



Tutorial 2g.03

Retaining wall construction

| Ref: CESAR-TUT(2g.03)-v2021.0.2-EN

1. PREVIEW

1.1. Tutorial Objectives

Before analysis of any retaining wall, it is important to have an accurate picture of the job site conditions. Every retaining wall must be engineered to withstand the pressure from the soils and other loads behind and above them. Standard gravity wall analysis considers sliding, bearing and overturning forces. On sites with slopes or surcharges, a global stability check will also be necessary.

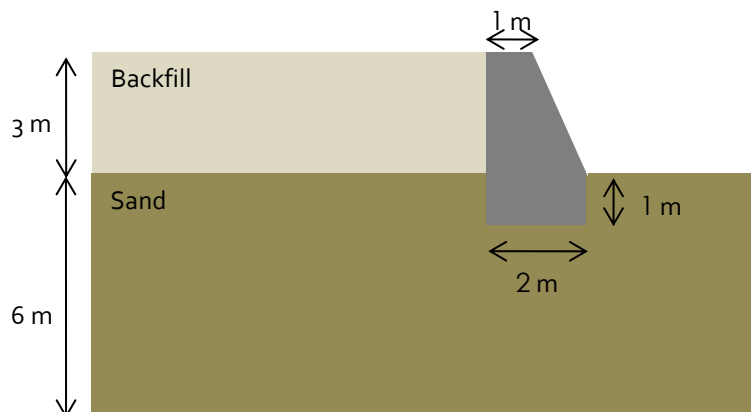
With CESAR-LCPC, we easily model the gravity wall and the stages of construction. The use of interface bodies to model the behaviour at the soil-structure interface is introduced. This will help to get an accurate value of forces applied to the wall.

1.2. Problem Specifications

General assumptions:

- Plane strain problem,
- Static analysis,
- No water table.

Geometry definition:



Boundary conditions:

A substratum is identified as the vertical bottom limit of the model.

Thus, supports conditions of the model are:

- Blocked horizontal displacements, u , on vertical sides.
- Blocked both horizontal, u , and vertical, v , displacements on bottom side.

The model dimensions are chosen according to the following usual recommendations.

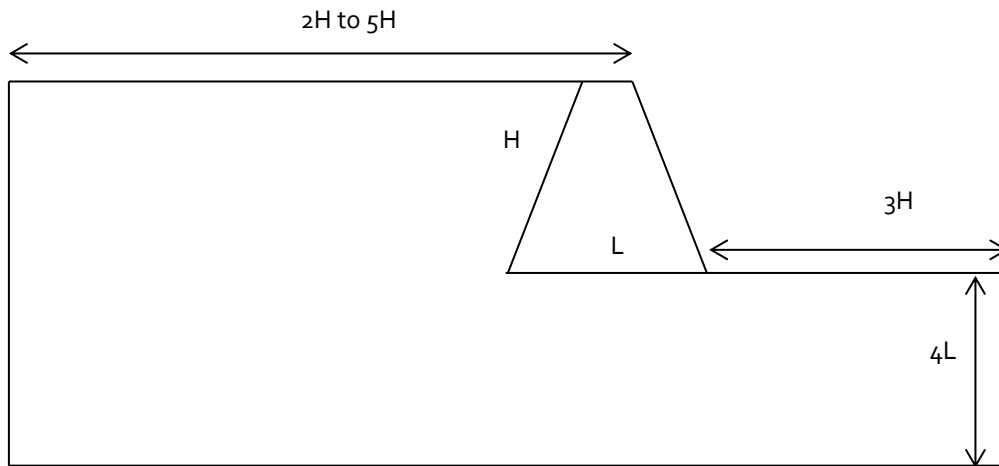


Figure 1 : Recommended dimensions for the modelling of a retaining wall

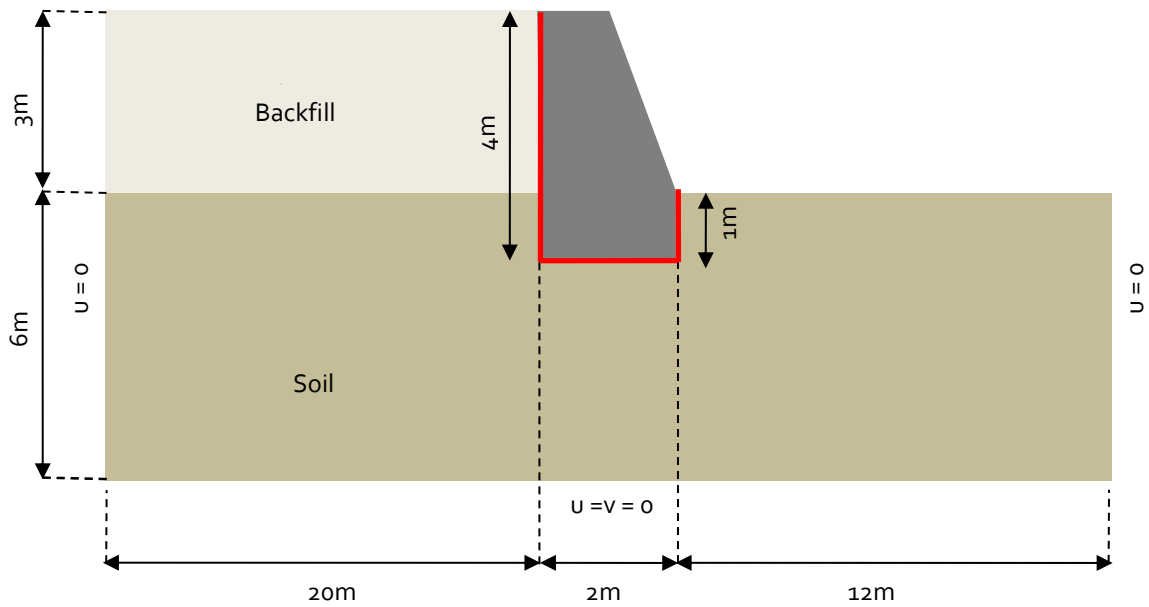


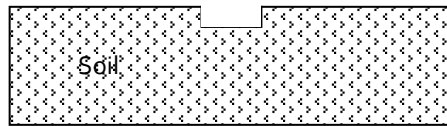
Figure 2 : Final dimensions of the model

