



**Tutorial 3g.06**

**Dewatering in a slope**

| Ref: CESAR-TUT(3g.06)-v2021.0.1-EN



# 1. INTRODUCTION

## 1.1. Tutorial objectives

Excess water infiltration came from rainfall may be a major cause of slope failure due to the rising of pore water pressure and the decrease in shear strength of the slope. It is then crucial to protect the slope in a way to discharge the ground water and to maintain the water pressure.

This tutorial aims to study the influence of subsurface drainage system in ground water flow. Two models will be compared: one without any drainage system and another with subsurface drainage system. After the hydraulic analysis, the study can be further conducted to hydro-mechanical analysis in order to observe the evolution of mechanical stress due to change in hydrostatic pressure.

This tutorial allows the user to learn:

- ✓ How to create a 3D volume from 2D geometry,
- ✓ How to obtain ground table water position and flow network in 3D model,
- ✓ How to conduct an uncoupled hydro-mechanical analysis with CESAR-LCPC.

## 1.2. Problems description

From the piezometer test, it is noted that the natural slope has an upstream pressure head of 20 m and the one in downstream of 16.6 m.

In a purpose to discharge the flow in the slope, a subsurface drainage system consists of field drains and horizontal collector drains is installed. The field drains are of 2 m width with 4 m spacing from the center axis to the slope edge. The horizontal collector drains are of 60x60 cm<sup>2</sup> square section. After the installation, the pressure head in downstream becomes 16 m.

Geometry:

