



Tutorial 2g.01

Anchored diaphragm wall

Ref.: CESAR-TUT(2g.01)-v2021.0.3-EN

1. PREVIEW

1.1. Tutorial Objectives

Excavation works in urban areas are often required for foundations of buildings. During these works, one must limit the interactions with existing constructions at the surface as well as with underground constructions (tunnels, sewers, other deep foundations). A staged analysis with appropriate modelling of structures, soil/structures interfaces and soil behaviour is therefore very helpful for the computation of differential settlements and efforts in the structures.

The present tutorial will guide the user through the steps of construction of the model, introducing the relevant tools offered by CESAR 2D.

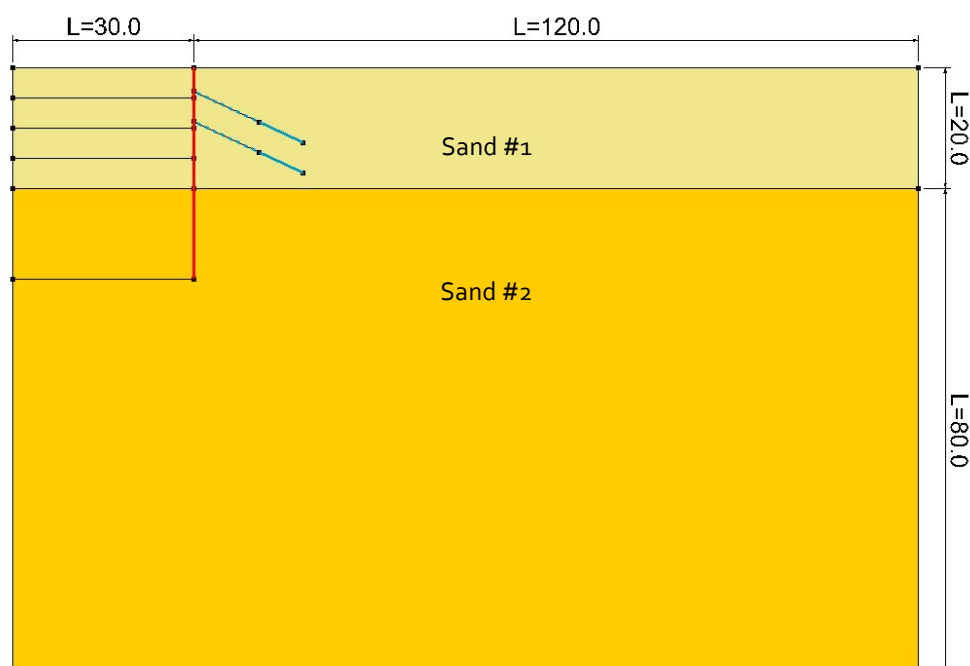
1.2. Problem specifications

Stages of construction

1. Effective initial stress field.
2. "Wished in place" setup of the wall setup and WTB lowering at -5 m
3. First excavation at -5 m
4. Anchor #1 setup
5. Water table lowering at -10 m
6. Excavation at -10 m
7. Anchor #2 setup
8. Water table lowering at -15 m
9. Excavation at -15 m


In the scope of the tutorial, we will limit to the description of the 6 first stages. User can proceed with the following stages of the analysis, copying of previous ones.

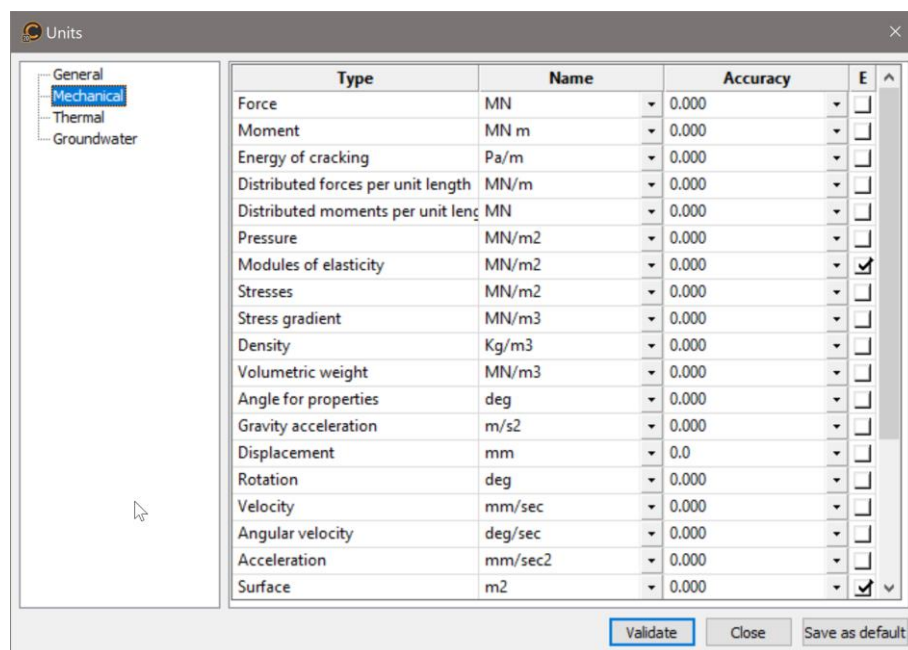
Geometry



2. GEOMETRY AND MESH

2.1. General settings

1. Run CLEO2D.
2. Set the units in the menu **Units**.
 - Select the **General/Length** option and set the unit **m** in the bottom left combo box.
 - Select the **Mechanical/Force** option and set the unit **MN** in the bottom left combo box.
 - Select the **Mechanical/Displacement** option and set the unit **mm**.
 - Select the **Mechanical/Surface** option and activate the scientific display (E).
 - Click on **Validate** to close.
3. In  **Working plane**, set the visible grid to 5 m (dX = dY = 5 m)





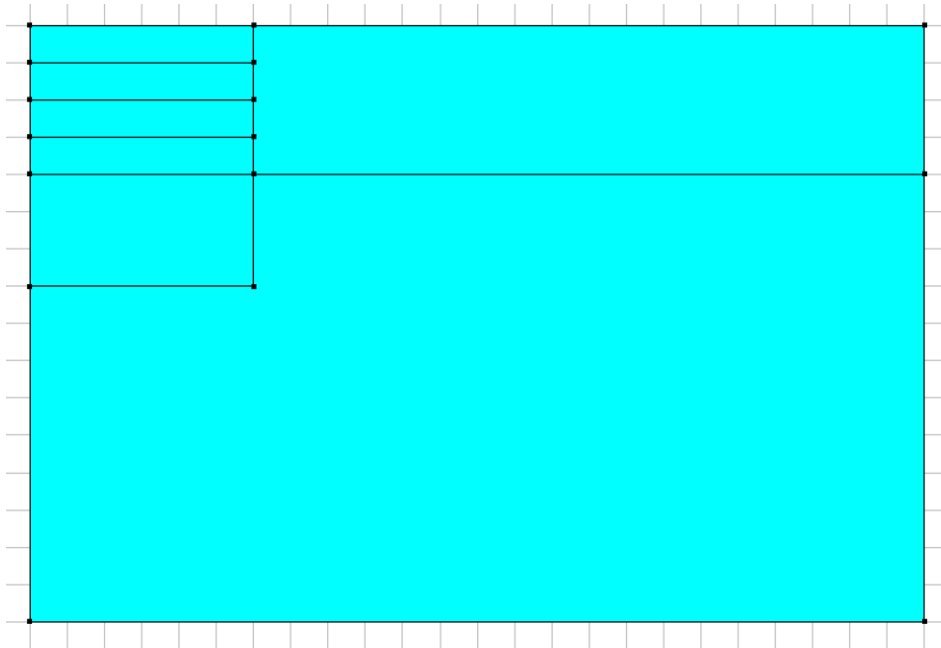
Use "Save as default" to set this system of units as your user environment.

2.2. Geometry

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


Drawing of the project:

Use the **Points**  and/or the **Lines**  feature to draw the geometry with help of the grid. We set the top of the upper layer at y = 0.




2.3. Bodies

Wall



1. Select the edges from 0 to -35 m at $x = 30$ m.
2. Activate tool **1D-bodies** .
 - Definition option: "Selection"
 - Body properties: "Generic"
3. **Apply**

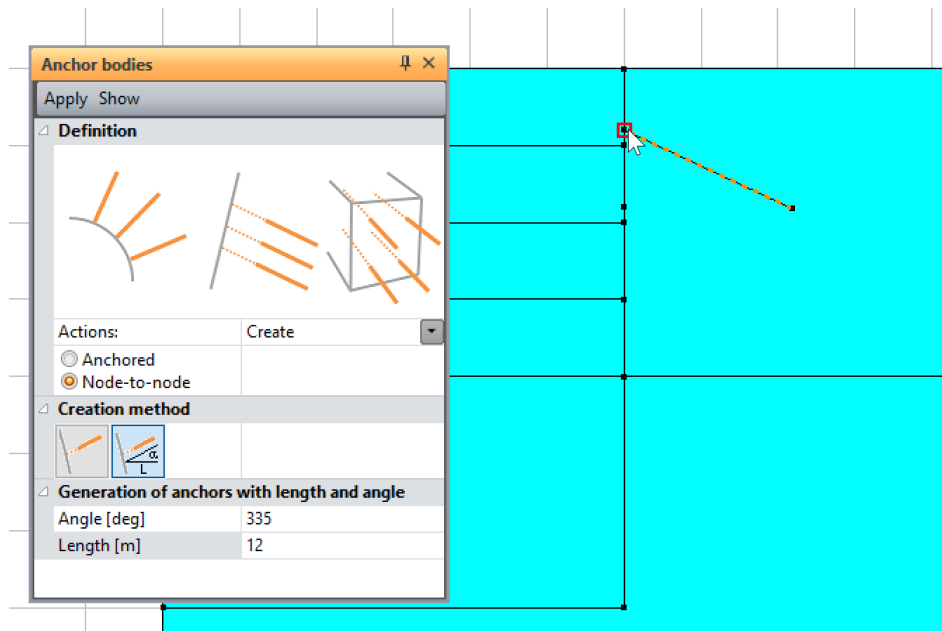
Anchors


First, define the attachment point of the anchors on the wall.