Material Properties:

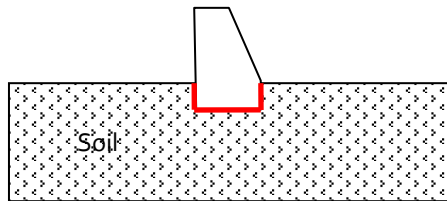
		γ_h (kN/m ³)	E (MPa)	ν	c' (kPa)	ϕ' (°)
Soil	Sand	19	50	0.3	0	36
Wall	Concrete	25	35000	0.2	-	-

Construction process:

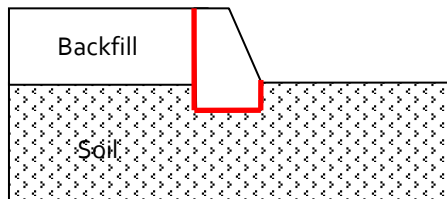
- Stage 0: Initial stress field
- Stage 1: Excavation in soil for the wall foundation.



- Stage 2: Building of the concrete wall with introduction of interface bodies at the soil-foundation interface (marked in red colour on the figure below).




- Stage 3: Backfill with introduction of interface bodies at the sand-wall interface (marked in red colour on the figure below).



- Stage 4: Analysis of the global stability, search of the safety factor using the c-phi reduction process.

2. GEOMETRY AND MESH

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 = 1m)





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

2.2. Geometry edition

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

Drawing of the geometry:

Using the tools **Points**  or **Lines**  and the grid snap, user will easily complete the geometry, as detailed below.

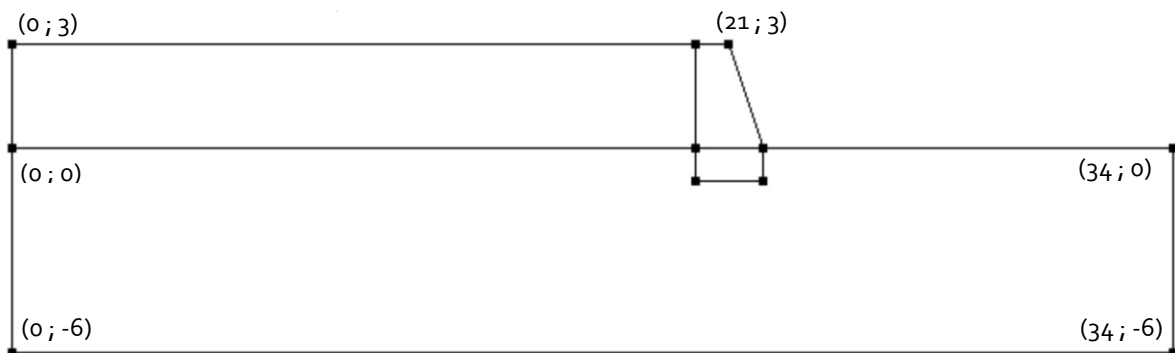




Figure 3: Edition of the geometry

Interface bodies:

Note here that we generate 2 sets of interfaces so that we can distinguish their behaviour.

1. Select the edges of the bottom part of the wall inside the sand.
2. Activate **Interface bodies** .
 - Activate **Interface bodies**
 - Set "Interface wall/sand" as body label.
 - **Apply**.
3. Select the edges of the top part of the wall in contact with the sand.

4. Activate **Interface element bodies** .
 - Set "Interface wall/backfill" as name of the body.
 - **Apply**.

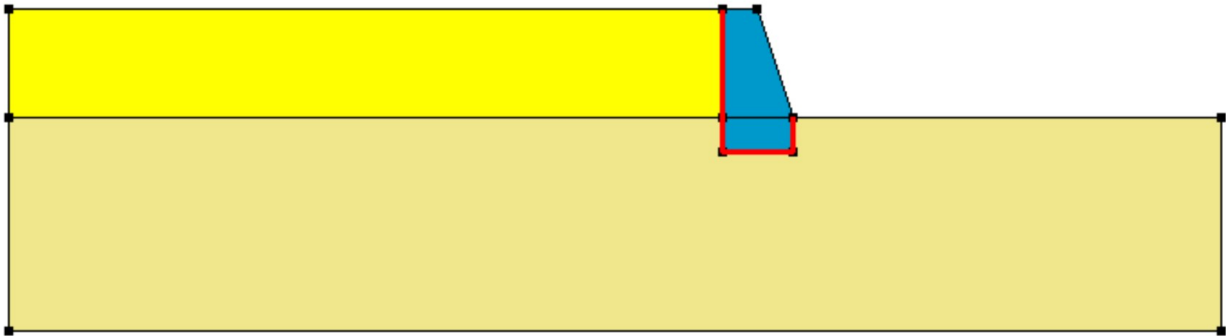





Figure 4: Final geometry

Bodies definition:


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

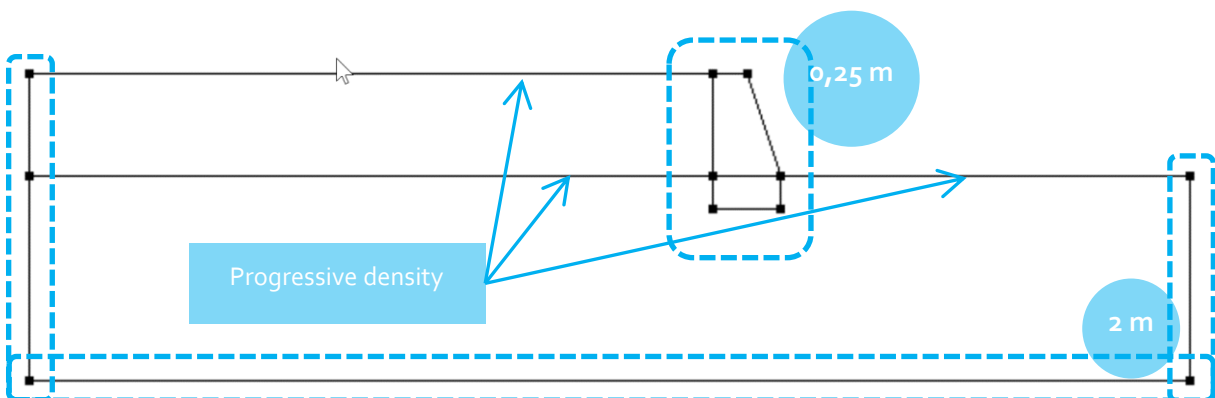
1. Click on  **Body properties**.
2. Right click on the area corresponding to the slope mass. Enter **Soil** as a name. **Apply**.
3. Repeat previous operation for other bodies.