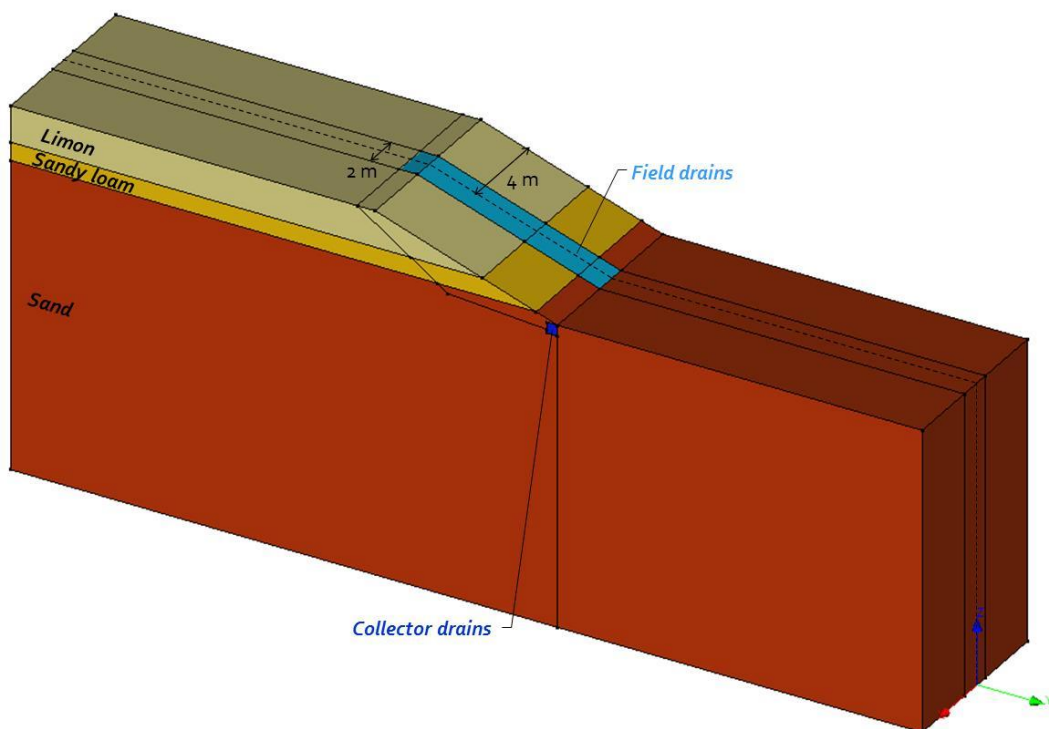


Figure 1: Model dimension in 3D view

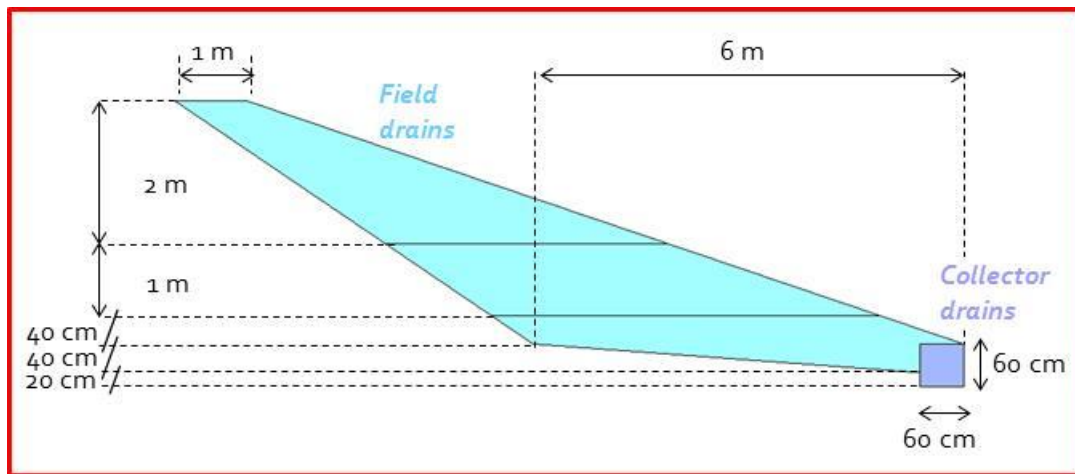
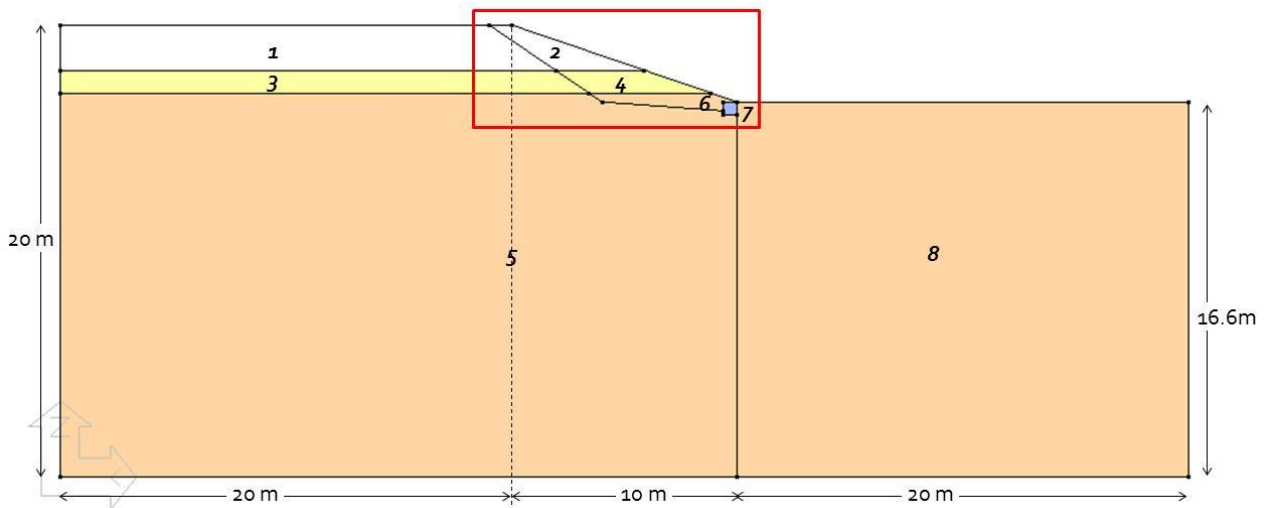


Figure 2: Model dimension in OZY plan

Analysis stages:

Hydrogeological analysis

1. Initialisation of flow network – Natural slope
2. Initialisation of flow network – Slope with subsurface drainage



Mechanical analysis

3. Hydro-mechanical analysis – Natural slope
4. Hydro-mechanical analysis – Slope with subsurface drainage




## 2. GEOMETRY AND MESH

### 2.1. Geometry

#### Working plane settings:


1. Go to the tab **GEOMETRY** tab
2. Click on  **Tools:**
  - Set OZY  as coordinates system plane.
  - Set the visible grid to 0.2 m ( $dX = dY = dZ = 0.2m$ )

#### Drawing of the geometry:

1. Click on . The point definition dialog is displayed.
  - Define the coordinates of the construction points (see Figure 1).
2. Use  **Lines** to link the points previously defined.
3. Click on  **Edge partitions** to divide the contour lines in the slope/field drains segments (between the segments 1-2-3-4-6).

#### Generation of the surfaces

After having the contour, we can now define the associated surface.


1. Activate  **Plane Surface**.
2. Select all previously defined edges.
3. **Apply**.

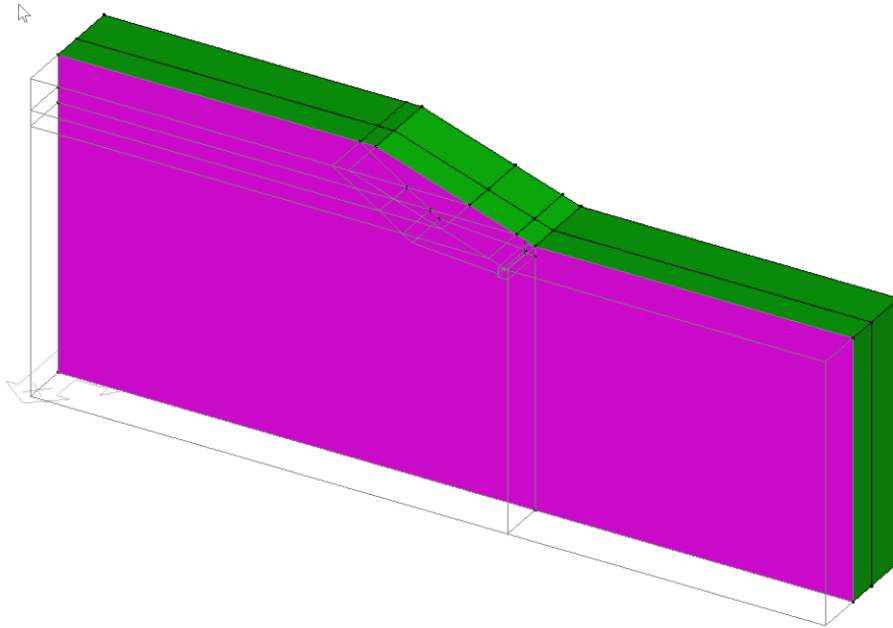
#### Generation of the volumes

The volume of the slope is constructed using an extrusion of the plane surfaces.

