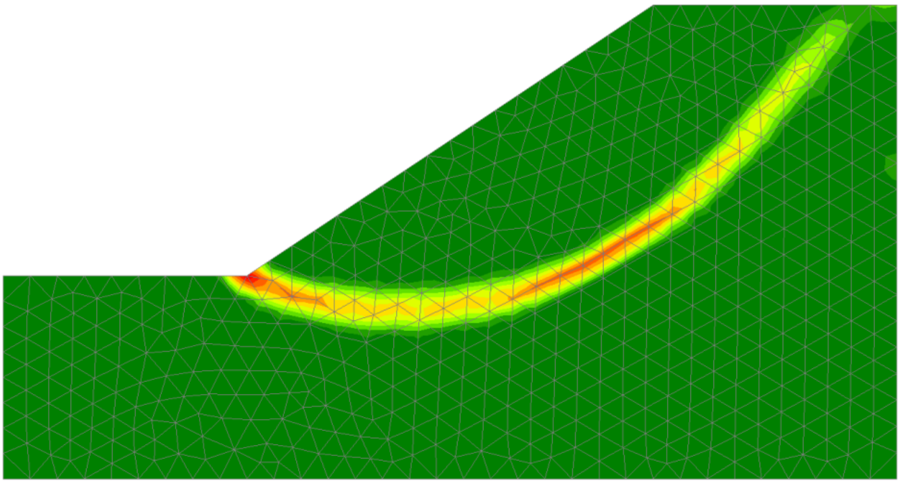


Figure 18: Total strains in the slope after c-phi reduction ("standard" settings)

9. APPENDIX : FACTEURS D'INFLUENCE SUR LES RESULTATS

All the results presented below lead to the same safety factor of 1.1.

The influence of the method and the precision of the iterative process are presented, followed by the influence of the mesh.

9.1. *Influence of the iterative process*

An alternative to the previous analysis will use the "Accelerated" settings.

In this context, the calculations are quicker (22'' with a computer equipped with Core i7 – 32 Go of RAM). The safety coefficient of 1,1 is reached but the failure mechanism is less identified (see figure below).

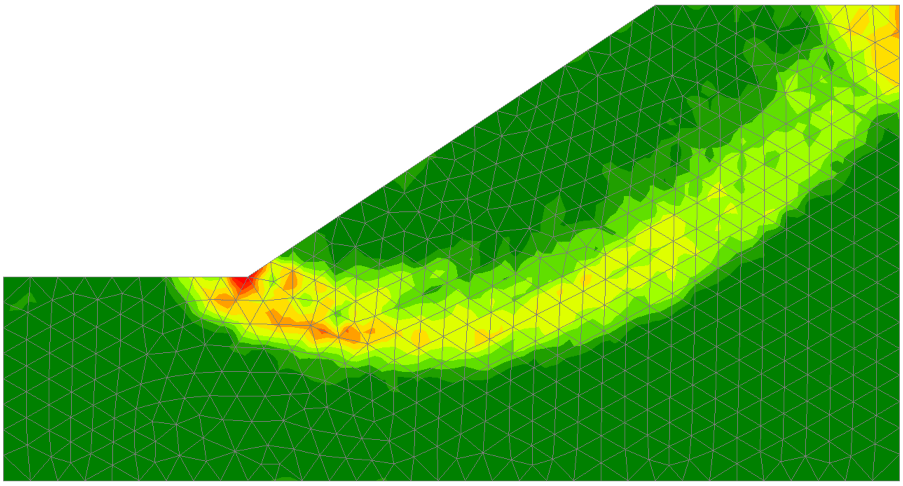


Figure 19: Total strains in the slope after c-phi reduction ("accelerated" settings)

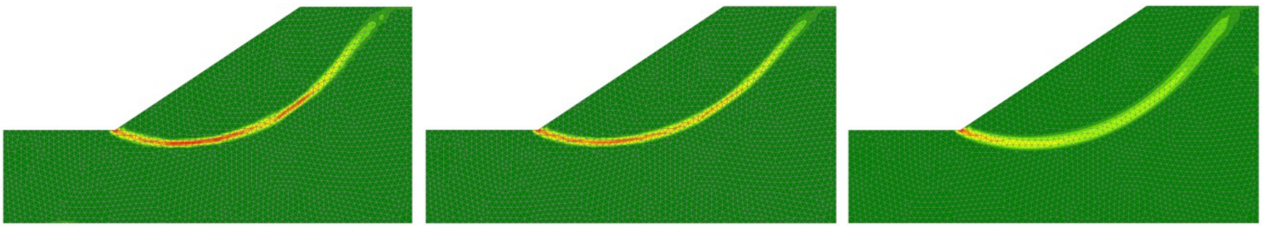
9.2. *Influence of the mesh*

The previous model is reanalysed on a densified mesh basis: a density of 0,5 m is applied (instead of 2 m).

The 3 alternatives of iterative settings are tested. The "custom" one is adapted from the "standard" values, with a precision of 1% instead of 0,1%.

In this case, the calculation times are highly different: 2'37'' for the "standard" settings versus 1'20'' for the 2 other settings sets.

The safety coefficient of 1,1 is reached by the 3 variants and the failure mechanism is correctly identified (see figure below).



"Accelerated" settings *"Standard" settings* *"Custom" settings*

Figure 20: Total strains in the slope after c-phi reduction

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