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




2.3. 2D Meshing


Density definition:

 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.





1. Go to the **MESH** tab on the project flow bar to start the definition of divisions along lines.

2. Select the contour of the gravity wall. Click on  **Fixed length division** to impose a length for the divisions of this segment. Enter **0,25 m** in the dialog box. Click on **Apply**.
3. Select the external contours of the model. Click on  **Fixed length division** to impose a length for the divisions of this segment. Enter **2 m** in the dialog box. Click on **Apply**.
4. Click on  **Variable division** to divide the segment with a variation of lengths. Tick **First/last division** to define the method. Enter **0,25 m** as **First division** and **2 m** as **Last division**. Click on horizontal segment linking the wall with external borders.

 The position of the click defines where the initial division is.

Surface meshing:

1. Select the area corresponding to the *Gravity wall*, the *Sand* and the *Backfill*.
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. Adjust it in **Study settings > Preferences**: linear = coarse, cubic = dense.

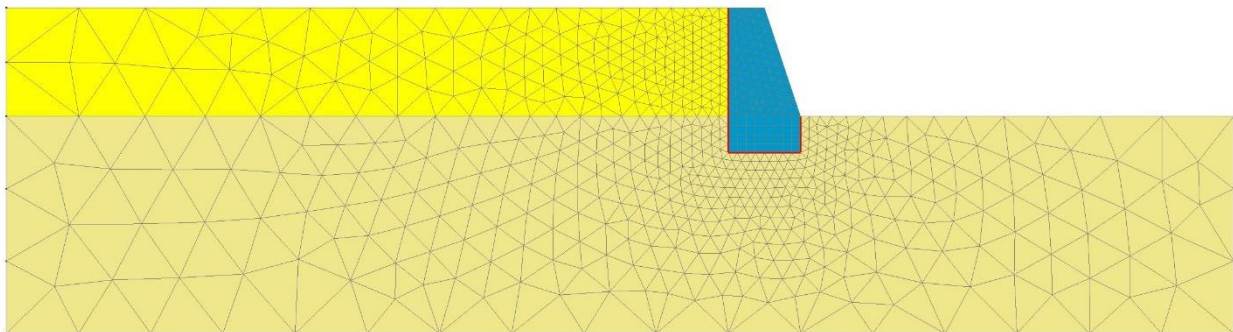


Figure 5: Example of mesh

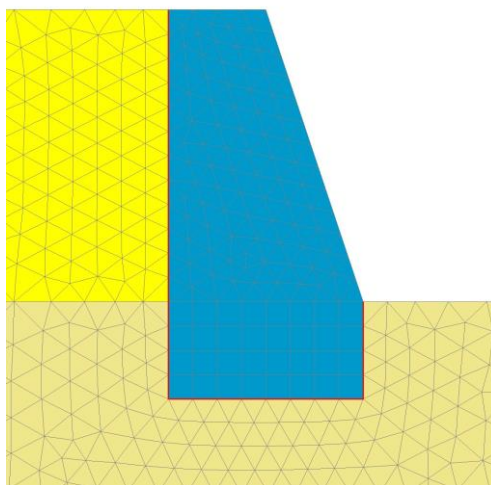


Figure 6: Zoom on the wall

3. CALCULATIONS SETTINGS

3.1. Initial stress field

The staged construction process requires the definition of an initial stress field before applying loads. For this project, we follow the procedure defined in the document "Getting started with CESAR-LCPC", Chapter "Initial stress field".

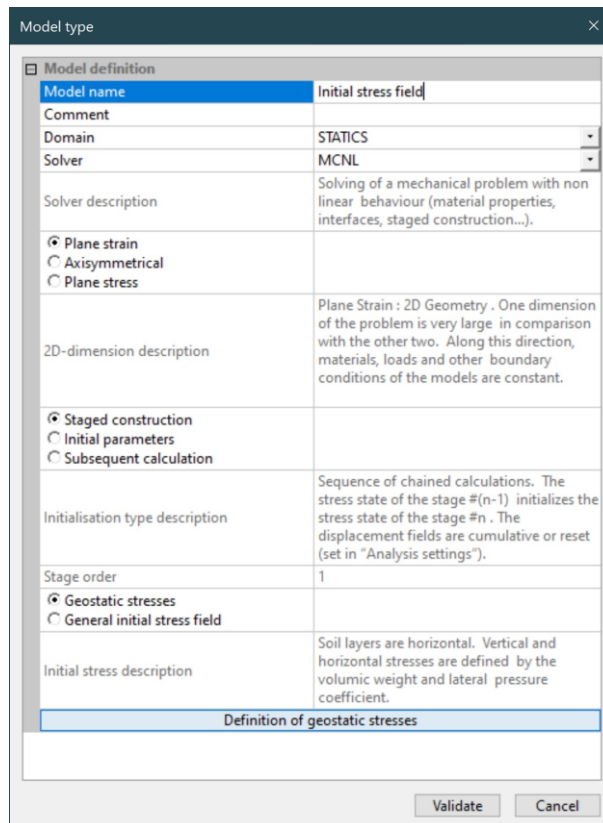
As the soil is composed of horizontal layers, the initial stress field is initialized with the Ko procedure.

Model definition:

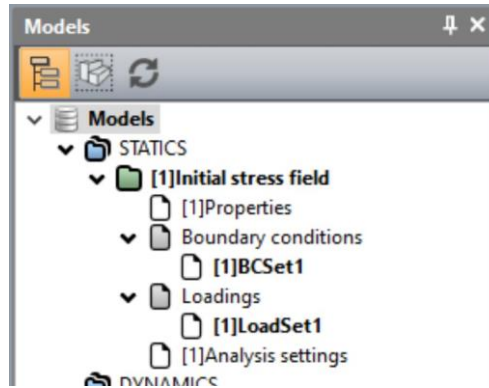
On the right side of the working window, the "Tree view" window displays the list of physical domains.