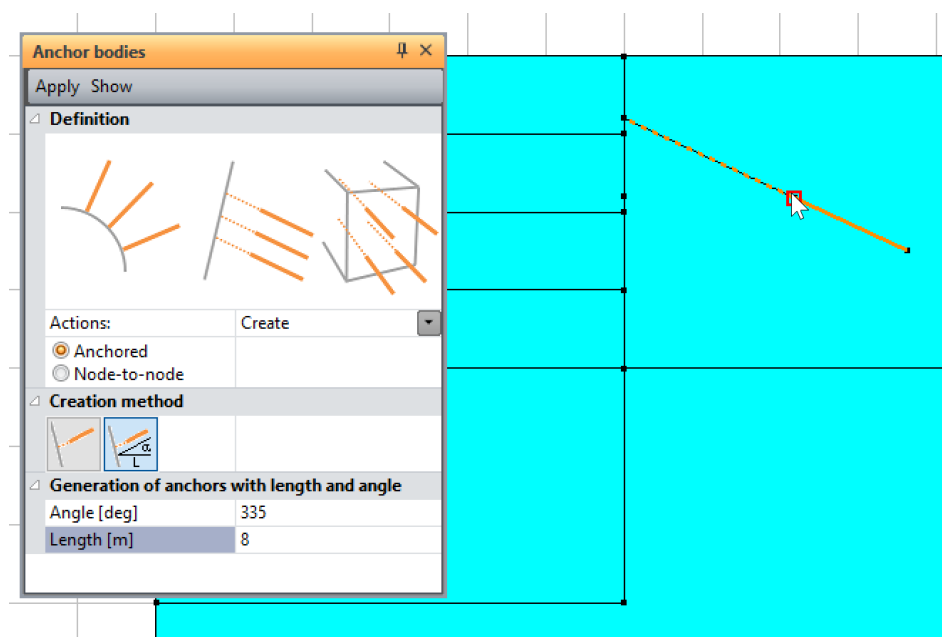
1. Use the on **Points** .
2. Set the first attachment point with coordinates: $x = 30$ m : $y = -4$ m.
3. **Apply**.

Now, we define the first anchor: a free part (node-to-node 1D-body) and a bonded part (anchored 1D-body).


1. Activate tool **Anchor-bodies** .
2. Action: "Create"
3. Body properties: "Node-to-node anchor"
4. Creation method:
 -  "Length/Angle"
 - Set length $L = 12$ m and angle $A = 335^\circ$
5. Click on the attachment point (on the wall).



6. Change body properties: "Anchored"
7. Creation method:
 -  "Length/Angle"
 - Set length $L = 8 \text{ m}$ and angle $A = 335^\circ$
8. Click on the attachment point, the end of the previous node-to-node anchor.



The second anchor is simply generated by copy of the first one.

1. Activate **Operations on bodies** 
 - Operation type: "Translation"
 - Number of operations: 1
 - Translation: $VX = 0$; $Vy = -5 \text{ m}$
2. Select both parts of the first anchor.
3. **Apply.**

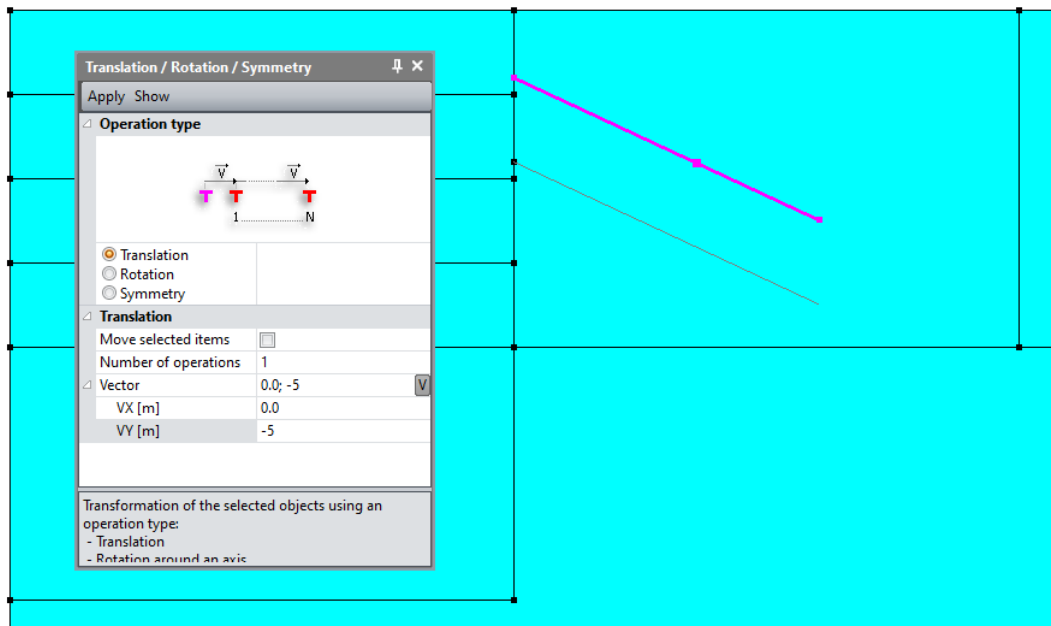




Figure 1: "Preview" of the second anchor before translation action

Interface bodies:

1. Select the part of the wall in layer **Sand #1**.
 - Activate the feature **Interface bodies** .
 - Set the "Body label": **Wall-Sand#1**
 - **Apply**.
2. Select the part of the wall in layer **Sand #2**.
 - Activate the feature **Interface bodies** .
 - Set the "Body label": **Wall-Sand#2**
 - **Apply**.

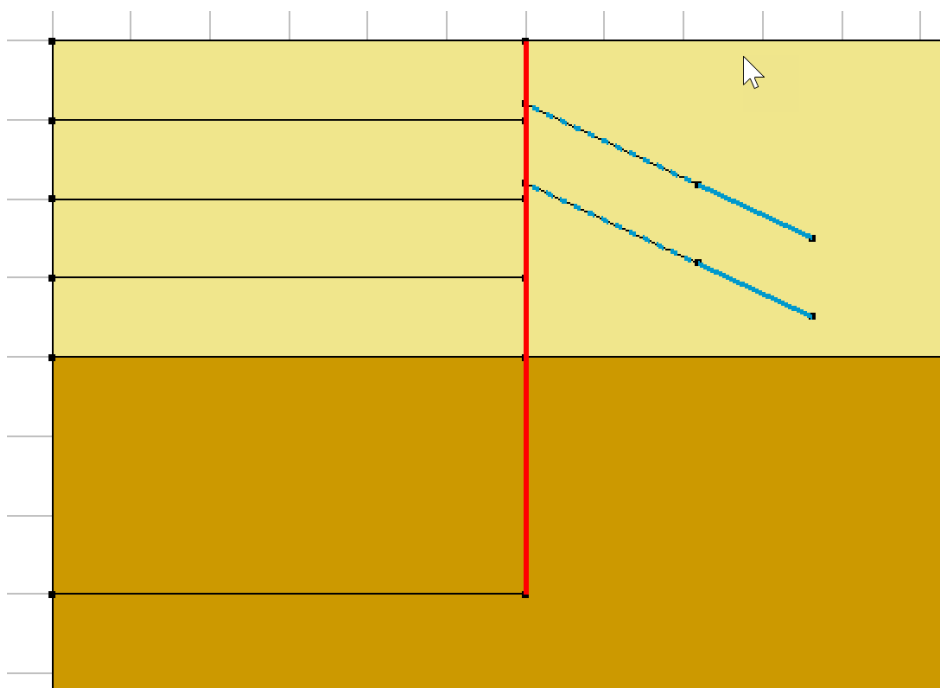





Figure 2: View of the model after colours application

2.4. Mesh

Density definition:

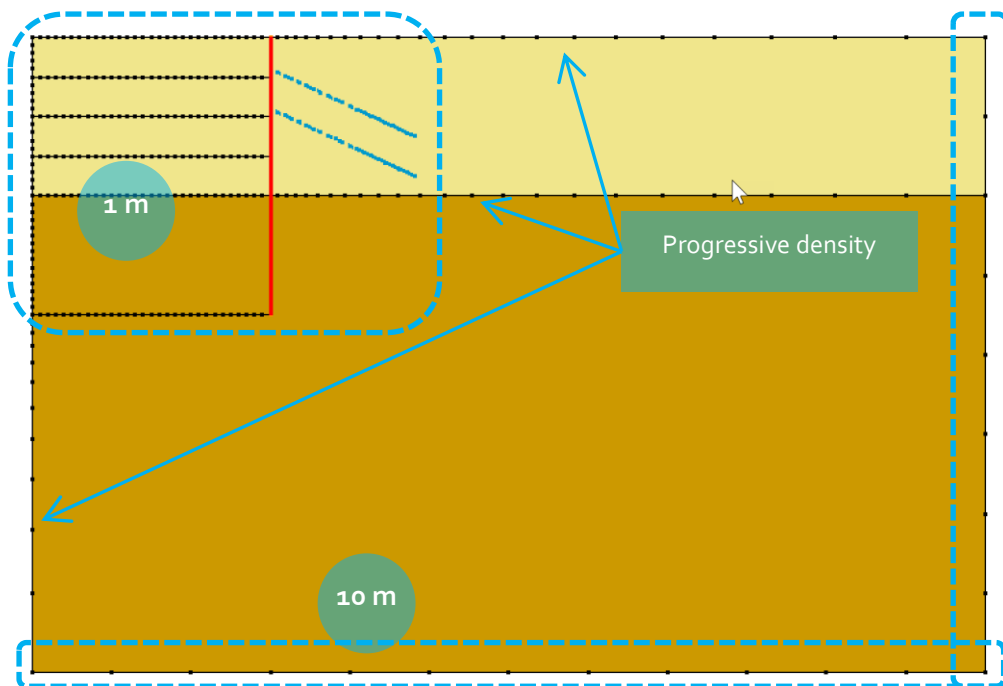


Define dense divisions in the area of high strains, i.e. in the vicinity of the wall and the anchors. Use a progressive density definition to generate a progressive evolution from small segments to large segments on the boundary edges.


1. Go to the **MESH** tab on the project flow bar to start the definition of divisions along lines.
2. Select all the contours in the box from (0 ; 0) to (60 ; -35).
3. Click on  **Fixed length density** to impose a length for the divisions of these segments. Enter **1 m** in the dialog box. Click on **Apply**.
4. Select the external contours of the model (segments at x = 150 m and y = -100 m).
5. Click on  **Fixed length density** to impose a length for the divisions of this segment. Enter **10 m** in the dialog box. Click on **Apply**.
6. Click on  **Progressive density** to divide the segment with a variation of lengths. Tick **First division** and **Last division** to define the method. Enter **1 m** as **First division** and **10 m** as **Last division**. Click on the segments between the box and external edges



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




1D-Body meshing

1. Select all the 1D bodies.
2. Click on the **1D-Body meshing** tool  . Chose **Quadratic** as interpolation type.
3. **Apply**.
4. Select the node-to-node anchors.
5. Chose **Linear** as interpolation type.
6. **Apply**.

Surface meshing:

It is important to notice that the anchors must be meshed before the surface bodies so that the surface mesher can connect to them.

1. Select the surface bodies.
2. Click on the **Surface meshing** tool . Chose **Quadratic** as interpolation type, **Triangle** as element shape and **Native** as surface mesh generator.
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 **Preferences> Study settings**: linear interpolation = coarse, cubic interpolation = dense.

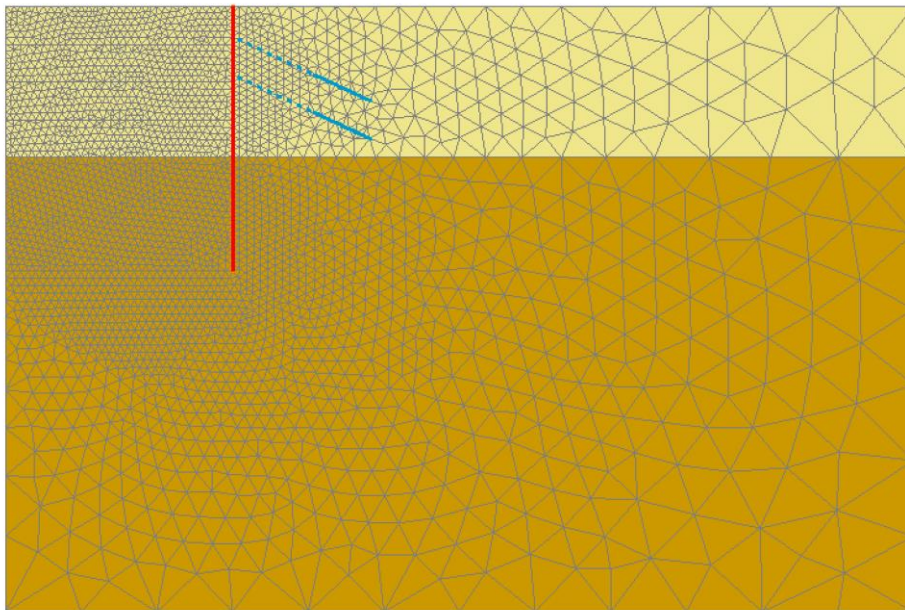


Figure 3: view of the mesh

3. CALCULATIONS SETTINGS

3.1. Stage #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 "CESAR 2D – User manual", chapter "Initial stress field".

The initial stress field is here an effective one defined by the water table position.

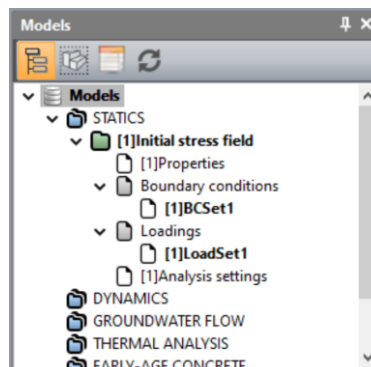
Model definition:

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 **Initial stress field** as "Model name".
4. Select **MCNL** as "Solver".
5. Tick "Plane strain" as model configuration, with "Staged construction".
6. Tick "General initial stress field" as initialization type.
7. Click on **Validate**.

Model definition	
Model name	Initial stress field
Comment	Stress field definition with WTB position
Domain	STATICS
Solver	MCNL
Solver description	Solving of a mechanical problem with non linear behaviour (material properties, interfaces, staged construction...).
<input checked="" type="radio"/> Plane strain <input type="radio"/> Axisymmetrical <input type="radio"/> Plane stress	
2D-dimension description	Plane Strain : 2D Geometry . One dimension of the problem is very large in comparison with the other two. Along this direction, materials, loads and other boundary conditions of the models are constant.
<input checked="" type="radio"/> Staged construction <input type="radio"/> Initial parameters <input type="radio"/> Subsequent calculation	
Initialisation type description	Sequence of chained calculations. The stress state of the stage #(n-1) initializes the stress state of the stage #n . The displacement fields are cumulative or reset (set in "Analysis settings").
Stage order	1
<input type="radio"/> Geostatic stresses <input checked="" type="radio"/> General initial stress field	
Initial stress description	The stress field initialisation is made by gravity loading of the soils either dry (total stresses) or saturated depending of the WTB position (effective stresses). Other options are offered.



Validate Cancel

The date tree is now as illustrated:





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 (**Sand #1** for example).
4. In **Elasticity parameters**, chose "Nonlinear elasticity (HSM) (+)" and define the requested parameters (cf. table of properties). Let "Undrained behaviour" as inactive.
5. In **Plasticity parameters**, chose "Plasticity criterions (user's combination)" then "Hardening Soil Model (HSM) (+)" as Criterion #1. Define the requested parameters (cf. table below).
6. Click on  to create other properties set. Repeat operations 3 to 5 for definition of a second set of material properties for **Sand #2**.
7. Click on **Validate** and **Close**.

Material properties for the interface bodies:

Properties of interfaces in CESAR-LCPC can be bond, sliding or Coulomb's friction. In this project, friction behaviour is set between wall and soil so that we can have an interaction coefficient of $2/3$.

1. Click on  **Properties for interface bodies.**
2. Give a name for the properties set name ("Contact wall/sand #1" for example).
3. Use "Joint" as interface element (default type).
4. As **Type**, chose "Coulomb's friction" and define E_i , R_t , c , ϕ and ψ . with a friction angle equals to $2/3$ of the soil one
5. Click on  to create other properties set. Repeat operations 2 and 3. for definition of the contact between wall and sand.
6. Click on **Validate** and **Close**.

Name of the set	Interface type	E_i (MPa)	R_t (MPa)	c (MPa)	ϕ (°)	ψ (°)
Wall/Sand #1	Coulomb's friction	18000	1	0	23,3	0
Wall/Sand #2	Coulomb's friction	30000	1	0	25,3	0




Recommendations for the properties of joint elements:

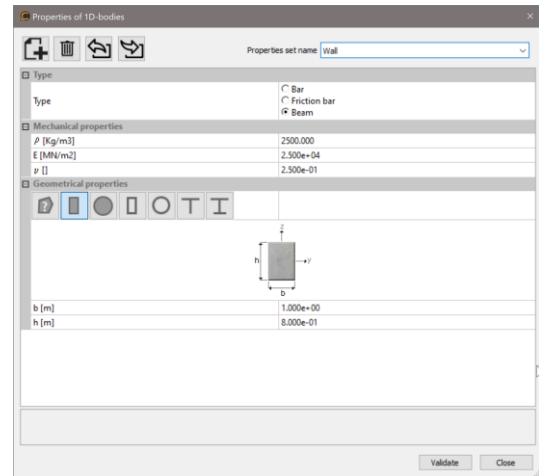
- Young modulus, E_i , is 100 times the value of the less stiff of two neighbouring materials;
- Tensile strength, R_t , is the limit value of the tensile stress normal to the interface necessary to cause debonding. Input of a high value of R_t would prevent the debonding.