1. On the **Selections toolbar**, activate only **Select faces** option.




2. Select all the surfaces to be extruded
3. Click on  **Extrusion** and set:
  - *Translation* as operation type,
  - *Generation of volume bodies* as extrusion type,
  - 1 as number of operations,
  - $(-3 ; 0 ; 0)$  as vector,
  - Tick the box "*Remove surface bodies*".
4. **Apply**
5. Repeat the procedure to complete the global volume
  - 2<sup>nd</sup> extrusion operation with  $(-2 ; 0 ; 0)$  as vector
  - 3<sup>rd</sup> extrusion operation with  $(-3 ; 0 ; 0)$  as vector



View of the model before last extrusion ("show" option)

#### Dissociation of bodies:



During extrusion operation, the initial faces selection is a group of 8 surfaces. However, the result of extrusion is one single body made of the association of 8 volume bodies. In order to distinguish the bodies with different material properties, it is necessary to split them.

1. Activate  **Explode bodies**. Select a volume body. **Apply**.
2. Repeat the operation for the other extruded bodies.

#### Merge bodies and bodies edition:










This step is optional but it facilitates the assignment of material properties and loading case for a group of elements during multiple analysis stages.


1. On the **Selection Toolbar**, activate only **Select bodies** option.
2. Merge bodies operation:
  - Activate  **Merge bodies**,
  - Select the volume bodies associated to "Field drains" (see Figure 1),
  - **Apply**,
  - Repeat the operation in order to group the other bodies associated to "Collector drains", "Limon", "Sandy loam", and "Sand" (see Figure 1).
3. Bodies edition:
  - Activate  Body properties.
  - Right-click on the body corresponding to the "Field drains". Specify a name and a colour for this body. Click on **Apply**.
  - Repeat previous operation for the other bodies.

## 2.2. Mesh

### Mesh density definition :

1. Go to the **MESH** tab on the project flow bar.
2. Select  **Front view** option to display the model in OZY plan.
3. Select all edges of the "Field drains" bodies. Click on  **Fixed length density** to divide the segment with a fixed length. Enter **0.5 m** in the dialog box. Click on **Apply**.
4. Select all edges of the "Collector drains" bodies. Click on  **Fixed length density** to divide the segment with a fixed length. Enter **0.3 m** in the dialog box. Click on **Apply**.
5. Click  **Variable density** to propagate from the previous area to the model limits:
  - Tick **First / Last interval** to define the method,
  - Enter **0.5 m** as **First division** and **5 m** as **Last division**,
  - Click on the lines which link the drainage system and the soil edges in order to affect this density.
6. Select all the soil edges. Click on  **Fixed length density** to divide the segment with a fixed length. Enter **5 m** in the dialog box. Click on **Apply**.
7. Select  **Isoparametric view** option to display the model in 3D plan.
8. Select the transversal line corresponding to the top of the slope:
  - Click on  **Fixed length density**,
  - Enter **1 m** in the dialog box,
  - Tick "Propagate density to opposite sides",
  - Click on **Apply**.

### Volume meshing generation:

1. Select all the volume bodies.
2. Open the tool  **Volume meshing**
  - Select "Quadratic interpolation" as **Interpolation type**.
  - Set "Regular mesh" as **Mesh type**.
  - **Apply**.

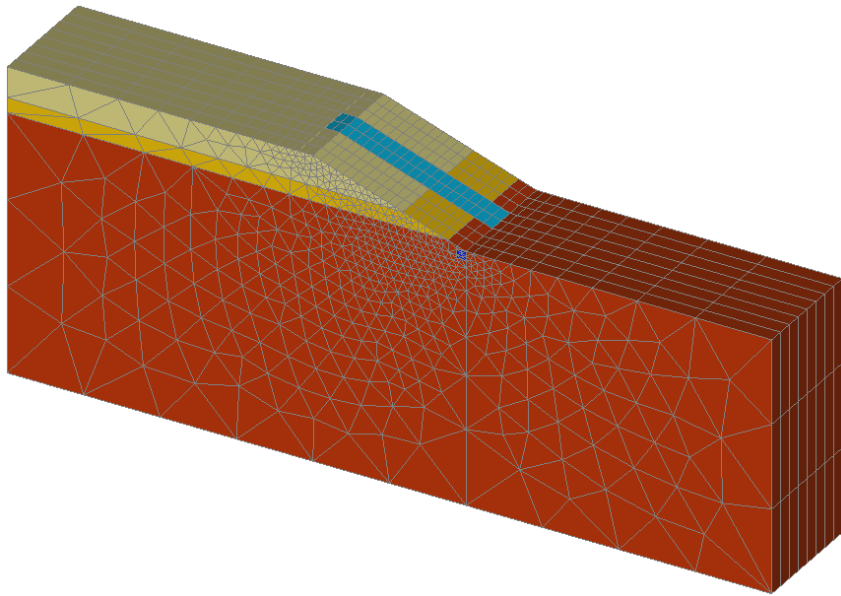


Figure 3: Example of mesh



### 3. HYDROGEOLOGICAL ANALYSIS

For analyzing the ground water flow in porous media with a potential unsaturated area, CESAR-LCPC provides NSAT module.

#### 3.1. Natural slope



##### Model definition:

1. Right click on **GROUNDWATER FLOW** in the **Study** tree view on the right side of the working window.
2. Click on **Add a model**.
3. A new toolbox is open for definition of the Model:
  - Enter **Hydro-Natural** as "Model name",
  - Select **NSAT** as "Solver",
  - Tick **Initial parameters** as calculation type,
  - Click on **Validate**.

