Cesar 2D

Tutorial 2g.14

Stability analysis of a slope with water table

Ref: CESAR-TUT(2g.14)-v2021.0.1-EN

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.

The pore pressure distributions are calculated by means of 2 different methods. The first one assumes hydrostatic conditions; the pore pressure at any point below the water table is calculated by measuring the vertical distance to the water table and multiplying by the unit weight of water. In the second method, the pore pressure field is calculated by steady-state analysis.

For both methods, there is zero pore pressure above the water table.

In this tutorial, the user will model the slope stability analysis (c-phi reduction process). Under the hydrogeological state, we will calculate the factor of safety of the slope, defined as follows:

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

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

2. PROBLEM SPECIFICATIONS

General assumptions

- Plain-strain problem,
- Static analysis,
- Steady-state water table,
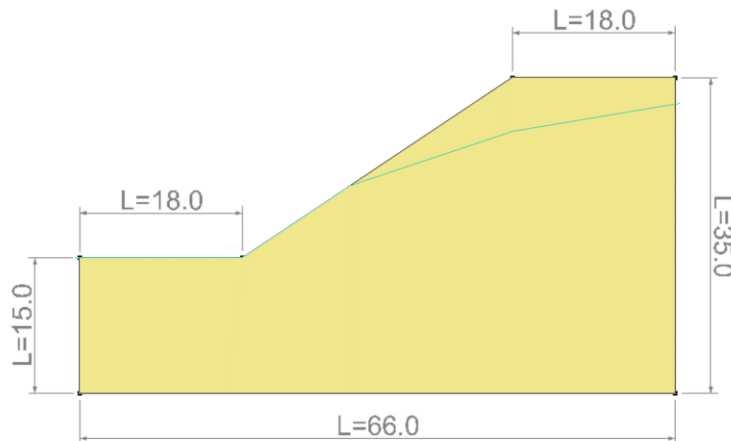- Non-linear behaviour of the soil.

Geometry

Figure 1: Situation of the problem

Material properties

The constitutive model is a standard isotropic elasticity associated to the Mohr-Coulomb criterion.

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

3. INPUTS FOR C/PHI REDUCTION ANALYSIS

3.1. 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.

3.2. 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.
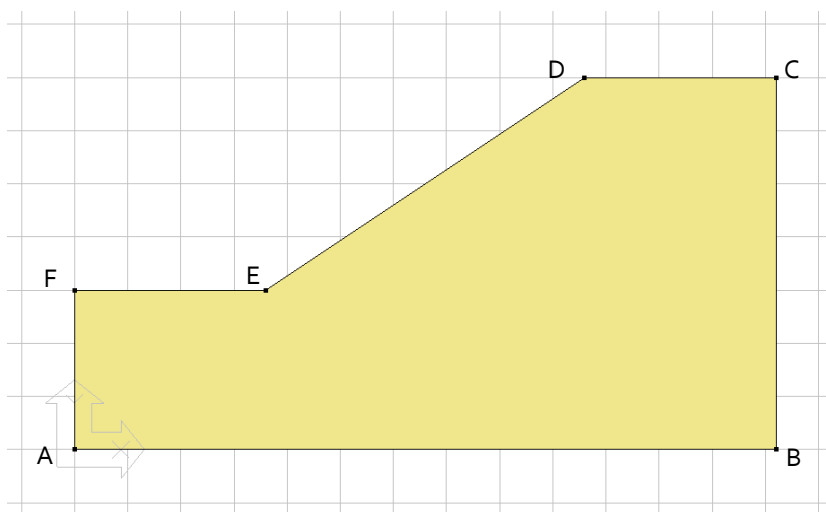9. Enter **(0 ; 0)**, and press **Apply**. Segment F is created.

Figure 2: Geometry construction

💡 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.

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. This has to be set in Preferences > Geometry Display. Tick "Body label".

3.3. *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*.

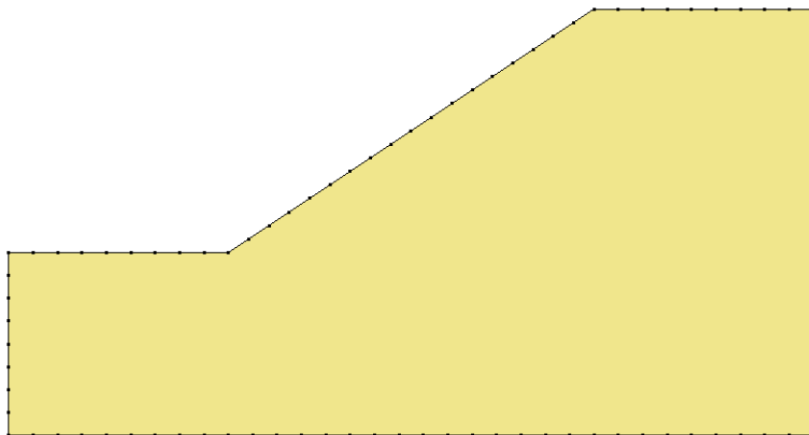⚠ The software algorithm will adjust the length for the best fit with the input value of length.