Material properties for the wall

The model is built under plane strain conditions; therefore, user must keep in mind that we consider 1 m in out-of-plane direction. When defining the wall properties set, this one has to be adapted to this condition.

The wall is made of concrete and is 80 cm thick.

1. Click on  **Properties for 1D bodies**.
2. Give a name for the properties set name (**Wall** for example).
3. Select "Beam" as type.
4. In "Mechanical properties" input the concrete properties: ρ , E and ν .
5. In "Geometrical properties":
 - Select the section type
 - Input $b = 1$ m (1 m in horizontal depth) and $h = 0.8$ m.
6. Click on **Validate** and **Close**.




Material properties for the anchors

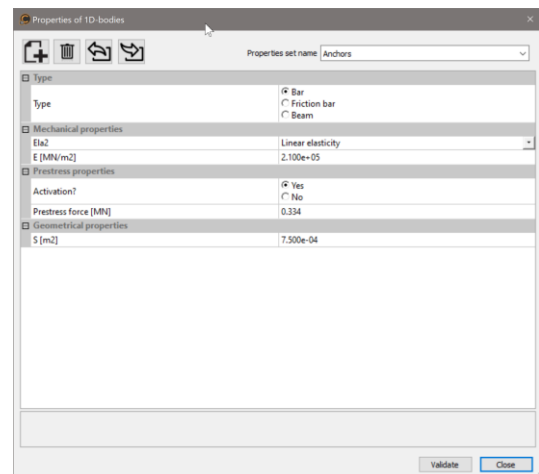
For supporting the wall during excavation, 2 layers of prestressed anchors are installed.

Again, as the model is built under plane strain conditions; we must adjust the sections and rigidity of the anchors so that it takes into account the spacing between the anchors in out-of-plane direction, as a ratio of anchors per meter. In the present exercise, the anchors (E , S) are set every 2 m (L), therefore $h(\text{equivalent}) = S/L$; and to keep the equivalence $E \times A = E(\text{equivalent}) \times S(\text{equivalent})$

- $E(\text{equivalent}) = E$
- $S(\text{equivalent}) = b \times h_e$ with $b = 1$ m (plane strain hypothesis)



Under the same hypothesis, the prestress force is F/L . This prestress is activated when the anchors are setup, before the excavation of next lower level.

1. Click on  **Properties for 1D bodies**.
2. Give a name for the properties set name (**Anchor** for example).
3. Select "Bar" as type.
4. In "Mechanical properties" input the elasticity modulus: $E = 210\,000$ MPa
5. In "Prestress properties":
 - Activation = Yes
 - Force = 0,334 MN
6. In "Geometrical properties":
 - Input $S = 7,5 \cdot 10^{-4}$ m²
7. Click on **Validate** and **Close**.

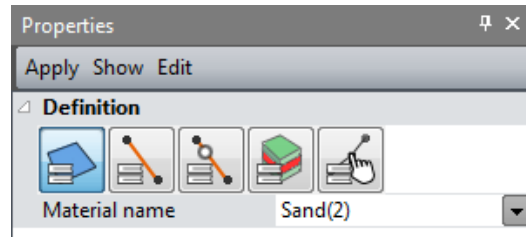




Assignment of data sets:

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 grid is displayed. Click on  **Properties for surface bodies**.

- Select the first layer surface bodies on the model window and the set of parameters **Sand #1** in the list. **Apply**.
- Repeat the operation for other surface bodies of the second layer with **Sand #2** properties set.




- Click on  **Properties for 1D bodies**.
 - Select the wall on the model, select the **Wall** set of properties in the grid. **Apply**.
 - Select the anchors on the model, select the **Anchor** set of properties in the grid. **Apply**.
- Click on  **Properties for interface bodies**.
 - Select the interface bodies of the **Wall** in **Sand #1**. Select the set of properties **Wall/Sand #1** in the grid. **Apply**.
 - Select the interface bodies of the **Wall** in **Sand #2**. Select the set of properties **Wall/Sand #2** in the grid. **Apply**.



Another method for assigning properties is possible with the tool  **Assign/view body properties**.

Active/inactive elements

- Select the following bodies : **Wall, Anchor 1** and **Anchor 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).

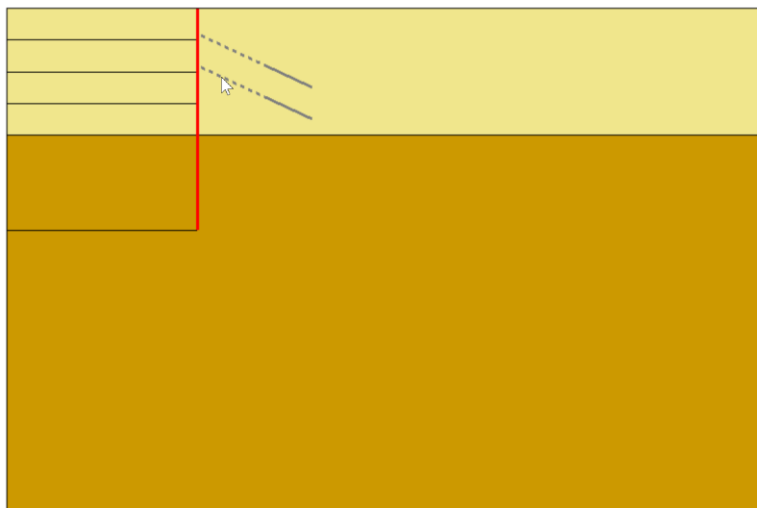



Figure 4: Display of the activated bodies

Boundary conditions:

1. Default name of the boundary conditions set, BCSet1, can be edited using: [F2] key + *Name*+ [Enter] key. Default name of the boundary conditions, BCSet1, set can be edited using the [F2] key. Rename it to **Standard fixities**.
2. Activate the **BOUNDARY CONDITIONS** tab.
3. On the toolbar, activate  to define classic side and bottom supports.
4. **Apply**. Supports are automatically affected to the limits of the model.



Several boundary conditions sets are possible. Right click on **Boundary conditions**, in the data tree, to generate another one.

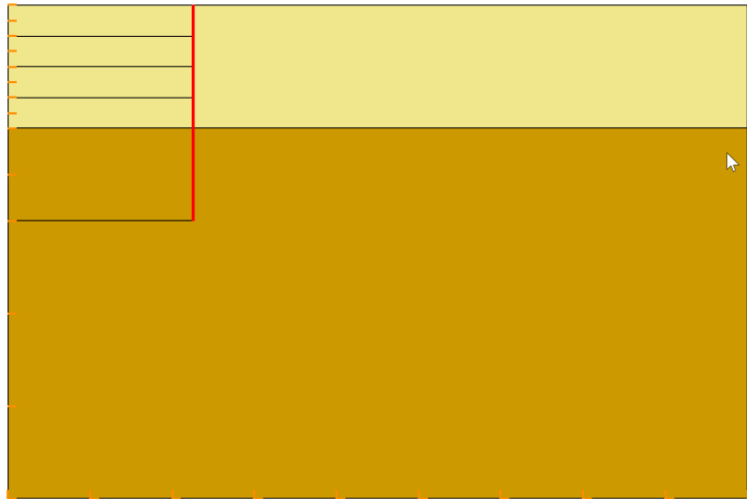




Figure 5: Display of the boundary conditions

Loading:

The principle of the effective stress field initialization consists of 2 steps:

- definition of the water table position,
- definition of the dry and saturated unit weights of the soil masses.

1. Activate the **LOADS** tab.
2. On the toolbar, activate  **WTB - isotropic pressure**.
 - Water head input: Constant
 - Value: - 3 m
 - Water unit weight: 0.010 MN/m³ (10 kN/m³) (default value)
 - **Validate**
3. On the toolbar, activate  **WTB – unit weights**.
 - Unit weight: 0.019 MN/m³ (20 kN/m³)
 - Saturated unit weight: 0.020 MN/m³ (20 kN/m³)
 - Select all the surface bodies. **Apply**.