##### Material properties for the volume bodies:



It will be convenient to define all the material properties in the material library of the study before assigning to a body.

1. Go to the **PROPERTIES** tab.
2. Click on  **Properties for volume bodies**.
3. Give a name for the properties set name (*Limon* for example).
  - In **Flow parameters**, choose "Van Genuchten & Gardner" as unsaturated law,
  - Set the parameter values as defined in the table below.
4. Click on  to create another properties set.
5. Modify the name for the properties set (*Sandy loam* for example).
  - In **Flow parameters**, choose "Van Genuchten & Gardner" as unsaturated law,
  - Set the parameter values as defined in the table below.
6. Repeat the operation until all material properties are set.
7. Click on **Validate** and **Close**.

		<i>Limon</i>	<i>Sandy loam</i>	<i>Sand</i>	<i>Field drains</i>	<i>Collector drains</i>
Saturated permeability coefficient	$k_{xs}, k_{ys}, k_{zs}$	$1.10^{-4}$	$5.10^{-5}$	$3.10^{-4}$	$1.10^{-2}$	$1.10^{-1}$
	$k_{xy}, k_{yz}, k_{zx}$	0.	0.	0.	0.	0.
Unsaturated water content function <sup>2</sup>	$\theta_s$	0.40	0.36	0.35	0.5	0.5
	$\theta_r$	0.10	0.28	0.12	0.01	0.01
	$\alpha$	0.20	0.34	0.22	0.05	0.05
	$n$	1.8	1.60	1.50	2.0	2.0
	$m$	0.44	0.38	0.33	0.5	0.5
Unsaturated permeability function <sup>2</sup>	$a$	0.02	0.03	0.04	0.01	0.01
	$nn$	1.5	1.9	2.2	3.0	3.0

⚠ The constant parameters for the van Genuchten water content function and the Gardner permeability function are chosen based on literature study. User can also define these parameters manually by curve-fitting parameters from the results of laboratory experimental test. For more information, please refer to CESAR Manual Theory.

⚠ Reference pressure is the atmospheric pressure equals to 100 kPa.

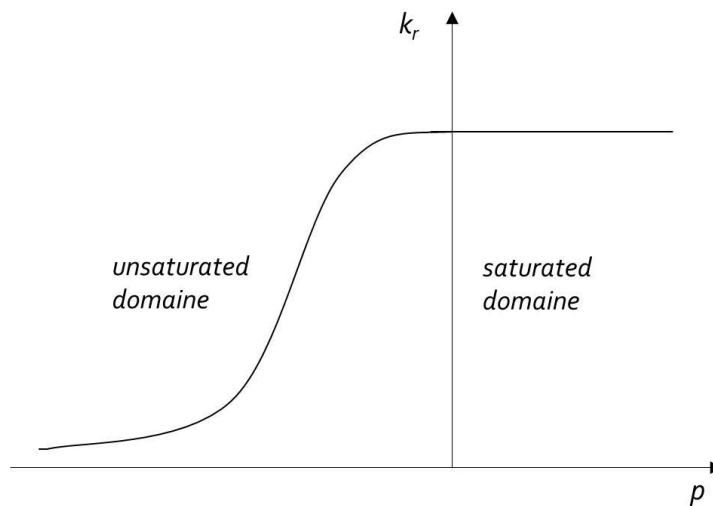




Figure 4: Permeability function in NSAT module

#### Assignment of properties sets:


In this stage of analysis, the subsurface drainage system hasn't yet installed. Hence, it is considered that the bodies corresponding to the subsurface drainage system have the natural soil properties.

1. Activate  **Assign properties** tool.
2. Click on  **Properties for volume bodies**.
3. Select all volume bodies corresponding to the surface bodies n° 1-2 (refer to Figure 2)
  - Select "Limon" properties set in the list.
  - **Apply**.
4. Select all volume bodies corresponding to the surface bodies n° 3-4 (refer to Figure 2)
  - Select "Loam" properties set in the list.
  - **Apply**.
5. Select all volume bodies corresponding to the surface bodies n° 5-8 (refer to Figure 2)
  - Select "Sand" properties set in the list.
  - **Apply**.

#### Boundary conditions:

By default, the boundary condition will be named BCSet1. User can edit the name of boundary condition by pressing the [F2] key.

1. Go to the **BOUNDARY CONDITIONS** tab.
2. On the **Selections toolbar**, activate only **Select faces** option.

3. Click on  $\bar{h}$  **Prescribed constant load**.
  - Select the surface bodies in upstream side. Enter **20 m** as load value. **Apply**.
  - Select the surface bodies in downstream side including the top surface of the collector drain. Enter **16.6 m** as load value. **Apply**.
4. Click on  **Seepage condition**. Select the surface bodies of the top of the slope and along the slope. **Apply**.