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.
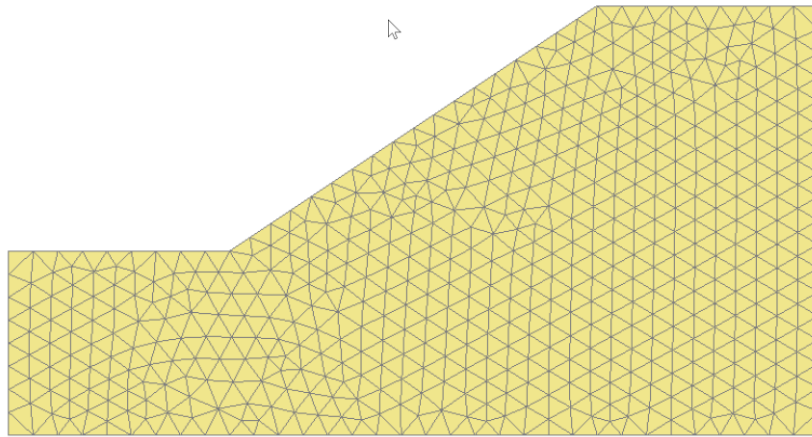
3.   Click on **Apply** to generate the mesh.

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.

3.4. Calculation properties for 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 **Steady state** as "Model name".
4. Select **NSAT** as "Solver".
5. Tick **Plane problem** as model configuration.
6. Click on **Validate**.
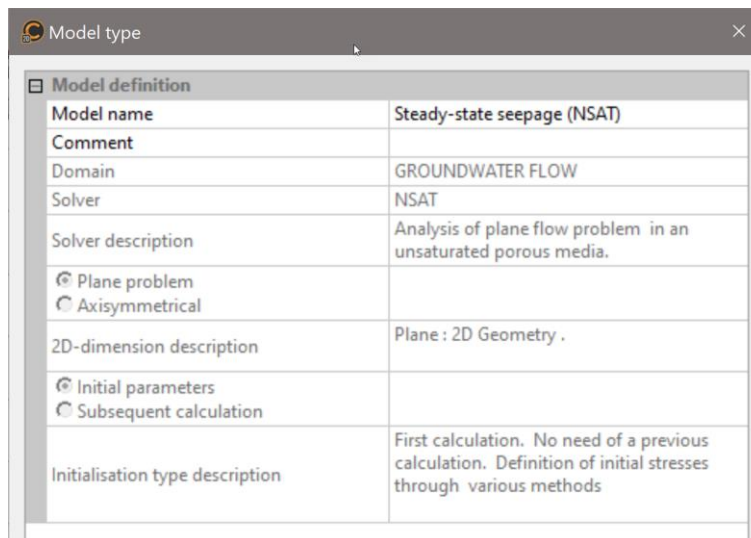
The date tree is now as illustrated below.
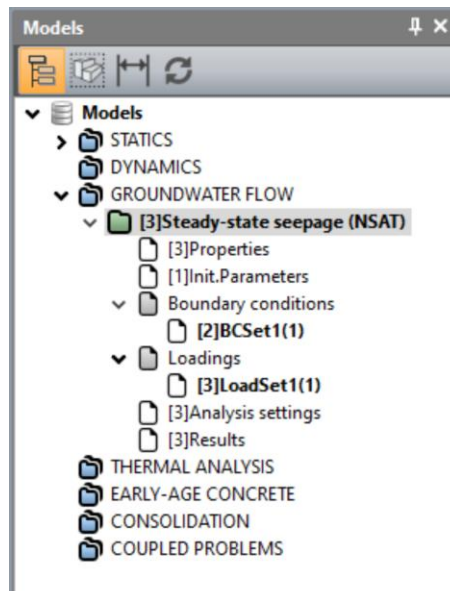
Material properties for the surface bodies:

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

1. Click on [icon] **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 as we are analysing under steady-state condition.

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
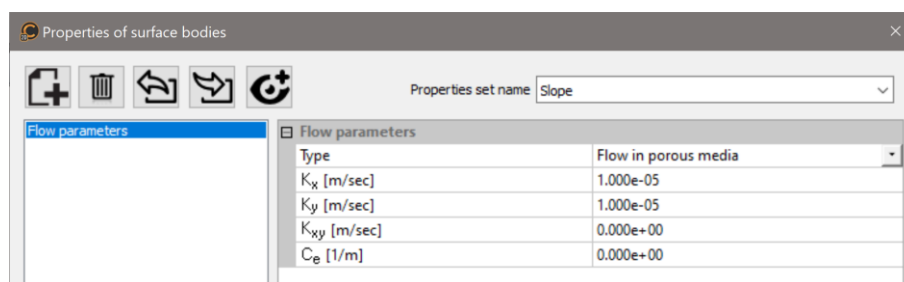
Figure 5: Toolbox for material library definition

Assignment of data sets:

As data sets are created, we affect them to the bodies of the model.