1. Right click on STATICS. Click on **Add a model**. A new toolbox is open for definition of the Model.
2. Enter **Initial stress field** as "Model name".
3. Select **MCNL** as "Solver".
4. Tick **Plane strain** as model configuration, with **Staged construction**.
5. Tick **Geostatic stresses** as initialization type. Click on **Definition of geostatic stresses**.
 - Click on **Insert** to define a new layer.
 - Enter the following values:

Height (m)	Volumic weight (MN/m ³)	Ko_x (-)	Ko_z (-)
0	0,019	0,5	0,5
 - Click on **Validate**.
6. Click on **Validate**.





The date tree is now as illustrated below.



Material properties for the solid bodies:



We initially define the material library of the study.

1. Go to **PROPERTIES** tab
2. Activate  **Properties for surface bodies.**
3. Give a name for the properties set name ("Concrete" for example).
4. In **Elasticity parameters**, chose "Isotropic linear elasticity" and define ρ , E and ν .
5. Click on  to create another properties set.
6. Give a name for the properties set name ("Sand" for example).
7. In **Elasticity parameters**, chose "Isotropic linear elasticity" and define ρ , E and ν .
8. In **Plasticity parameters**, chose "Mohr-Coulomb without hardening" and define c , ϕ and ψ .
9. Click on **Validate** and **Close**.

Name of the body	Law of elasticity	Plasticity criterion	γ_h (kg/m ³)	E (MPa)	ν	c' (MPa)	ϕ' (°)	ψ (°)
Wall	Isotropic linear	-	2500	35000	0,2	-	-	
Sand	Isotropic linear	Mohr-Coulomb	1900	50	0,3	0,005	36	0

Material properties for the interface bodies:

Properties of interfaces in CESAR-LCPC can be adherent, slipping or Coulomb's friction. In this case we chose Coulomb's friction with a friction angle equals to 2/3 of the soil one.

1. Click on  **Properties for interface bodies.**
2. Give a name for the properties set name ("Bonded" for example).
3. As **Type**, chose "Coulomb's friction" and define E_i , R_t , c , ϕ and ψ .
4. Click on  to create another properties set. Repeat operations 2 and 3. for definition of the friction interface.
5. Click on **Validate** and **Close**.

Name of the body	Interface type	Ei (MPa)	Rt (MPa)	c' (MPa)	ϕ' (°)	ψ (°)
Friction	Coulomb's friction	5000	1	0	24	0






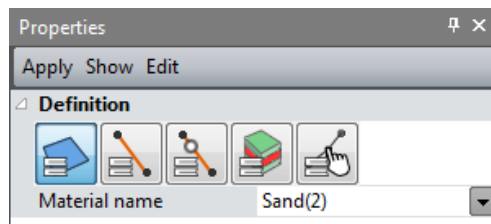
Recommendations for the properties of joint bodies:

- Young modulus, E_i , is the 100 times the value of the less stiff of two neighbouring materials;
- Tensile strength, R_t , is the limit value of the tensile stress normal to the interface necessary to cause debonding. Thus; input of a high value of R_t prevents the debonding.


Assignment of data sets:

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

1. Click on  **Assign properties** tool. On the left side, the grid is updated.
2. Click on  **Properties for surface bodies**.
 - Select the set of parameter "Sand"
 - Select the bodies of the soil mass, the backfill and the base of the wall.
 - **Apply**.
3. Click on  **Properties for interface bodies**.
 - Select all the interface body on the model and the set of parameters "Friction" in the list.
 - **Apply**.



Active/inactive bodies


1. Select the bodies above the soil mass at the initial state.
2. Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**.

The bodies are now marked with the neutral colour (grey is the default one).



Figure 7: Status of the model when bodies are set inactive at stage #1

Boundary conditions:

1. Activate 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.



Figure 8: Display of the boundary conditions





Default name of the boundary condition, BCSet1, set can be edited using the [F2] key.

Loading:

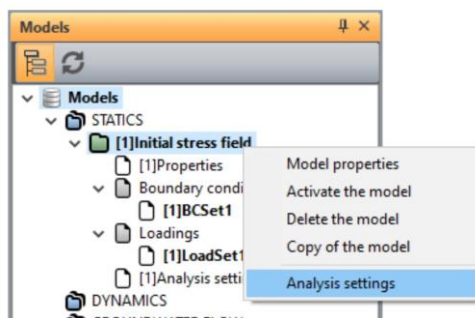
There is no load to be defined at this step of the analysis as we are initializing the initial stress field.

Analysis settings:

1. Activate the **ANALYSIS** tab.
2. Click on  **Options**. Set the numbering option to "No renumbering".
3. Click on  **Analysis settings**. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 1
 - Max number of iterations per increment: 1000
 - Tolerance: 0,01
 - Algorithm type:
 - Solution method: 1-Method of initial stresses
 - Algorithm type: Pardiso
 - Analysis type: Standard
4. **Validate** to close.



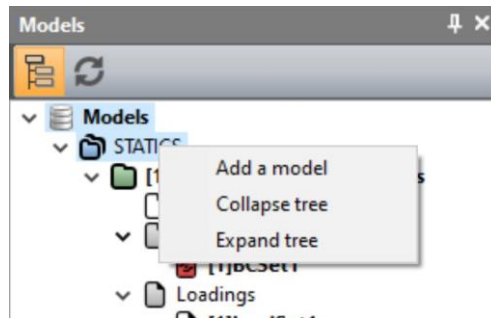
User can also use the STUDY tree to access these settings: right click on the calculation title. In the displayed list, select **Analysis settings**.

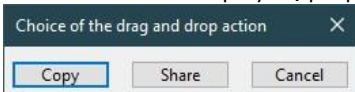


3.2. Stage #1, excavation for wall foundation

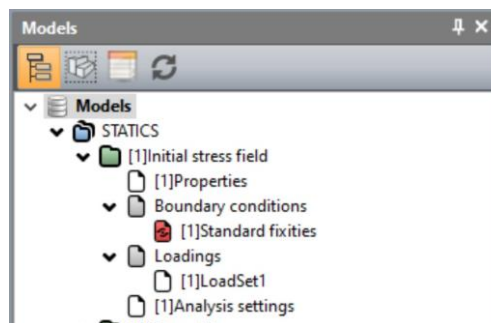
In this step, we remove the soil where the wall will be constructed.