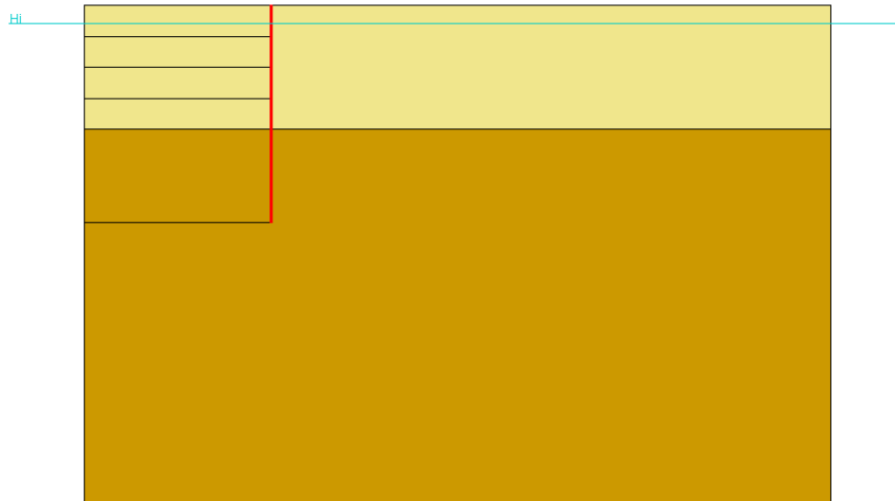
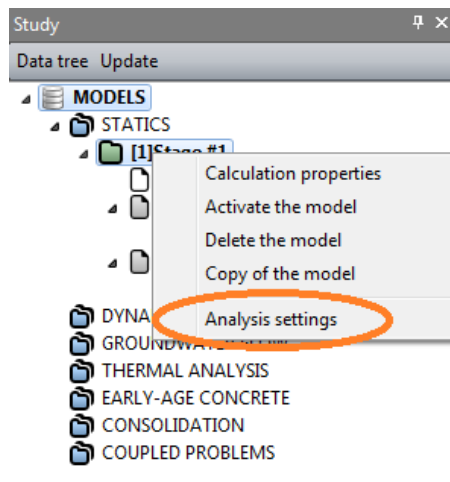


Figure 6: Display of the WTB

Analysis settings:

1. Activate the **ANALYSIS** tab.

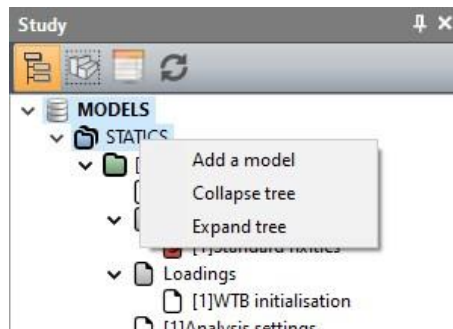


2. In the **STUDY** tree, right click on the calculation title. In the displayed list, select **Analysis settings**.
3. In the **General parameter** section, enter the following values:
 - Iteration process:
 - Max number of increments: 2
 - Max number of iterations per increment: 500
 - Tolerance: 0,01
 - Solution method: 1 – Method of initial stresses
 - Algorithm type: Pardiso
 - Analysis type: Standard
 - Storage: activate storage of total and plastic strains.
4. **Validate.**

3.2. Stage #2: wall setup and WTB lowering at -5 m

In this step, we model the setup of the wall ("wished-in-place" hypothesis) and we lower the water level inside the excavation area from -3 m to -5 m.

Model definition:



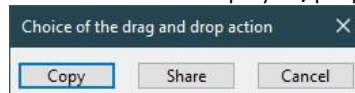
1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.

2. A toolbox is displayed for definition of this new model.

- Change the model name: **WIP + WTB lowering**.
- MCNL Solver
- Plane strain
- Select "Staged construction". Default order is set to 2.
- **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:

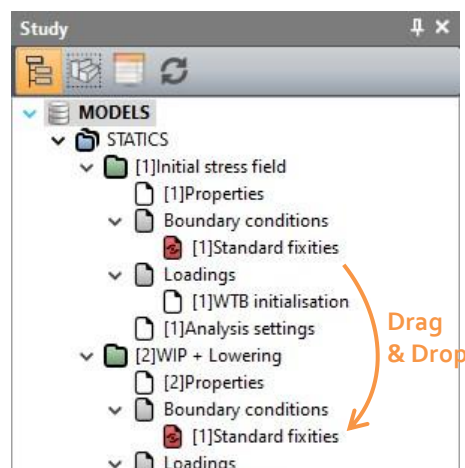


Select **Copy**.


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.



Activation of the wall and interface properties



1. Activate the **PROPERTIES** tab.
2. Select the two rows of anchors.
 - Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
3. Apply friction properties to the interfaces of the wall with **Sand #1** and **Sand #2**.

Boundary conditions:

There is no change in the boundary conditions.

Loading:

We generate the loading set for the WTB lowering.

1. In the Calculations tree, the default name of the loading set, LoadSet1, can be edited using: [F2] key + Name+ [Enter] key. Rename it **WTB lowering**.
2. On the toolbar, activate  **WTB - isotropic pressure**.
 - Define the Initial hydrostatic field:
 - o Water head input: Constant
 - o Value: -3 m
 - Define the Actual hydrostatic field:
 - o Water head input: Variable
 - o Number of points: 4
 - o Points: (0 ; -5) (30 ; -5) (30 ; -3) (120 ; -3)
 - Water unit weight: 0.010 MN/m³ (default value)
 - **Validate**
3. On the toolbar, activate  **WTB – unit weights**.
 - Unit weight: 0.019 MN/m³ (20 kN/m³)
 - Saturated unit weight: 0.020 MN/m³ (20 kN/m³)
 - Select all the surface bodies. **Apply**.

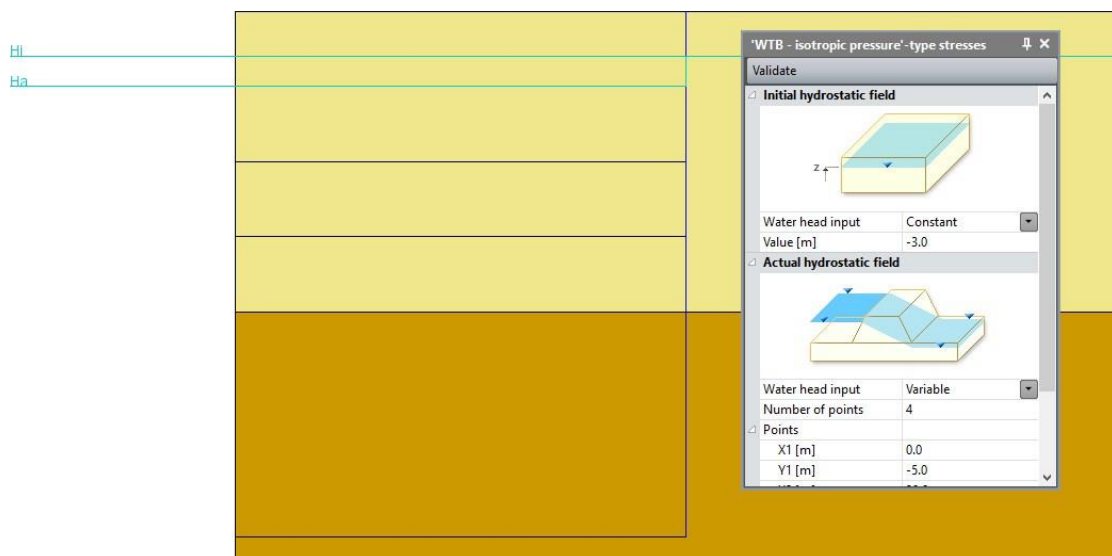


Figure 7: WTB lowering inside the excavation

Analysis settings:

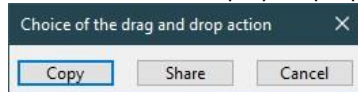
The displacements are reset so that the settlements associated to the initial stress field are not taken into account.

1. Activate the **ANALYSIS** tab.
2. In the STUDY tree, right click on the calculation title. In the displayed list, select **Analysis settings**.
3. In the **General parameter** section, tick "Displacement initialisation".
4. All other parameters are unchanged.
5. **Validate**.

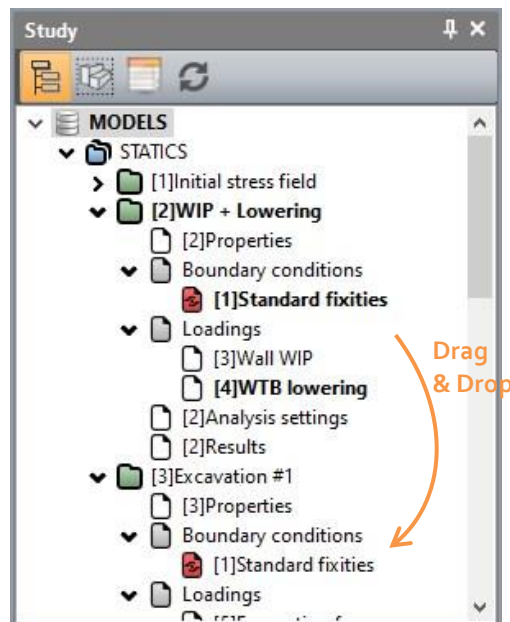
3.3. Stage #3: excavation at -5 m

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.
 - Change the default model name: **Excavation #1**.
 - Select "Staged construction". Default order is set to 3.
 - **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 two rows of anchors and the excavated soil.
 - Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**. The body is now marked with default neutral colour.