1. Click on [icon] **Assign properties** tool.

2. On the left side, a new window is displayed. Click on [icon] **Properties for surface bodies**.

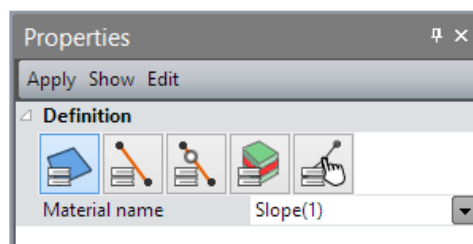3. Select the body of the slope on the model window and a set of parameters in the list.

4. **Apply**.

Figure 6: Toolbox for properties assignment

Boundary conditions:

1. Go to the **BOUNDARY CONDITIONS** tab.

2. Activate $\overline{h}$ 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. Facultative. It is possible to modify the default name assigned to the boundary conditions set. Press [F2] ; enter **Water heads** for example.
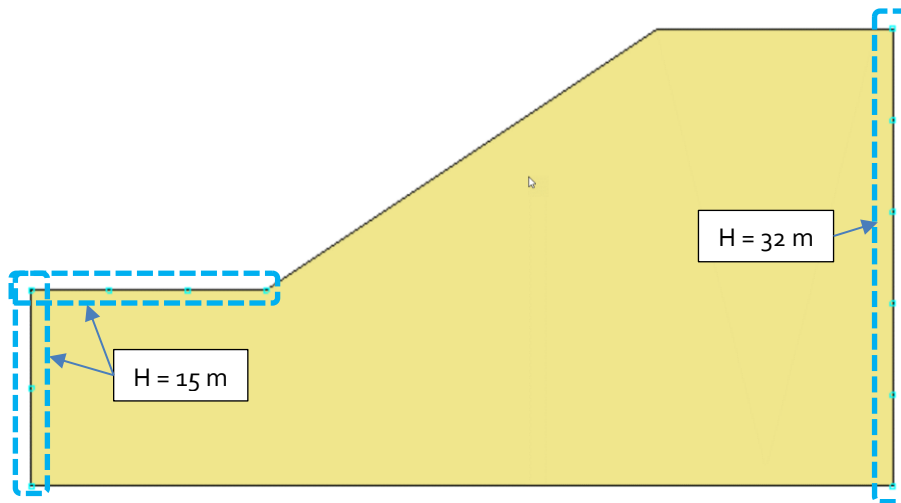
Figure 7: Applied water heads

Analysis settings:

1. In **ANALYSIS** tab, activate **Analysis settings** ☑.
2. In the **Time steps** section, select the "Steady state".
3. In the **Subsequent calculation** section, enter the name of a file with ".rst" extension (we use here iniwtb.rst). This file will be used to initialise the hydrostatic field in the stability analysis model.
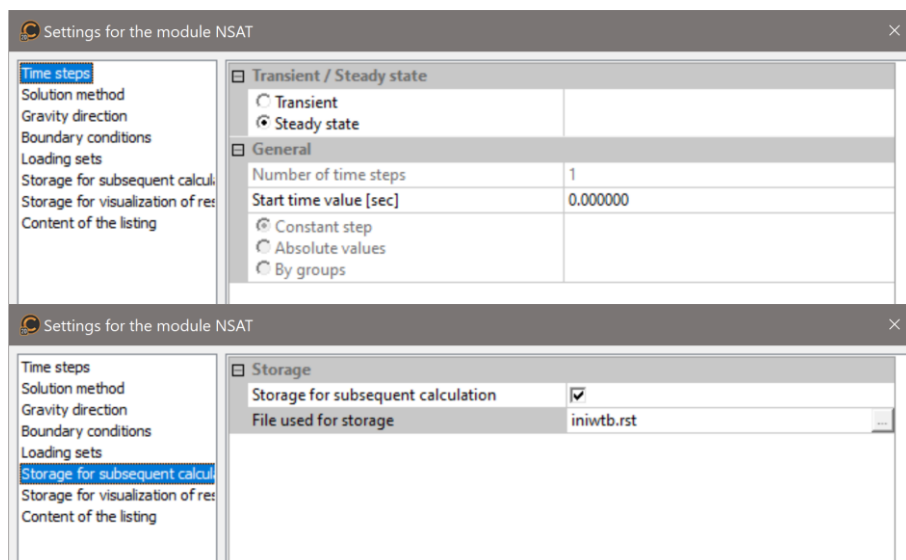4. Other settings are set with their default values.
5. Close, *OK*.