Model definition:



1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.
2. A toolbox is displayed for definition of this new model.
 - Set the model name: **Stage #1**.
 - Select staged construction. Default order is set to 2.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of **Initial stress field** to **Stage #1**.
 - Select "Properties" of **Initial stress field**,
 - Place it on "Properties" of **Stage #1**
 - A toolbox is now displayed, proposing several options:

Select **Copy**.
4. Using "drag and drop" action, copy the Boundary conditions set of **Initial stress field** to **Stage #1**.
 - Select the "Standard fixities" set of **Initial stress field**,
 - Place it on "BCSet1" of **Stage #1**
 - A toolbox is now displayed, proposing several options:
Select **Share** as the boundary conditions will not change during all the calculation stages.

The date tree is now as illustrated below.



Active/inactive bodies


1. Select the body of the lower part of the wall.
2. Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**.




Figure 9: Status of the model when bodies are set inactive at stage #1

Boundary conditions:

No changes.

Loading:

1. Activate the **LOADS** tab.
2. On the toolbar, activate  **Excavation forces**. This feature is described in the user manual.
 - Activate the excavation forces,
 - Set the value of lambda to 1,
 - Select **Initial stress field** as stress field origin.
3. **Apply**.

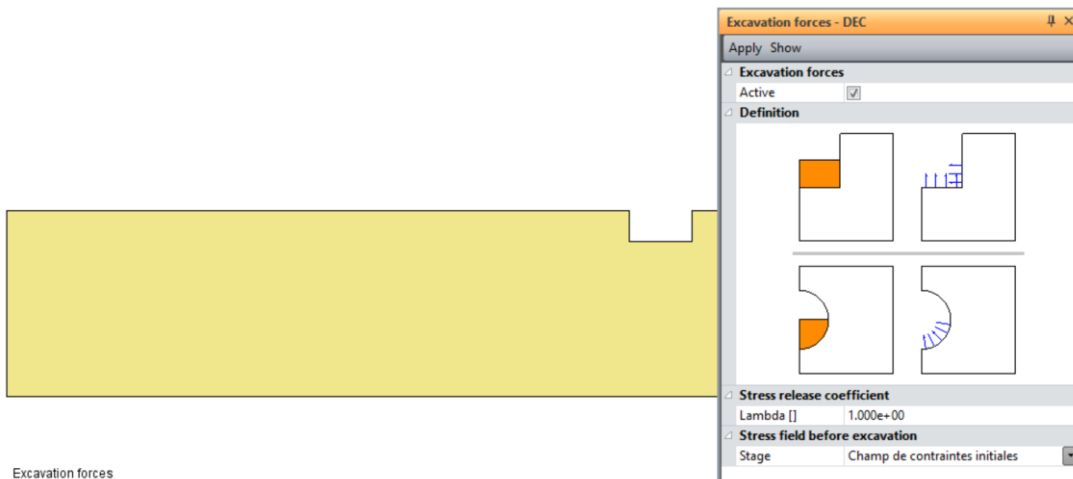


Figure 10: Display of the excavation forces

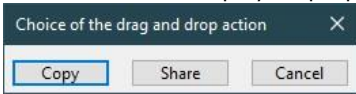
Analysis settings:

No changes.

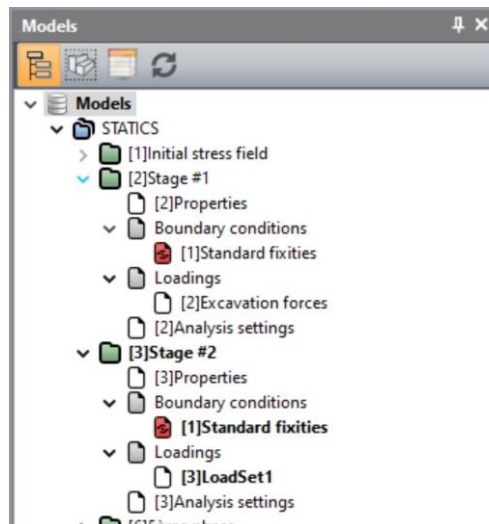
3.3. Stage #2, wall construction

In this step, we activate the wall. To unbalance the model, the self-weight of the wall concrete is applied.


Model definition:

1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.
2. A toolbox is displayed for definition of this new model.
 - Set the model name: **Stage #2**.
 - Select staged construction. Default order is set to 3.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of **Stage #1** to **Stage #2**.
 - Select "Properties" of **Stage #1**,
 - Place it on "Properties" of **Stage #2**
 - A toolbox is now displayed, proposing several options:

Select **Copy**.
4. Using "drag and drop" action, copy the Boundary conditions set of **Stage #1** to **Stage #2**.
 - Select the "Standard fixities" set of **Stage #1**,
 - Place it on "BCSet1" of **Stage #2**
 - A toolbox is now displayed, proposing several options:
Select **Share** as the boundary conditions will not change during all the calculation stages.

The date tree is now as illustrated below.



Active/inactive bodies

1. Activate the **PROPERTIES** tab.
2. Select the bodies of the **Wall**.
3. Click on  **Activate/deactivate bodies**.
4. In the grid, tick "Active".
5. **Apply**. The bodies are now marked with their initial colour.

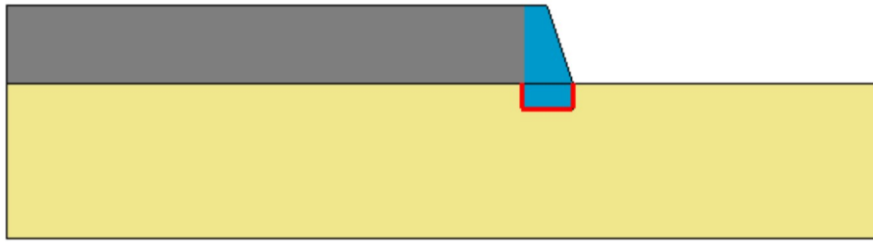




Figure 11: Status of the model when groups are set active at stage #2

Interface properties


1. Activate the **PROPERTIES** tab.
2. Click on  **Assign properties** tool.
3. Click on  **Properties for interface bodies**.
 - Select all the interface bodies and the set "Sliding" in the list.
 - **Apply**.