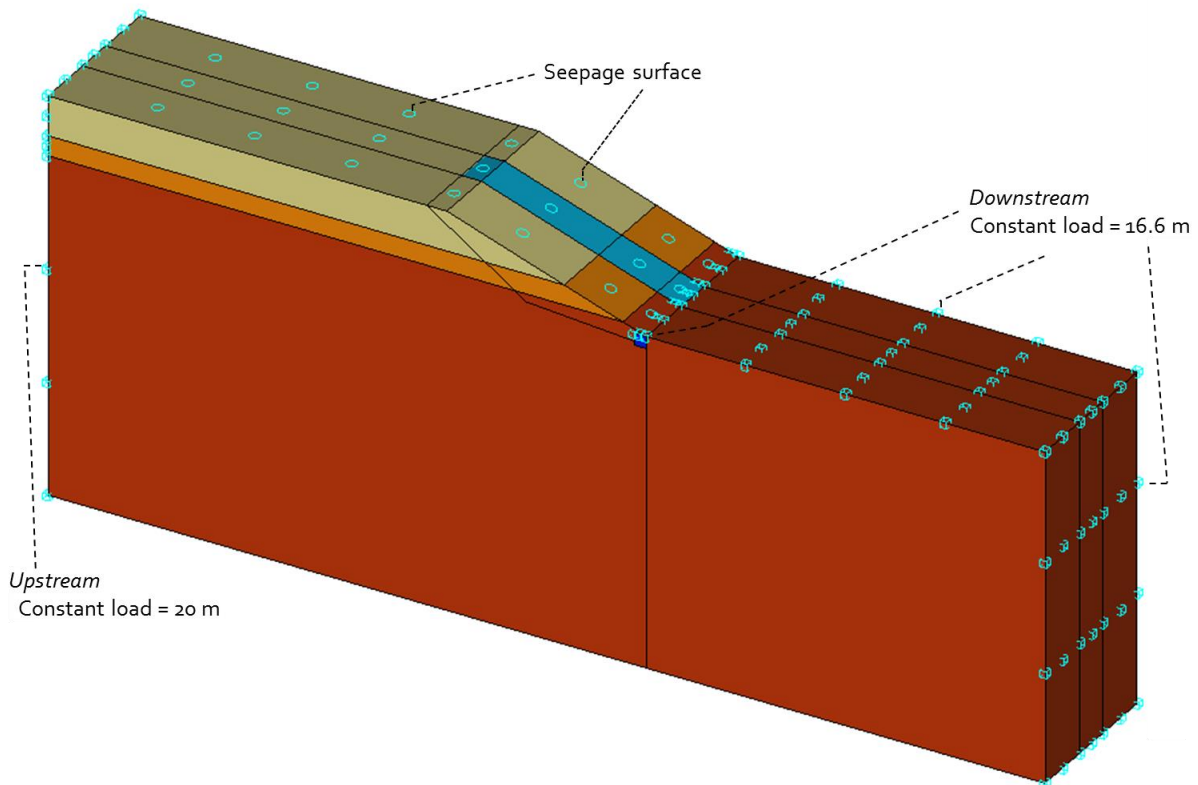



Figure 5: Boundary condition in the natural slope

Analysis settings:

1. Go to the tab **ANALYSIS** then activate the **Analysis settings** .
2. In the **Time steps** section, choose "Steady state".
3. In the **Solution method** section:
  - Select "Fixed point method"
  - Max number of iterations: 100
  - Tolerance: 0,001
4. In the **Storage for subsequent calculation** section:
  - Activate "Storage for subsequent calculation"
  - Specify the folder path and the name of the file where the calculated hydraulic pressure will be stored (extension \*.rst).
5. **Validate**.




### 3.2. Slope with subsurface drainage system

#### Model definition:

1. In the data tree, right click on the previous model title and then select **Copy of the model**.
2. A toolbox is displayed for definition of this new model.
  - Change the model name: **Hydro-Drainage**.
  - Select "Initial parameters" as the type of calculation.
  - **Validate** to close the toolbox for model definition.
3. A toolbox is displayed for setting the sharing options. By default, all the setting of the previous model is copied to the new model.
  - Untick "Properties" and "Boundary conditions",
  - Let the default check mark in "Loadings".
  - Click on **Validate**.


#### Assignment of properties sets:

Here the subsurface drainage system is installed. We have to modify the material properties to the properties of subsurface drainage system.


1. Select  **Isoparametric view** option to display the model in 3D plan.
2. Activate  **Assign properties** tool.
3. Click on  **Properties for volume bodies**.
4. Select the volume bodies corresponding to the "Field drains" (refer to Figure 1)
  - Select "Field drains" properties set in the list.
  - **Apply**.
5. Repeat the operation to assign the properties of "Collector drains".

#### Boundary conditions:

By default, the boundary condition of the previous model is copied to the actual model. In this case, we only have to modify the pressure head in the downstream.

1. Go to the **BOUNDARY CONDITIONS** tab.
2. On the **Selections toolbar**, activate only **Select faces** option.
3. Click on  **Prescribed constant load**.
  - Select the surface bodies in downstream side including the top surface of the collector drain. Enter **1.6 m** as load value. **Apply**.





#### Analysis settings:

1. Go to the tab **ANALYSIS** then activate the **Analysis settings** .
2. No changes in **Time steps** and **Solution method**.
3. In the **Storage for subsequent calculation** section:
  - Activate "Storage for subsequent calculation"
  - Specify the folder path and the name of the file where the calculated hydraulic pressure with subsurface drainage system will be stored (extension \*.rst).
4. **Validate**.

## 4. RESULTS AND ANALYSIS

### 4.1. Flow network contour

We will compare the flow network in both models in order to see the influence of subsurface drainage system on the slope.

1. Go to tab **RESULTS**.
2. Click on  **Type of results to display**.
  - Select **Undeformed** as *Mesh*,
  - Tick the box "Active" and select **Water head** in *Contour plots* section,
  - **Apply**.
3. Click on  **Mesh settings**.
  - Select **Group border** as *Mesh border*,
  - **Apply**.
4. 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*,
  - Select **Manual** as *Scale* and put **16.** as *manual inferior value* and **20.** as *manual superior value*,
  - Select **Groundwater** as *Color scheme*.
  - **Apply**.
5. Click on  **Legend**.
  - Select **Contour plot** as *Legend type*,
  - Tick the box "Legend border",
  - **Apply**.