*Figure 8: Toolbox for input of calculation parameters*

3.5. Calculation properties for c/phi reduction 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 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**.
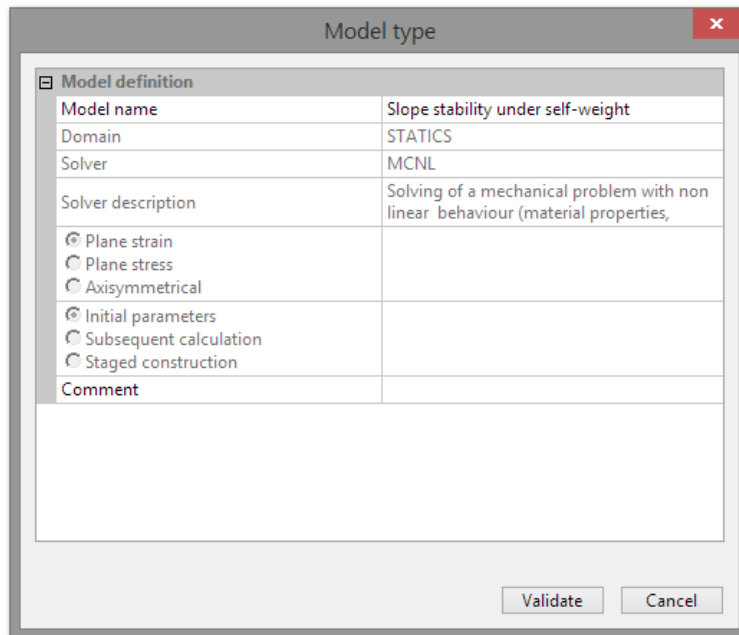
The date tree is now as illustrated below.
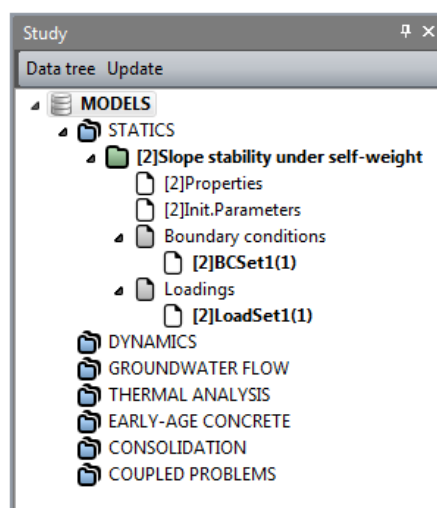
Material properties for the solid elements:

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

1. Click on ![icon] *Properties for 2D elements*.
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 *Validate* and *Close*.

	ρ (kg/m³)	E (kN/m²)	ν	c (kN/m²)	φ (°)	ψ (°)
Slope	1882	10 000	0,3	42	15	0
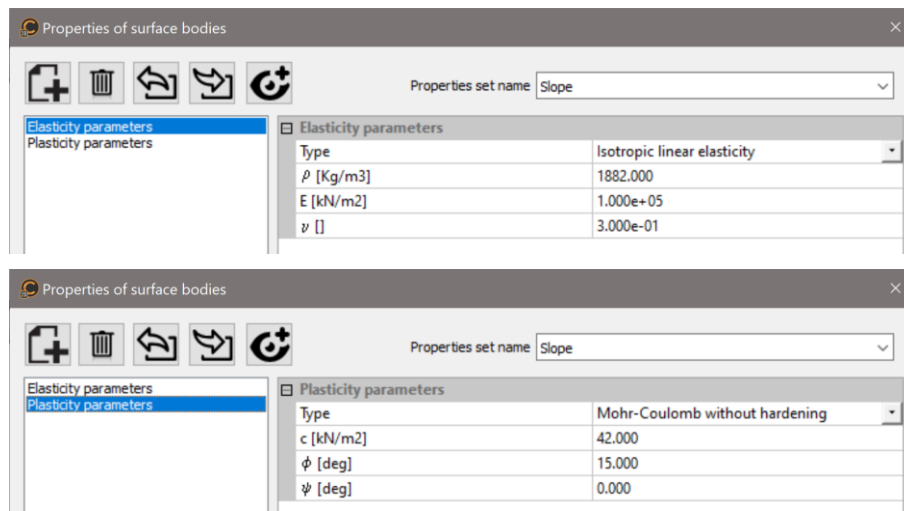
Figure 9: Toolbox for material library definition

Assignment of data sets:

As data sets are created, we affect them to the bodies of the model.