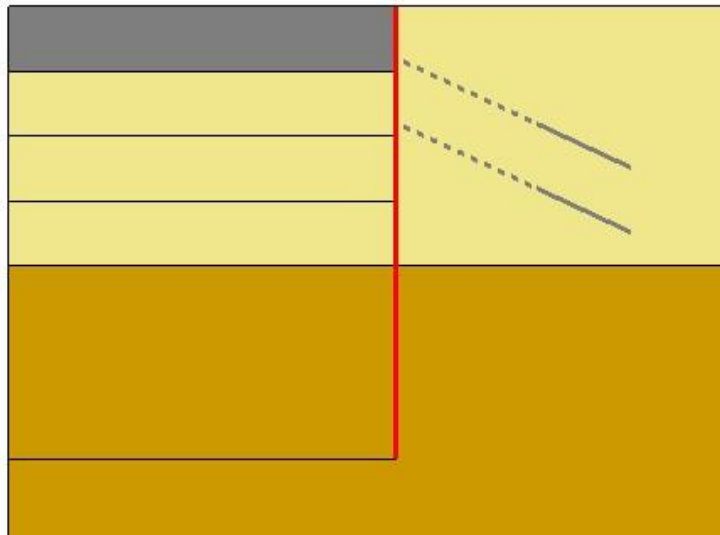



Figure 8: Status of the model when body is set inactive at stage #3

Boundary conditions:

There is no change in the boundary conditions.

Loading:

1. In the Calculations tree, the default name of the loading set, LoadSet1, can be edited using: [F2] key + *Name* + [Enter] key. Rename it **Excavation forces #1**.
2. Activate the **LOADS** tab.
3. On the toolbar, activate  **Excavation forces**.
 - Activate.
 - Set the "Stress release coefficient" to 1.
 - Set the "Stress field before excavation": here, the previous stage, i.e. **WIP + lowering**.
 - **Apply**.

If user wants to visualize where the excavation forces are located, click on **Show**. This feature displays the boundaries where we apply the stress-release due to the removal of soil (bodies that were inactivated in step PROPERTIES).

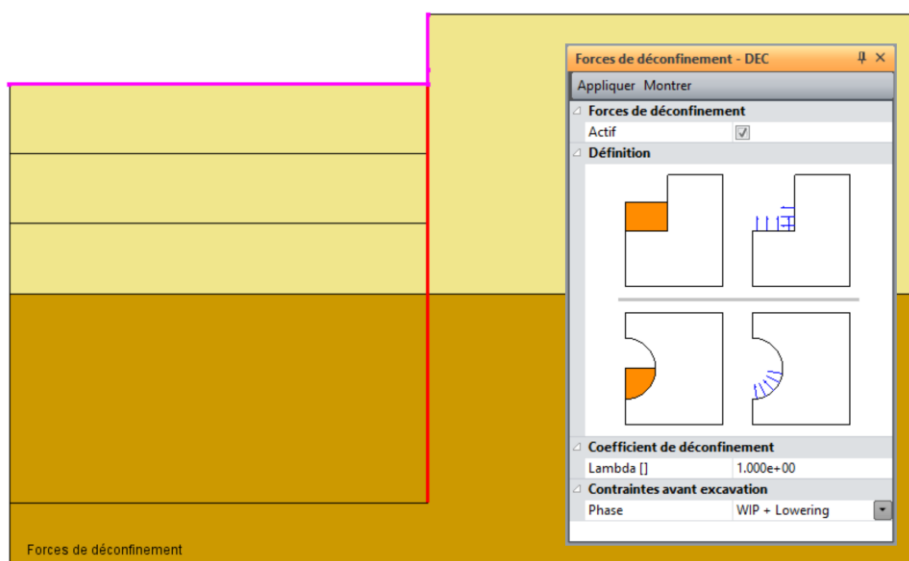
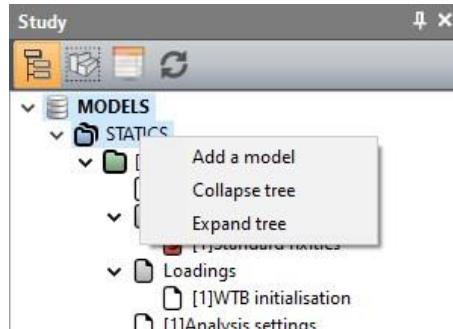


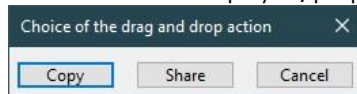
Figure 9: Definition of the excavation forces at -5 m

3.4. Stage #4: setup of the first layer of anchors

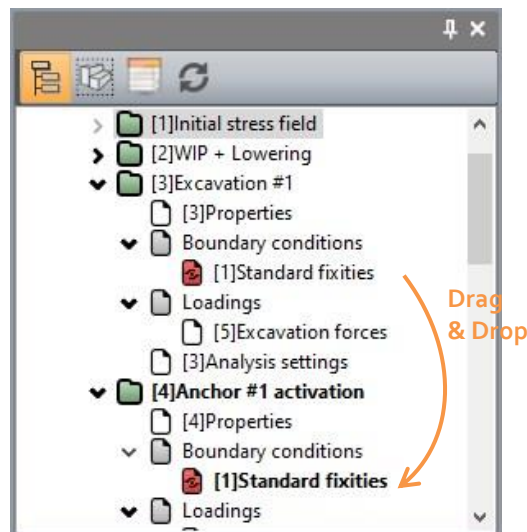
Model definition:




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.
 - Change the default model name: **Activation of anchor #1**.
 - Select "Staged construction". Default order is set to 4.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of Stage #3 to Stage #4.
 - Select "Properties" of Stage #3,
 - Place it on "Properties" of Stage #4
 - A toolbox is now displayed, proposing several options:


4. Using "drag and drop" action, copy the Boundary conditions set of Stage #3 to Stage #4.
 - Select the **Standard fixities** set of Stage #3,
 - Place it on "BCSet1" of Stage #4
 - 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 2nd row of anchors and the excavated surface body.
 - Click on  **Activate/deactivate bodies**.
 - In the grid, tick "Inactive".
 - **Apply**. The body is now marked with the neutral colour.

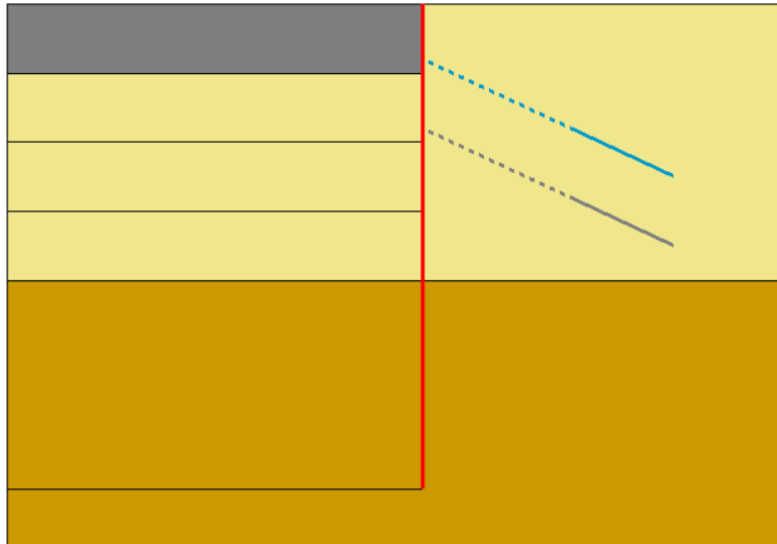


Figure 10: Status of the model when the 1st anchor is set active at stage #4

Boundary conditions:

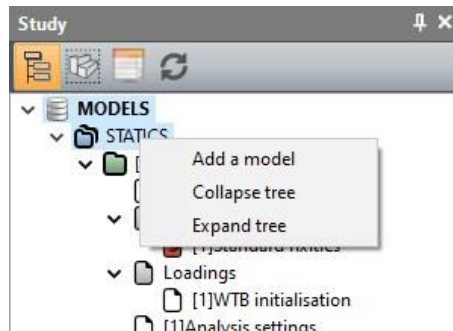
No additional loading set has to be defined. In this stage, the prestress forces in the anchor are activated.

Analysis settings:

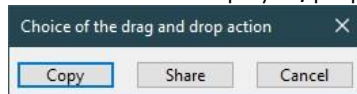
There is no change in the analysis settings.

3.5. Stage #5: WTB lowering at -10 m

Model definition:



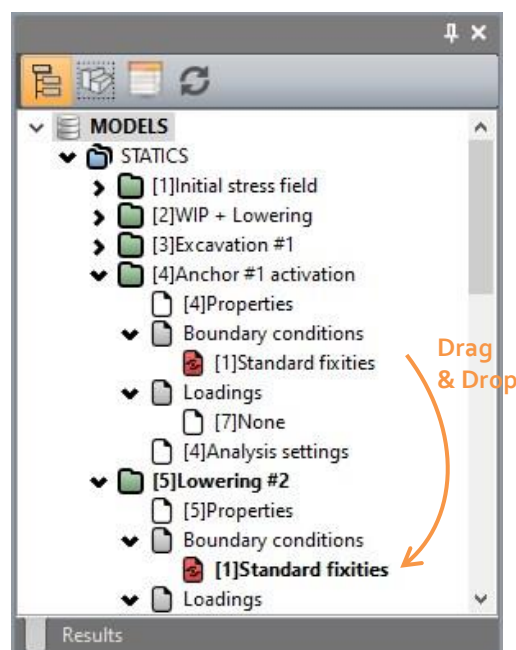
1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.
2. A toolbox is displayed for definition of this new model.
 - Change the default model name: **Lowering #2**.
 - Select "Staged construction". Default order is set to 5.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of Stage #4 to Stage #5.
 - Select "Properties" of Stage #4,
 - Place it on "Properties" of Stage #5
 - A toolbox is now displayed, proposing several options:



Select **Copy**.
4. Using "drag and drop" action, copy the Boundary conditions set of Stage #4 to Stage #5.
 - Select the **Standard fixities** set of Stage #4,
 - Place it on "BCSet1" of Stage #5,
 - 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.