Boundary conditions:

There is no change in the boundary conditions.

Loading:

We assign the self-weight of the wall.

1. Activate the **LOADS** tab.
2. On the toolbar, activate  **Gravity forces**.
 - Click on **Automatic selection**. This feature detects the activated bodies.
 - **Apply**.

The gravity forces activation is mentioned as a label at the bottom left of the model window.

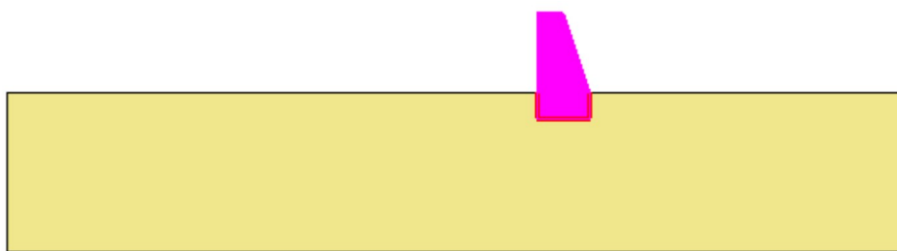


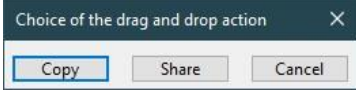
Figure 12: Gravity forces applied on the wall

Analysis settings:

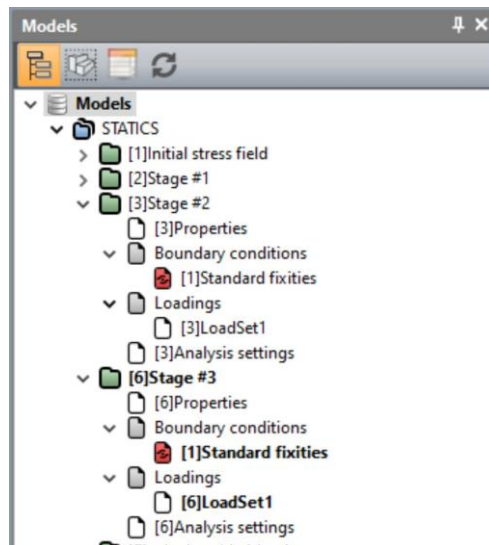
There is no change in the analysis settings.

3.4. Stage #3, backfill works


Model definition:

1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.
2. A toolbox is displayed for definition of this new model.
 - Set the model name: **Stage #3**.
 - Select staged construction. Default order is set to 4.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of **Stage #2** to **Stage #3**.
 - Select "Properties" of **Stage #2**,
 - Place it on "Properties" of **Stage #3**
 - A toolbox is now displayed, proposing several options:

Select **Copy**.
4. Using "drag and drop" action, copy the Boundary conditions set of **Stage #2** to **Stage #3**.
 - Select the "Standard fixities" set of **Stage #2**,
 - Place it on "BCSet1" of **Stage #3**
 - A toolbox is now displayed, proposing several options:
Select **Share** as the boundary conditions will not change during all the calculation stages.

The date tree is now as illustrated below.



Active/inactive bodies

1. Activate the **PROPERTIES** tab.
2. Select the first backfill layer behind the wall.
3. Click on  **Activate/deactivate bodies**.
4. In the grid, tick "Active".
5. **Apply**. The body is now marked with its initial colour.

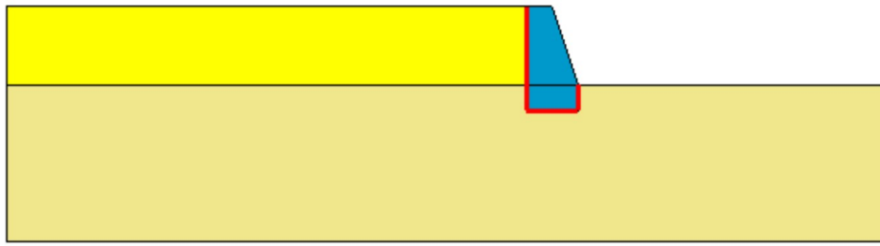



Figure 13: Status of the model when groups are set active at stage #3

Boundary conditions:

As we activate the sand layer, we fix the boundary on its limit.

1. Activate the **BOUNDARY CONDITIONS** tab.
2. On the toolbar, activate  to define side and bottom supports.
3. **Apply**. Supports are automatically updated to the limits of the mesh, including the activated sand layer.

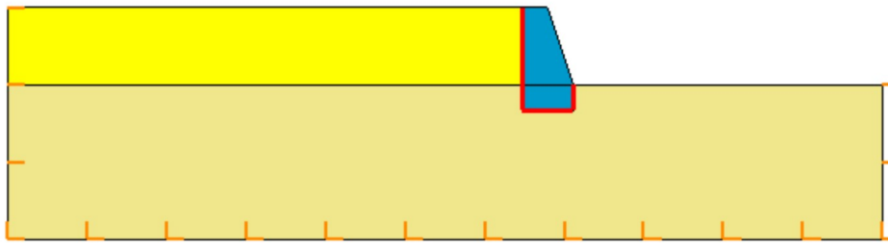



Figure 14: Status of the model when boundary conditions are updated at Stage #3

Loading:

1. Activate the **LOADS** tab.
2. On the toolbar, activate  **Gravity forces**.
 - Click on **Automatic selection**. This feature detects the groups activated in the actual stage (the sand body).
 - **Apply**.

Analysis settings:

There is no change in the analysis settings.