1. Click on ![icon] *Assign properties* tool.

2. On the left side, a new window is displayed. Click on ![icon] *Properties of surface bodies*.
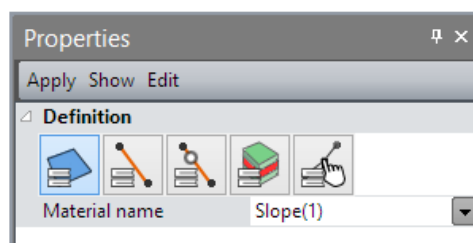3. Select the body on the model window and a set of parameter in the list.
4. *Apply*.

Figure 10: Toolbox for properties assignment

Boundary conditions:

1. Go to the **BOUNDARY CONDITIONS** tab.

2. On the toolbar, activate [icon] 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.
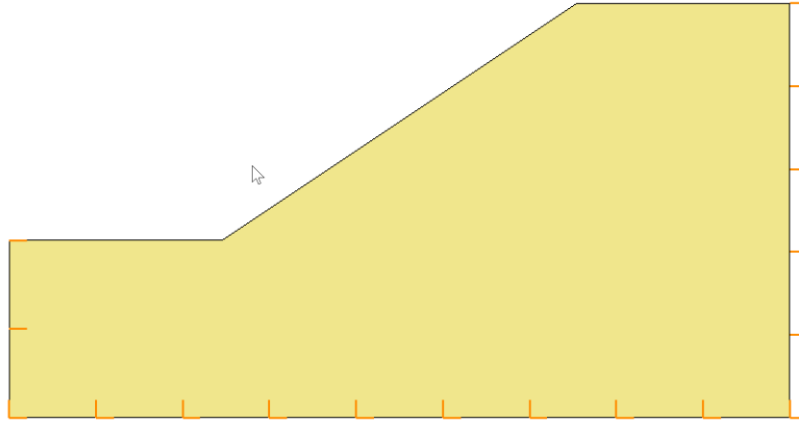

Figure 11: display of the boundary conditions

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 – Isotropic.**
   - Select "File" as **Water head input**.
   - Define *iniwtb.rst* as initial water head file.
   - *Validate*.

3. In present version, in addition, user must define the dry and saturated weight of the bodies. Activate **WTB – Unit weight**.
   - Set 18.82 kN/m3 as unit weight,
   - Set 18.82 kN/m3 as saturated unit weight.
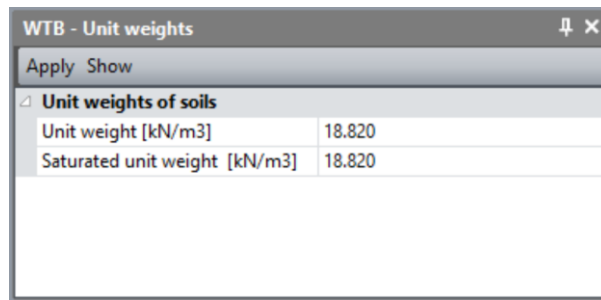   - Select the body of the slope.
   - *Apply*.
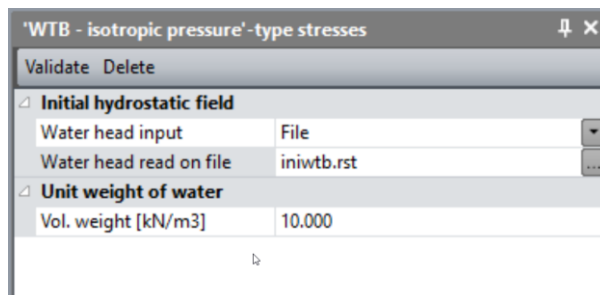
Figure 12: wtb definition using the steady-state water head field

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 – Isotropic.**
   - Select "Variable" as **Water head input**.
   - The WTB is defined by 5 points: (0; 15) , (18; 15), (30; 23), (48; 29), (66; 32).
   - ***Validate***.