Boundary conditions:

No additional loading set has to be defined. In this stage, the prestress forces in the anchor are activated.

Loading:

As the wall is activated, we apply its weight.

1. In the Calculations tree, the default name of the loading set, LoadSet1, can be edited using: [F2] key + *Name* + [Enter] key. Rename it **Lowering #2**.
2. Activate the **LOADS** tab.
3. Right-click on **Loadings**. Select **Add loading set**. Name it **WTB lowering**.
4. On the toolbar, activate  **WTB - isotropic pressure**.
 - Define the Initial hydrostatic field:
 - o Water head input: Variable
 - o Number of points: 4
 - o Points: (0 ; -5) (30 ; -5) (30 ; -3) (150 ; -3)
 - Define the Actual hydrostatic field:
 - o Water head input: Variable
 - o Number of points: 4
 - o Points: (0 ; -10) (30 ; -10) (30 ; -3) (150 ; -3)
 - Water unit weight: 0.010 MN/m³ (default value)
 - **Validate**
5. On the toolbar, activate  **WTB – unit weights**.
 - Unit weight: 0.019 MN/m³ (20 kN/m³)
 - Saturated unit weight: 0.020 MN/m³ (20 kN/m³)
 - Select all the surface bodies. **Apply**.

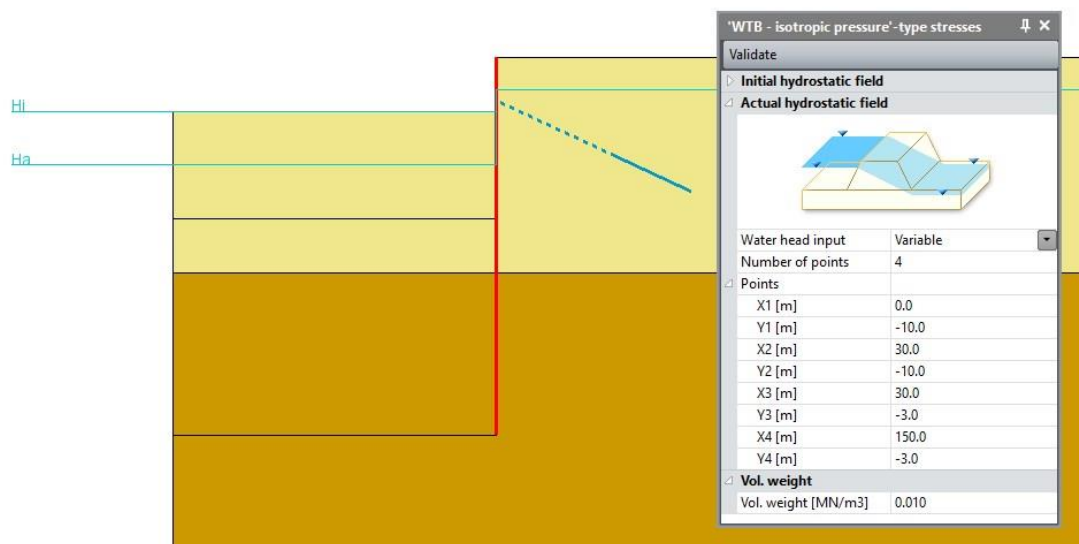


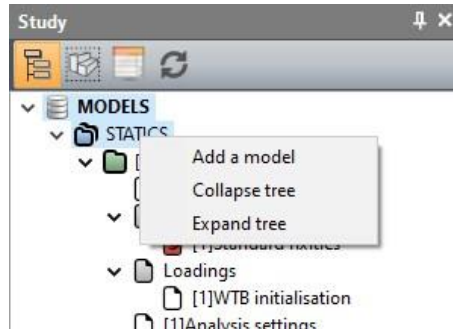
Figure 11: WTB lowering at -10 m inside the excavation

Analysis settings:

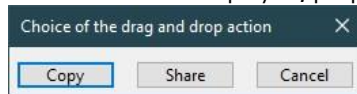
There is no change in the analysis settings.

3.6. Stage #6: excavation at -10 m

Model definition:

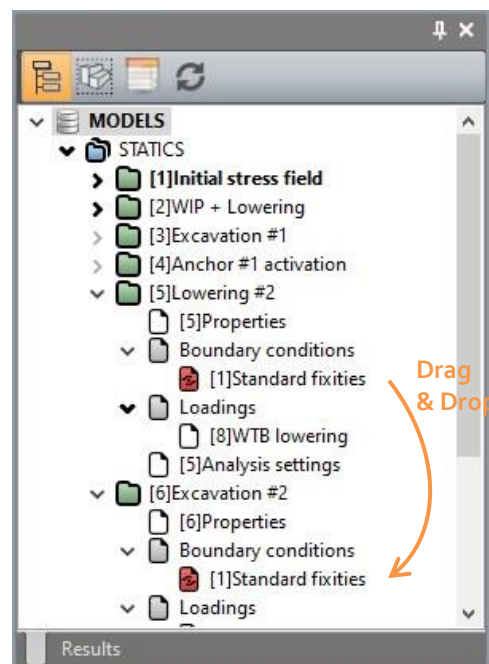


1. In the STUDY tree, right click on STATICS. In the displayed list, select **Add a model**.
2. A toolbox is displayed for definition of this new model.
 - Change the default model name: **Excavation #2**.
 - Select "Staged construction". Default order is set to 6.
 - **Validate**.
3. Using "drag and drop" action, copy the Properties set of Stage #5 to Stage #6.
 - Select "Properties" of Stage #5,
 - Place it on "Properties" of Stage #6
 - A toolbox is now displayed, proposing several options:




 - Select **Copy**.
4. Using "drag and drop" action, copy the Boundary conditions set of Stage #5 to Stage #6.
 - Select the **Standard fixities** set of Stage #5,
 - Place it on "BCSet1" of Stage #6
 - 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 2nd row of anchors and the excavated surface bodies.
 - Click on  Activate/deactivate bodies.
 - In the grid, tick "Inactive".
 - **Apply**. The body is now marked with default neutral colour.

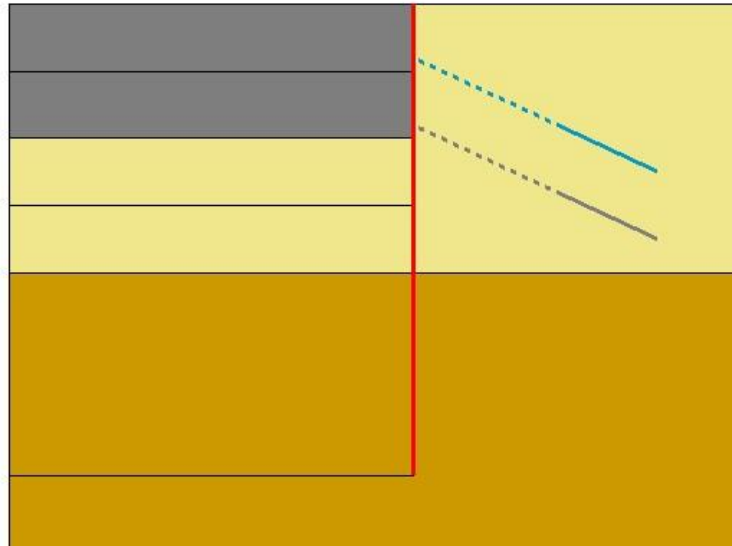


Figure 12: Status of the model when body is set inactive at stage #6

Boundary conditions:

There is no change in the boundary conditions.