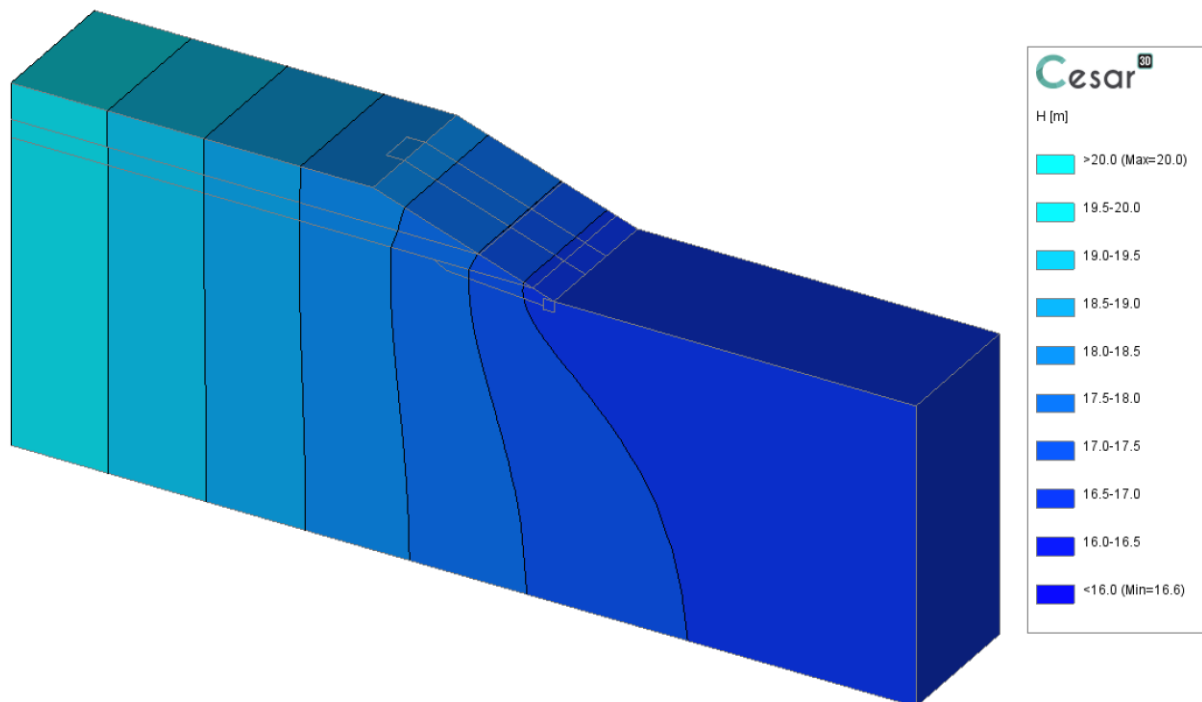


Figure 6: Water head contour in natural slope condition

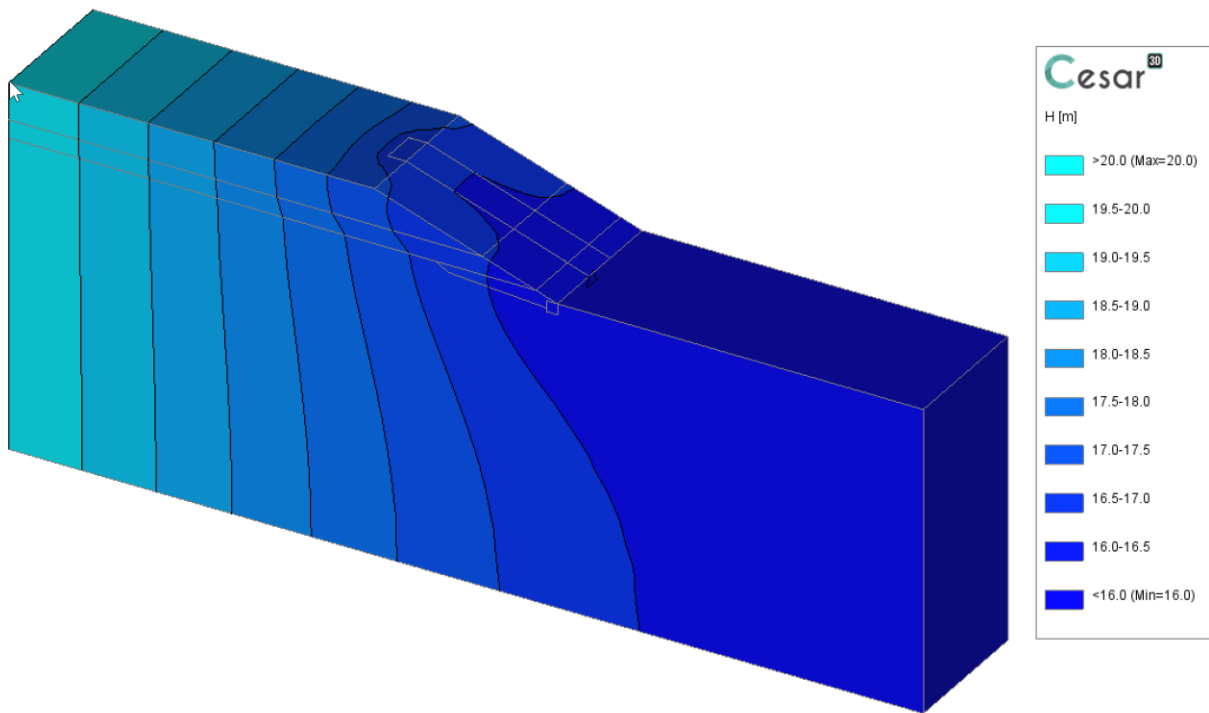



Figure 7: Water head contour in the slope with subsurface drainage system

#### 4.2. Water table on the slope

For a better comparison analysis, we will compare the values of water head in the centre axis of the slope.