3. In present version, in addition, user must define the dry and saturated weight of the bodies. Activate **WTB – Unit weight**.
   - Set 18.82 kN/m3 as unit weight,
   - Set 18.82 kN/m3 as saturated unit weight.
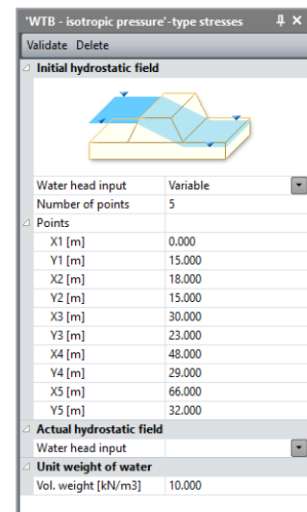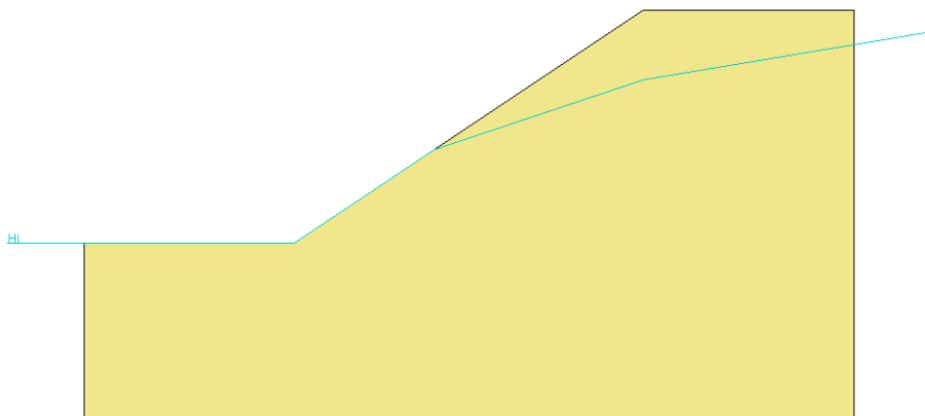   - Select the body of the slope.
   - ***Apply***.

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
- Analysis type: *c/phi reduction*
  Non-convergence detection   ☑
  Minimum Value   0,01
  Maximum Value   5
  Precision   0,01
- Storage
  Storage of total strains: ☑
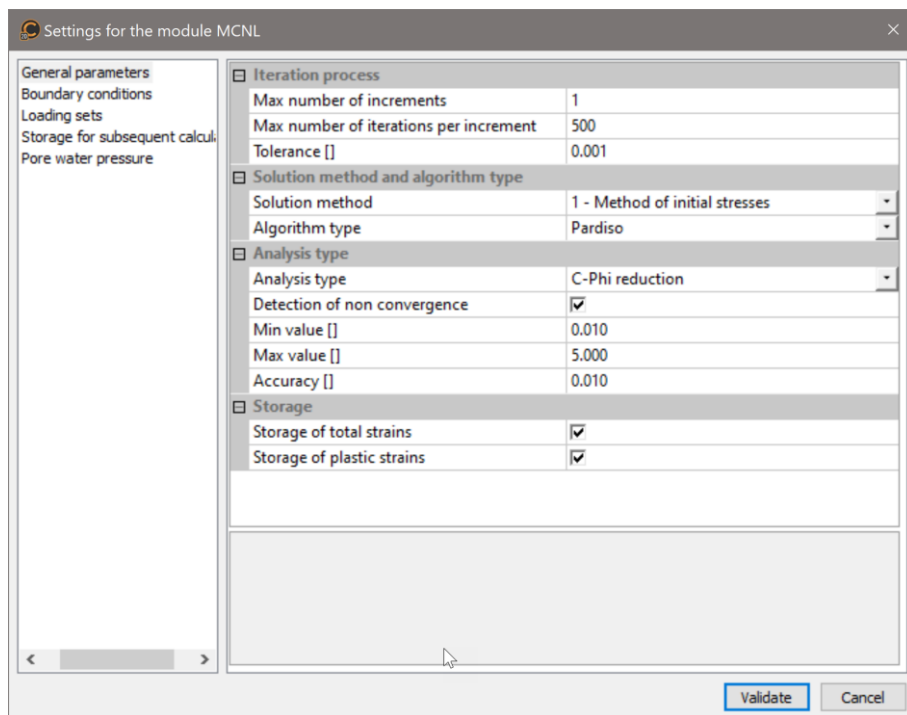 Storage of total strains: ☑

3. Close, *OK*.

Figure 14: Toolbox for input of calculation parameters

4. 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".
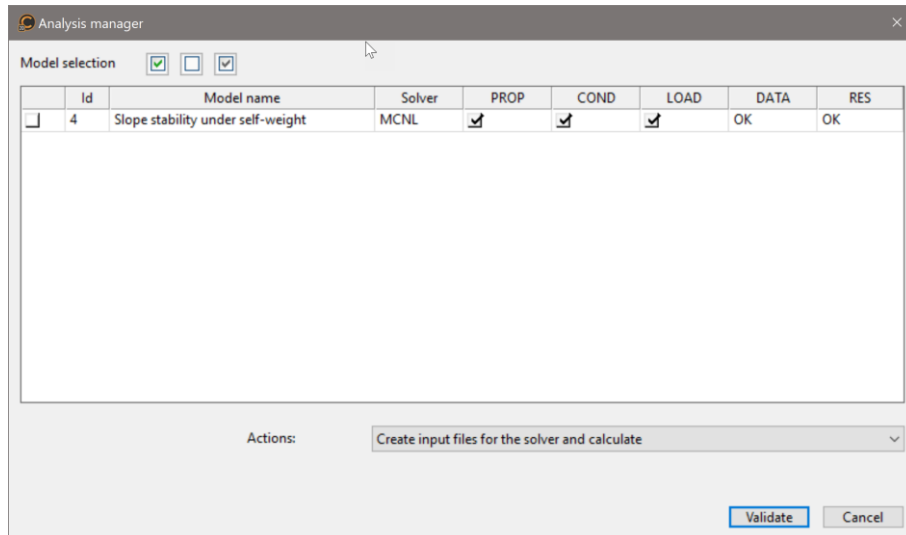
⚠ CESAR-LCPC detects if the models are ready for calculation. All steps should be validated with a tick mark.