Loading:

Repeat operations of previous load set **Excavation forces #1** and generate load set **Excavation forces #2**. With "Stress field before excavation" being here **Lowering #2**, the previous stage,

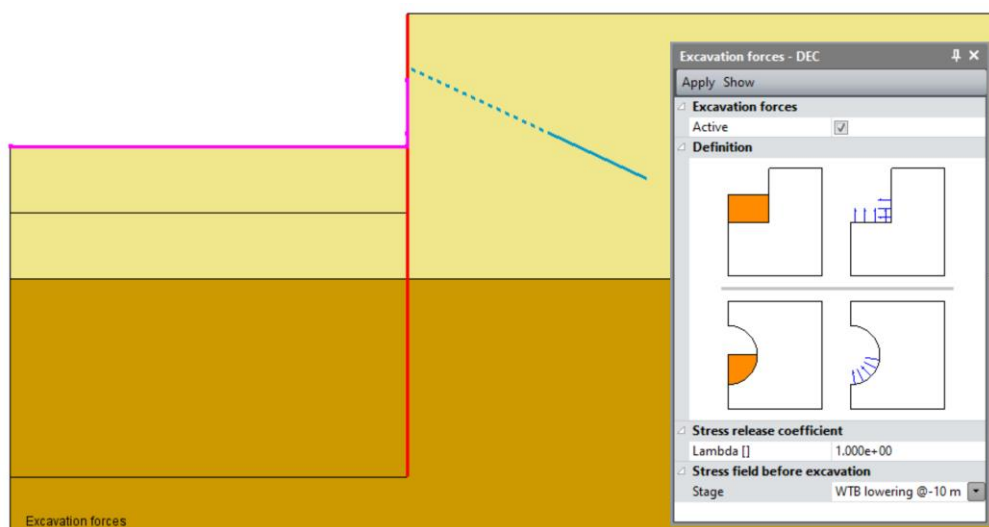




Figure 13: Definition of the excavation forces at -10 m

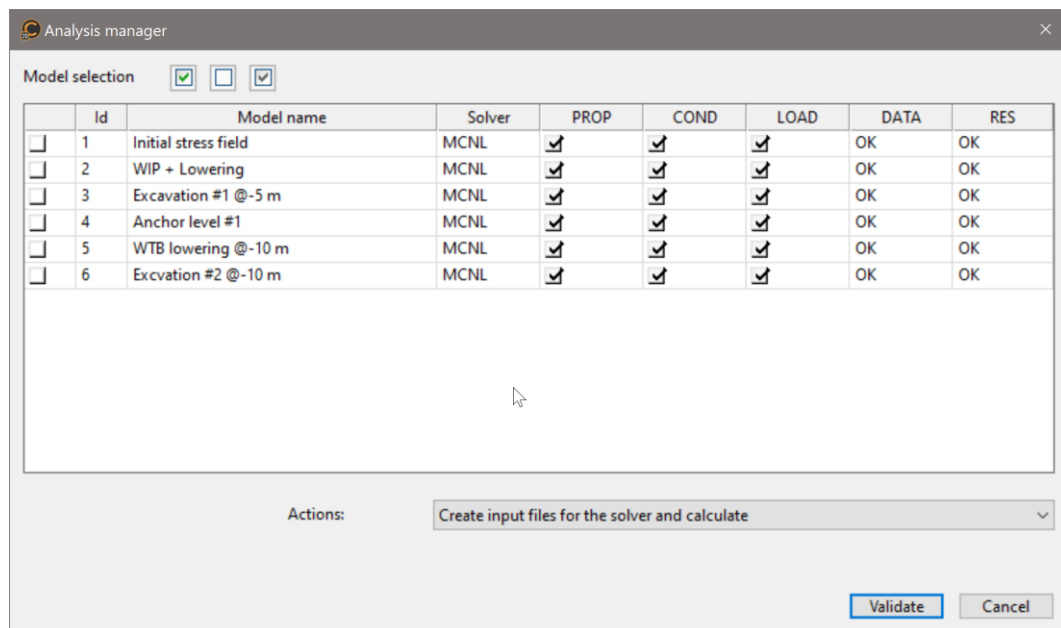
4. SOLVE

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 stages using .
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".






The program detects if the models are ready for calculation. All steps should be validated with a tick mark.

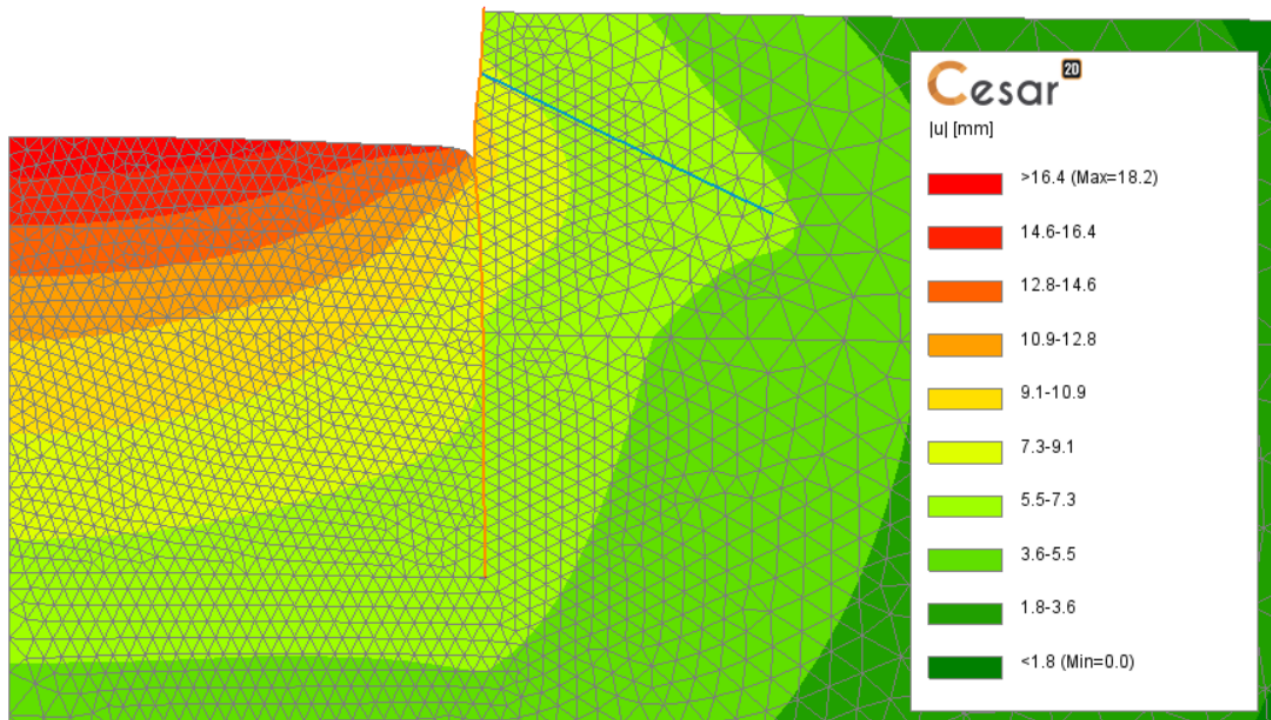


All the messages during the analysis will be shown in an **Output Window**. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The result is saved as a binary file (*.RSV4) in the temporary folder (.../TMP/), defined during setup. The detailed analysis information is also saved in a text file (*.LIST).


5. RESULTS

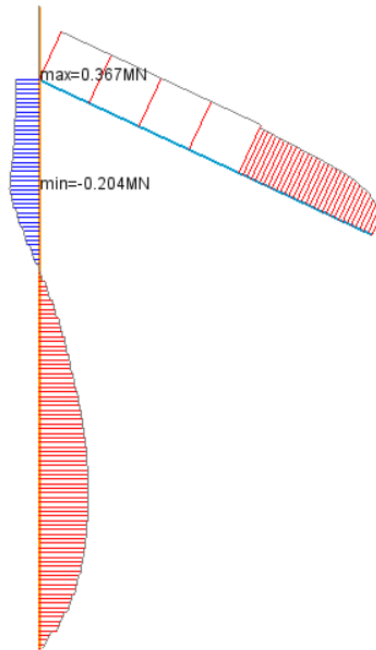
Typical results obtained for the 5th increment of the last phase are provided in the following figures.

1. Activate the **RESULTS** tab.
2. Click on  **Result Types**
 - Select **Deformed** as Mesh,
 - Tick **Contour plots**, in the list select **|u|**, total displacement,
 - 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 scalar type,
 - Set that a value of **10 mm** is represented by **1 m**,
 - **Validate**.






User can display forces inside the 1D-bodies. It is advised to display on undeformed bodies.

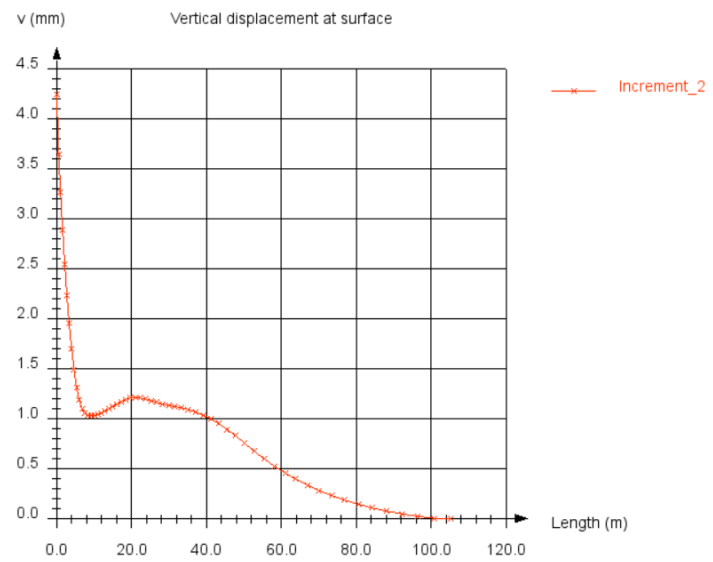
1. Click on  **Result Types**
 - Select **Deformed** as Mesh,
 - Tick **Active**, for the **1D-bodies results**,
 - Select **N** (normal forces) as Parameter.
 - Click on **Apply**.



To get the displacement behind the wall at the surface, 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 surface of the soil.
3. Click on  **.Lines set.**
 - Give a name to this line set, **Surface** for example.
 - **Add.**
4. Activate  **Charts for a line set.**
 - Select **v** as parameter,
 - Select **Surface** 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.**



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