1. Go to tab **CHARTS**.
2. Activate **Points** on the **Entities for charts** toolbar.
  - Create 3 points (A, B and C) to draw the line of centre axis.
3. Activate **Lines** on the **Entities for charts** toolbar.
  - Click on two edges then double click until a red line appears.
4. Activate **Line Set** on the **Entities for charts** toolbar.
  - Click on the two lines on the centre axis of the slope,
  - Enter a name (for example "Axis") in the *Line set* window,
  - Click on **Add**.
5. Activate **Charts for a line Set**  on the **Charts** toolbar.
  - Select "Water head" as *Parameter*,
  - Select the cut line of centre axis as *Line set*,
  - Select the time step
  - Click on **Apply**.

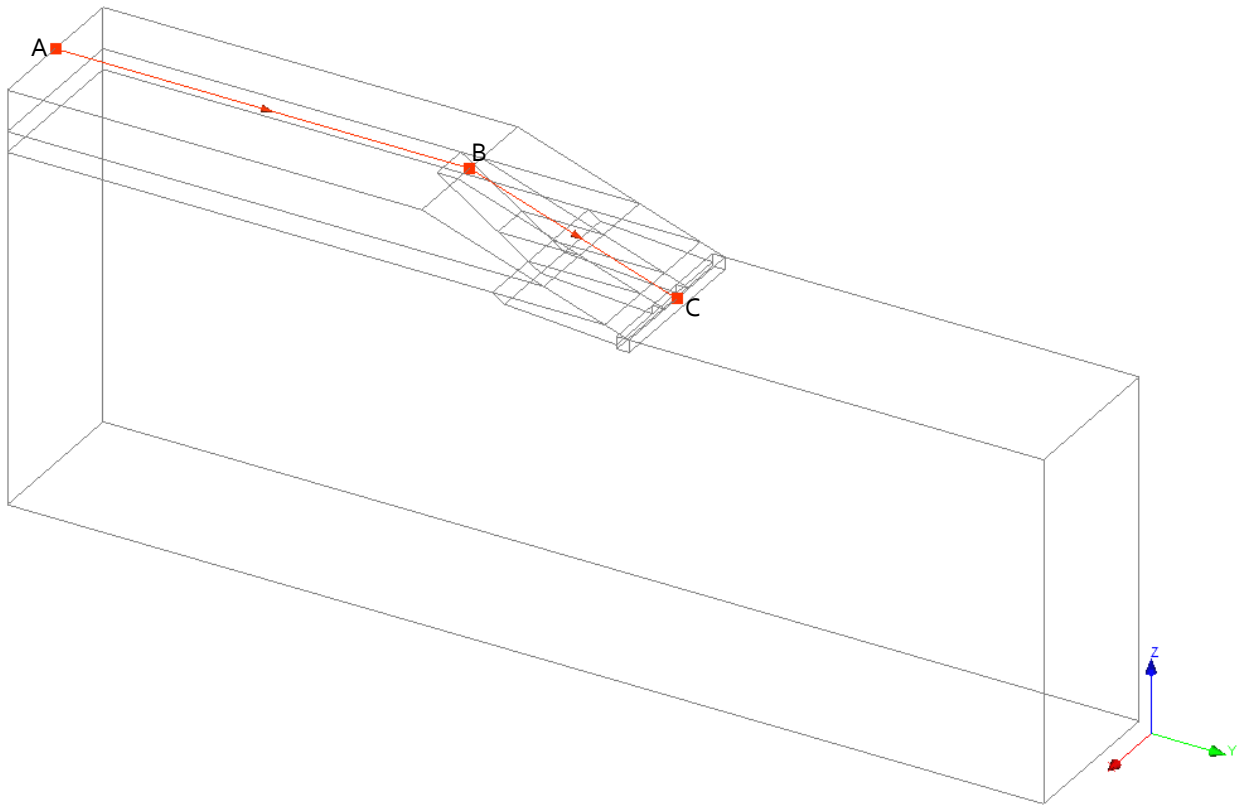


Figure 8: Cut line in the centre axis of the slope

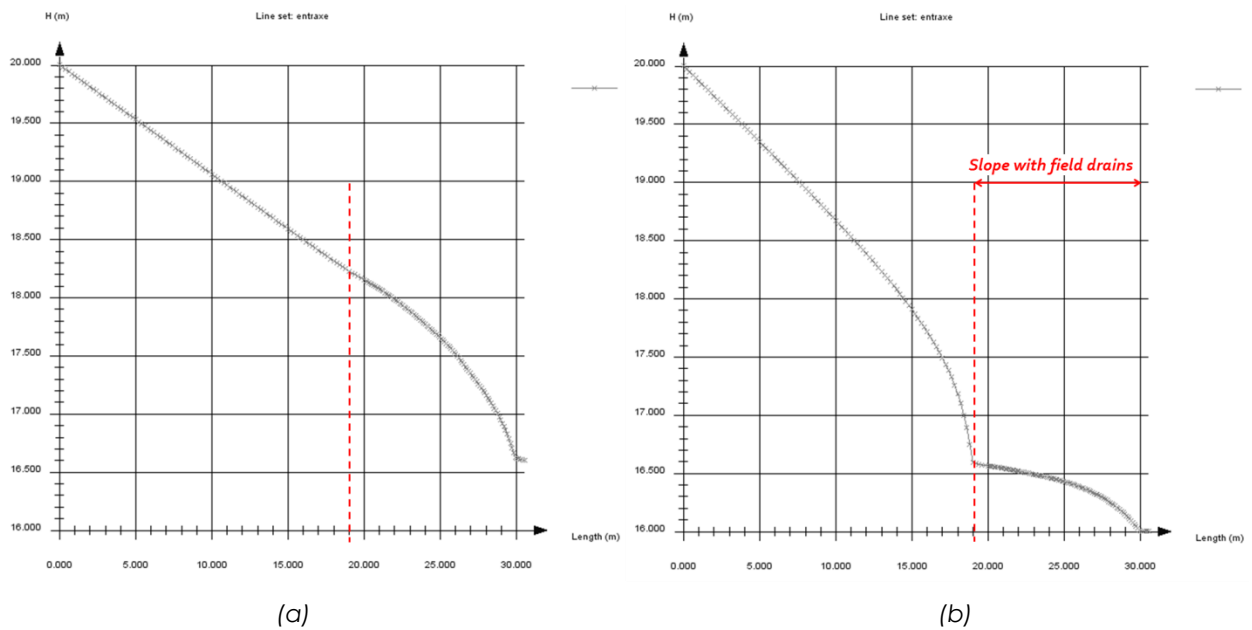


Figure 9: Comparison of water head values (a) Natural slope (b) Slope with subsurface drainage system

Figure 9 shows clearly the aquifer drawdown phenomenon due to the installation of subsurface drainage system. The water table on the top of the slope (at the upstream side) has progressively lowered about 0.3 to 1.6 m. The drainage system allows intercepting the aquifer. It can be seen also in Figures 7-8 that the aquifer drawdown occurs in the transversal plane of the slope.

## 5. FURTHER ANALYSIS: HYDRO-MECHANICAL ANALYSIS



After analysing the dewatering on the slope, a further analysis can be conducted in order to examine the mechanical stability of the slope. In this case study, an uncoupled analysis is chosen. The calculated hydraulic pressure resulting from the NSAT module will be used as a loading pressure condition for the MCNL module.

### Model definition:

1. Right click on **STATICS** in the **Study** tree view on the right side of the working window.
2. Click on **Add a model**.
  - Change the model name: **Mech-Natural**,
  - Select "MCNL" as *Solver*,
  - Select "Initial parameters" as the type of calculation,
  - **Validate** to close the toolbox for model definition.

### Material properties:

Follow the same procedure to register the material database.

1. Go to the tab **PROPERTIES**.
2. Activate  **Properties for volume bodies**.
3. Click on  to create another properties set.