💡 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).

5. RESULTS

5.1. 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 👁 **Type of results to display**.
   - In **Contour plots**
     - ○ Activate
     - ○ Select **H**
   - Click on **Apply**.

3. 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**.

4. Click on **Legend**.
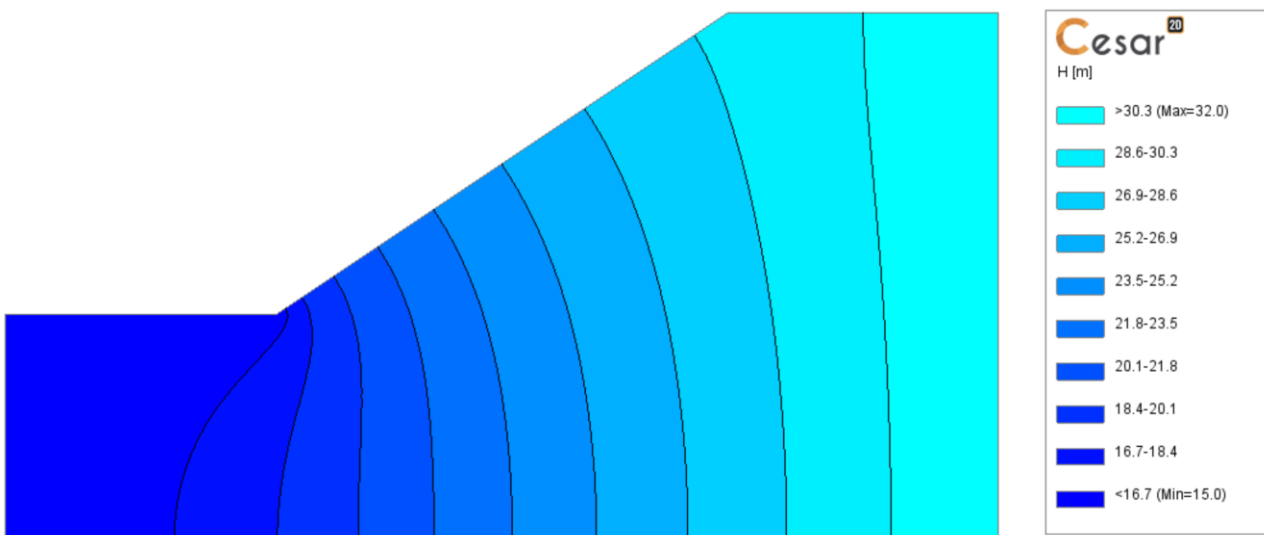   - In **Legend**, select **Contour plot**
   - Click on **Apply**.

H [m]

- >30.3 (Max=32.0)
- 28.6-30.3
- 26.9-28.6
- 25.2-26.9
- 23.5-25.2
- 21.8-23.5
- 20.1-21.8
- 18.4-20.1
- 16.7-18.4
- <16.7 (Min=15.0)

Figure 15: Display of the water head H

Display of the water head field.

1. Go to the **RESULTS** tab.

2. Click on 👁 ▦ *Type of results to display*.
   - In **Contour plots**
     - ○ Activate
     - ○ Select **P**
   - Click on *Apply*.

3. Click on ☑▦ *Contour plot settings*.
   - In **Contour plot settings**
     - ○ Inactivate
     - ○ Style: Areas
   - In **Contour edges**
     - ○ Activate
     - ○ Color set: Black
   - In **Contour plot**
     - ○ Number of contours: 10
     - ○ Color scheme: Groundwater
   - Click on *Apply*.