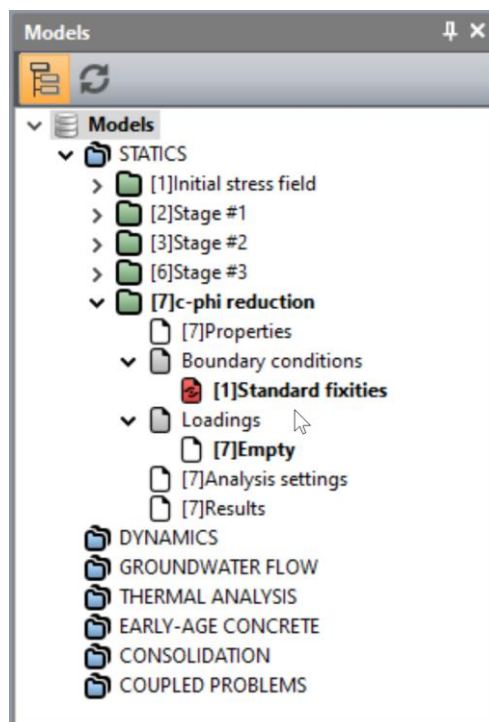
3.5. C-phi reduction analysis

The c-phi reduction starts from the existing stress field state - the one calculated at stage #3. No changes from the Stage #3 edition, exception made of the loading sets (no load is applied) and the calculation settings.


Model definition:

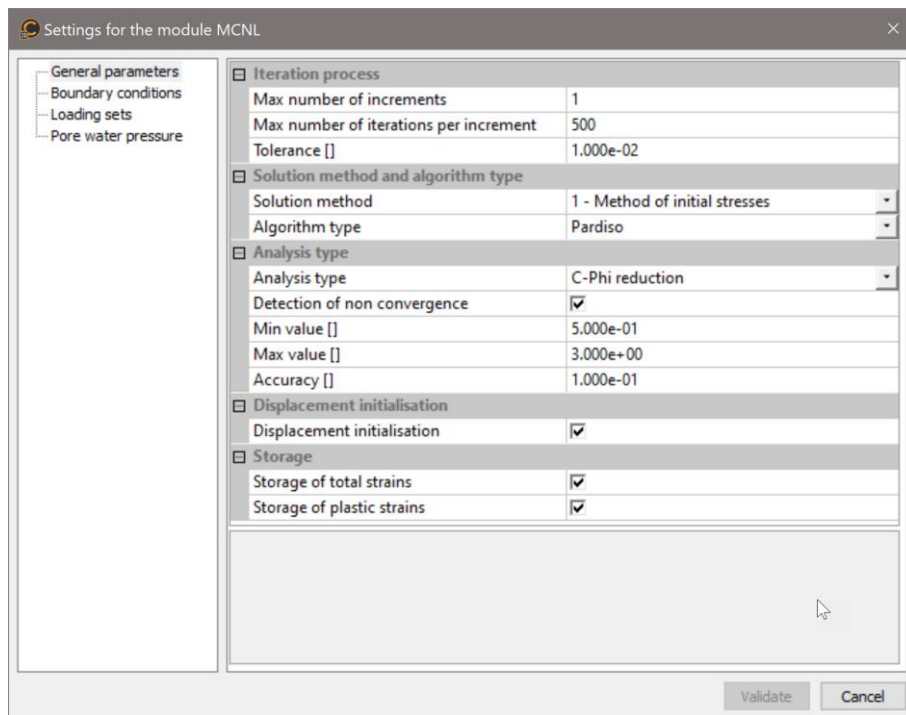
1. In the data tree, right click on the previous model title (**Stage #3**). In the displayed list, select **Copy of the model**.
2. A toolbox is displayed for definition of this new model.
 - Change the model name: **c-phi reduction**.
 - Select staged construction. Default order is set to 6.
3. **Validate**.
4. A toolbox is displayed for setting the sharing options.
 - Untick "Properties",
 - Tick "Boundary conditions",
 - Untick "Loadings".

The date tree is now as illustrated below.




Analysis settings:


1. Activate the **ANALYSIS** tab.
2. Click on  **Analysis settings**. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 1
 - Max number of iterations per increment: 1000
 - Tolerance: 0,01
 - Algorithm type:
 - Solution method: 1-Method of initial stresses
 - Algorithm type: Pardiso
 - Analysis type:
 - Detection of non convergence:
 - Min value: 0,5
 - Max value: 3
 - Accuracy: 0,1
3. **Validate** to close.

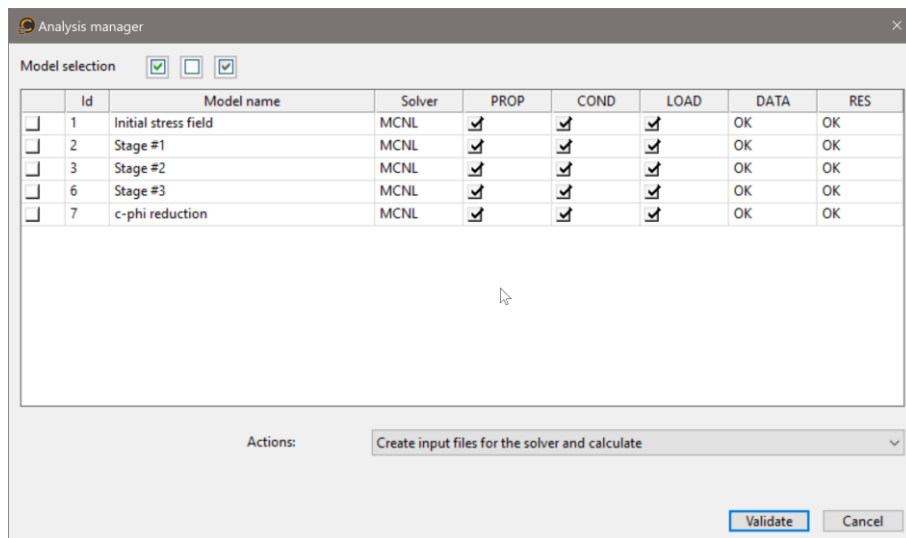



4. SOLVE

We launch the calculations simultaneously. It is obviously possible to launch the calculations one by one.

1. Activate the **ANALYSIS** tab.
2. Click on  **Analysis manager**.
3. Select all the models.
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".

 We detect here if the models are ready for calculation. All steps (PROP, COND, CHAR) should be validated with a tick mark.






 All the messages during the analysis will be shown on the working space. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct.

5. RESULTS

The safety factor reached with the c-phi reduction analysis is 2,45. For information, when choosing sliding interface, the safety factor is reduced to 1,87.

Typical results obtained at the end of the last phase are provided in the following figures.