4. Modify the name for the properties set, *Limon* for example.
  - In **Elasticity parameters**, choose "Isotropic linear elasticity". Set the values of  $\rho$ ,  $E$  and  $\nu$  as specified in the table below.
  - In **Plasticity parameters**, choose "Drucker-Prager without hardening" as criterion. Set the values of  $c$ ,  $\phi$  and  $\psi$  as specified in the table below.
5. Click on Validate and **Close**.

		Limon	Sandy loam	Sand	Field drains	Collector drains
Elasticity parameters	$\gamma_h$ (kN/m <sup>3</sup> )	18	17	18	20	20
	$E$ (MPa)	38	22	27	50	50
	$\nu$	0.30	0.45	0.25	0.25	0.25
Plasticity parameters	$c$ (kPa)	-	50	-	-	-
	$\phi$ (°)	32	20	34	40	-
	$\psi$ (°)	2	-	4	10	-

### Assignment of properties sets:


Assign the properties for "Limon", "Sandy loam", and "Sand".

### Initial parameters:

No initial mechanical stress applied.



### 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.

### Loading set:




Default name of the boundary condition, LoadSet1, set can be edited using the [F2] key.

There will be two types of loading: one of gravitational force and another of calculated hydraulic pressure resulting from NSAT module.

1. Go to the **LOADS** tab.
2. On the toolbar, click on  **Gravity forces**. Select all the bodies. Click on **Apply**.
3. On the toolbar, click on  **WTB-Isotropic pressure**.
  - On the **Initial hydrostatic field**, select "File" as *Water head input*.
  - Specify the storage file of the calculated hydraulic pressure from the groundwater analysis **Hydro-Natural**.
  - **Validate**.
4. On the toolbar, activate  **WTB - unit weights**.
  - Unit weight: 0.019 MN/m<sup>3</sup> (20 kN/m<sup>3</sup>)
  - Saturated unit weight: 0.020 MN/m<sup>3</sup> (20 kN/m<sup>3</sup>)
  - **Validate**.

### Analysis settings:

1. Go to the **ANALYSIS** tab, then 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: 500
    - Tolerance: 0,001
  - Method of resolution: 1- initial stresses
  - Solver type: Multifrontal
3. **Validate**.

Edited by :



8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tél. : +33 1 49 76 12 59

[cesar-lcpc@itech-soft.com](mailto:cesar-lcpc@itech-soft.com)

[www.cesar-lcpc.com](http://www.cesar-lcpc.com)

© itech - 2020