4. Click on ▦ *Legend*.
   - In **Legend,** select **None**
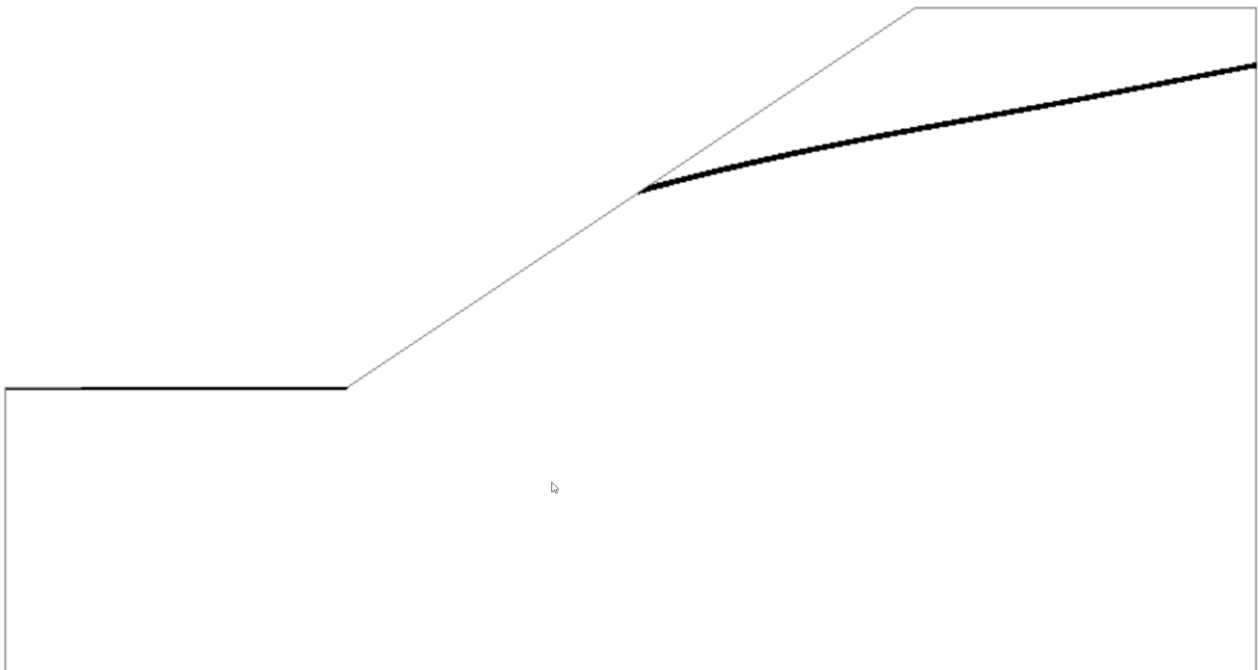   - Click on *Apply*.

Figure 16: Display of the free surface (p = 0)

5.2. Stability analysis

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

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

The calculated safety factor is 1,11.

Display of the rupture mechanism in the slope.

1. Go to the **RESULTS** tab.

2. Click on 👁 *Type of results to display*.
    - In **Deformation**, select **Deformed** as Mesh,
    - In **Contour plots**
        o Activate
        o Select **ε_1,p**
    - Click on *Apply*.

3. Click on ☑ *Contour plot settings*.
    - In **Contour plot settings**
        o Activate
        o Style: Areas
    - Click on *Apply*.

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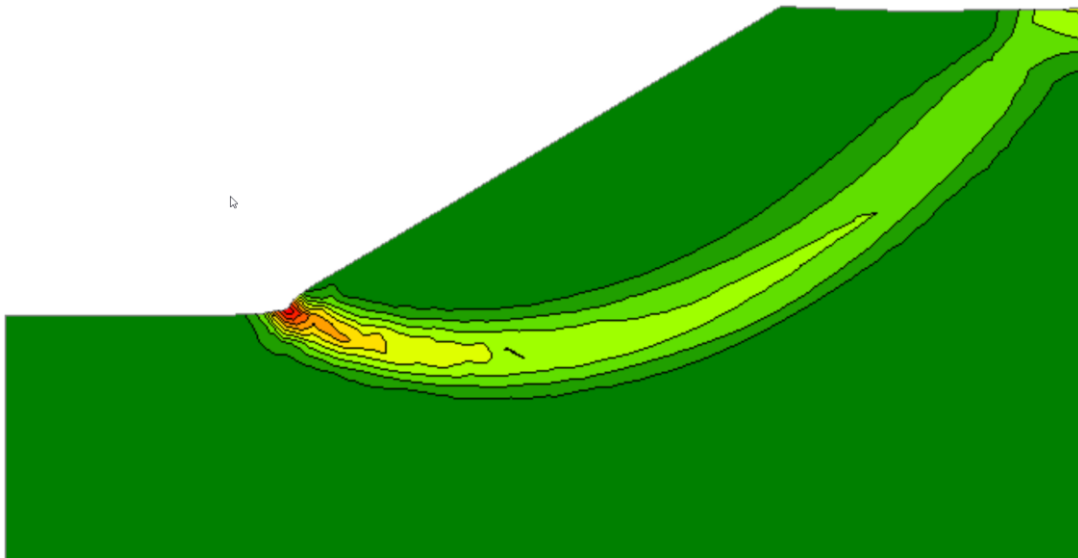

Figure 17: Plasticity and deformed mesh

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