Cesar

Release note

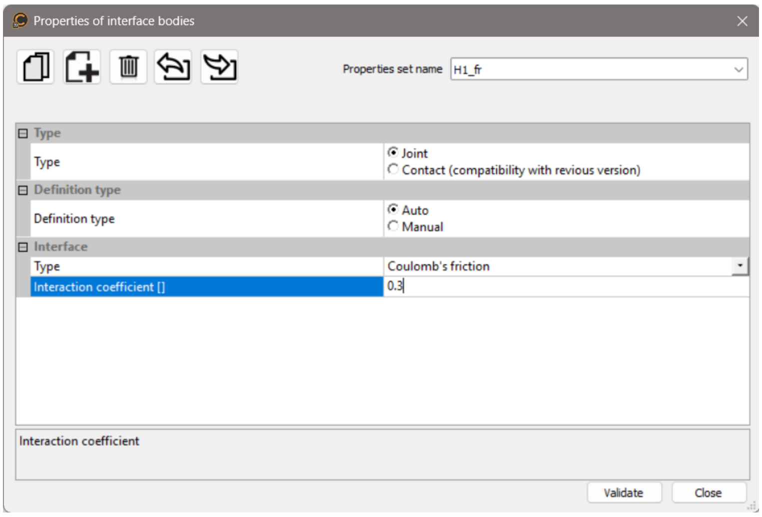
Version 2025.0

Table of contents

1. Material properties

1.1. Automatic definition of interface properties

Under PROPERTIES, access the ***Properties for interface bodies***.



When the user integrates interfaces in his model (1D or 3D), he has to define:
- the type of contact: bonding, sliding or friction
- the values of parameters associated with the type of contact. In the case of friction: E_c, R_t, c_c, φ_c.

If the interface crosses several materials (e.g. a pile founded in a multi-layered soil), the definition must be adjusted according to the properties of the materials (soils).

In common cases (geotechnical engineering in particular), interface properties can easily be deduced from the properties of the materials in contact.
- E_c, stiffness coefficient of the fictitious material. Its value equals 100 times the lowest modulus of the materials in contact,
- R_t, tensile limit. Its value is generally taken at a very high level to prevent delamination,
- c_c, cohesion. Its value is generally taken null if we want to link directly the normal stress to the shear stress. Else, c_c can be considered as the unit lateral coefficient q_s.
- φ_c, friction angle. Its value generally defines the relation between the maximal tangential stress τ and the normal stress σ_n: $\tau = \tan \varphi_c \cdot \sigma_n$

The aim of the “Auto” option is to provide the user with an "automated" definition of interface properties. The input is simplified as the user will now define
- The type of contact: bonding, sliding or Coulomb’s friction.
- In the case of “coulomb’s friction” selection, the value of an interaction coefficient so that:

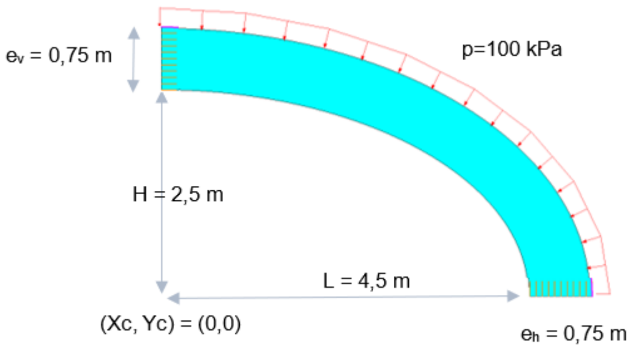
$$\tau = coef\_int \cdot \sigma_n$$

From this identification, we establish the following properties:
- $E_c = 100 \times \min(E1, E2)$
- $R_t = 10^6$ MPa
- $c_c = 0$
- $\varphi_c = \tan^{-1}(coef\_int)$

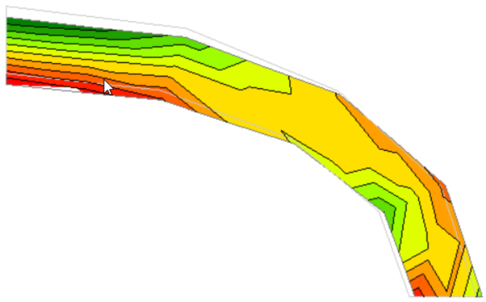
1.2. New elastic models (IELAS=18/19)

The software offers a number of elastic constitutive models for solid bodies, but the proposed range does not include a general anisotropic model.
This lack is corrected, in a particular framework, which aims to exploit the calculations of modules by homogenization for periodic materials.
We therefore add two values of the IELAS indicator (IELAS = 18 and 19), but new variants could be introduced later depending on the needs.