1. Activate the **RESULTS** tab.
2. Click on  **Result Types**
 - Select **Deformed** as Mesh,
 - Tick **Contour plots**, in the list select **|u|** (total displacements),
 - Click on **Apply**.
3. Click on  **Contour plots settings**.
 - Tick **Contouring**, as isovalues style,
 - Tick **Contour lines** and select **Grey** color,
 - **Apply**.
4. Click on  **Displacement settings**.
 - Select **Manual** as scale type,
 - Set that a value of **100mm** is represented by **2m**,
 - **Validate**.

The figure below clearly illustrates the rupture mechanisms at the front and at the back of the wall.

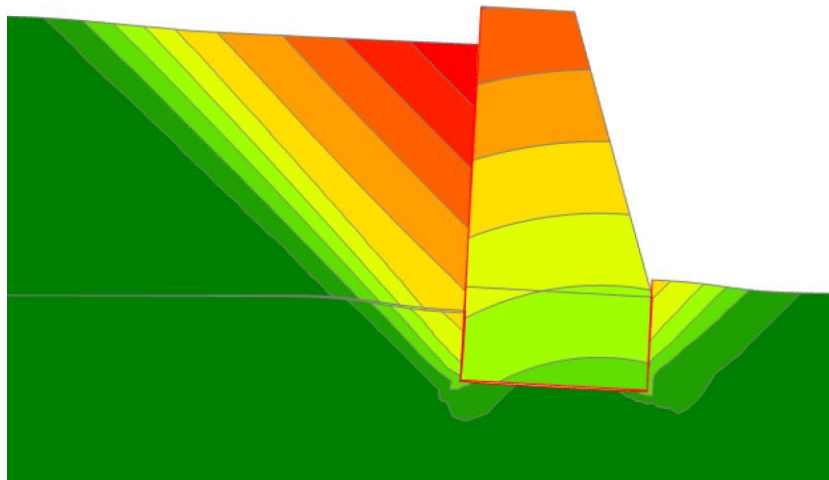




Figure 15: Total displacements $|u|$ with sliding interfaces.

In order to emphasize this rupture scheme, we display one of the plastic strains.

1. Click on  **Result Types**
 - Select **Deformed** as Mesh,
 - Tick **Contour plots**, in the list select **ϵ_{1p}** ,
 - Click on **Apply**.

Display of the scalar can be set inactive in some bodies (for example in the wall) using the tool  **Display results**.

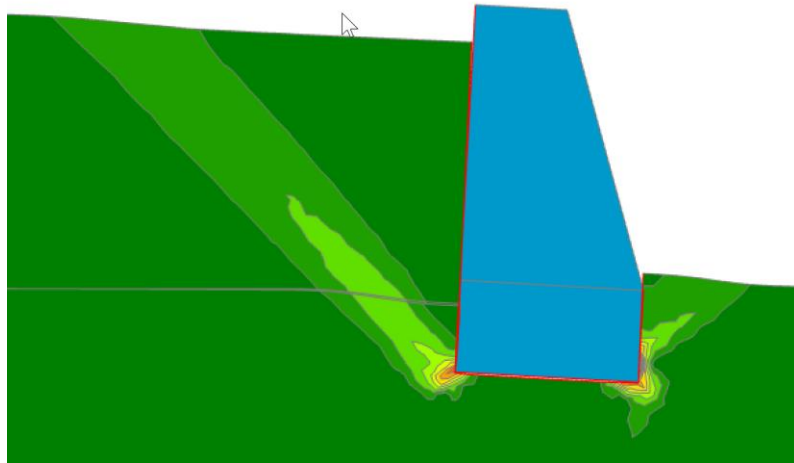





Figure 16: Display of plastic strains ϵ_{1p}

The wall rotation at the end of construction is derived from the horizontal displacement along the back of the wall.

To get this displacement, we generate a chart along a line cut. Line cut is defined following this procedure:

1. Activate the **CHARTS** tab.
2. Select the edges at the extrados of the wall.
3. Click on  **.Lines set.**
 - Give a name to this line set, **Back** for example.
 - **Add.**
4. Activate  **Charts for a line set.**
 - Select **u** as parameter,
 - Select **Back** as line set.
 - **Apply.**

The line set is created. It is displayed on the mesh with arrows indicating the orientation. This orientation can be modified using  **Inverse orientation.**

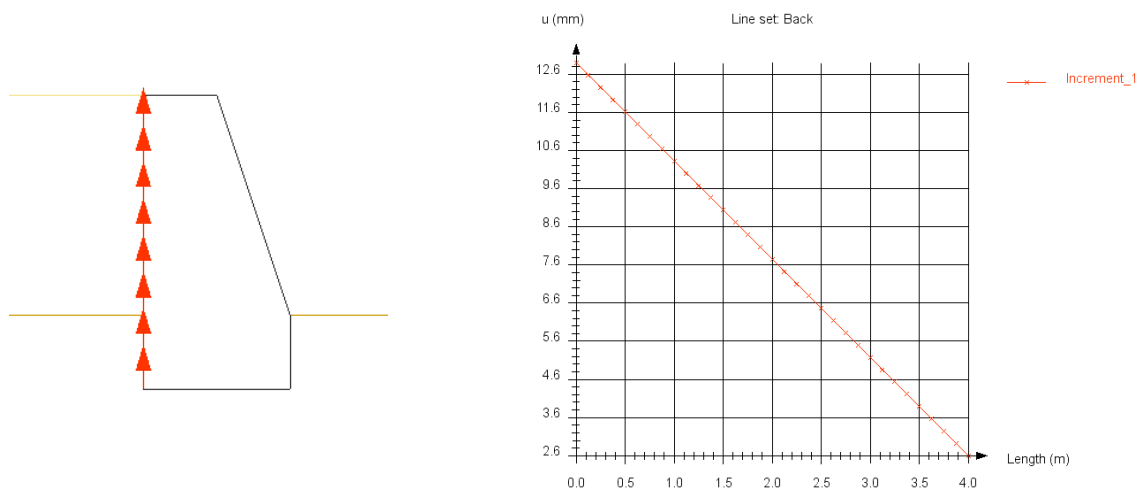


Figure 17: Plot of horizontal displacement at the back of the wall

Edited by:



8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tel.: +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com

© itech - 2020