As an illustration, the model can be used for masonry. A vault under pressure is presented below. The user provides homogenized moduli (brick + mortar) and the geometry of the mean ellipsis that defines the orientation of the masonry blocks.



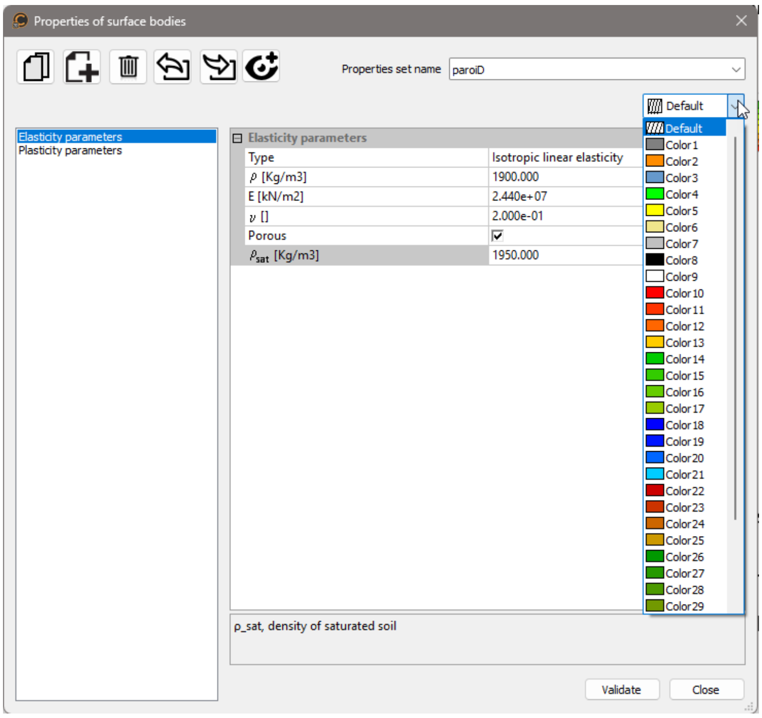
Geometry (half-vault)

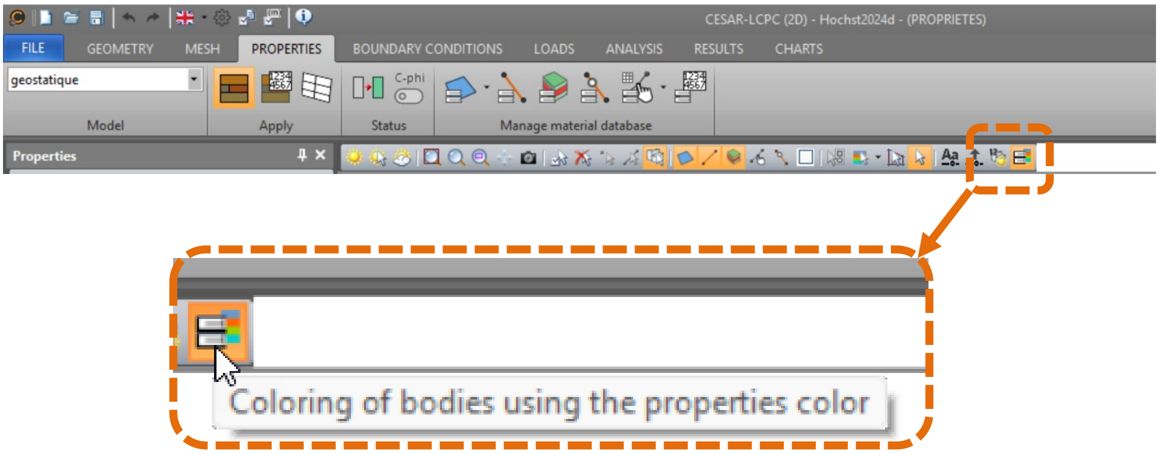


Stresses σ_{xx}

1.3. Associate a colour to a property set

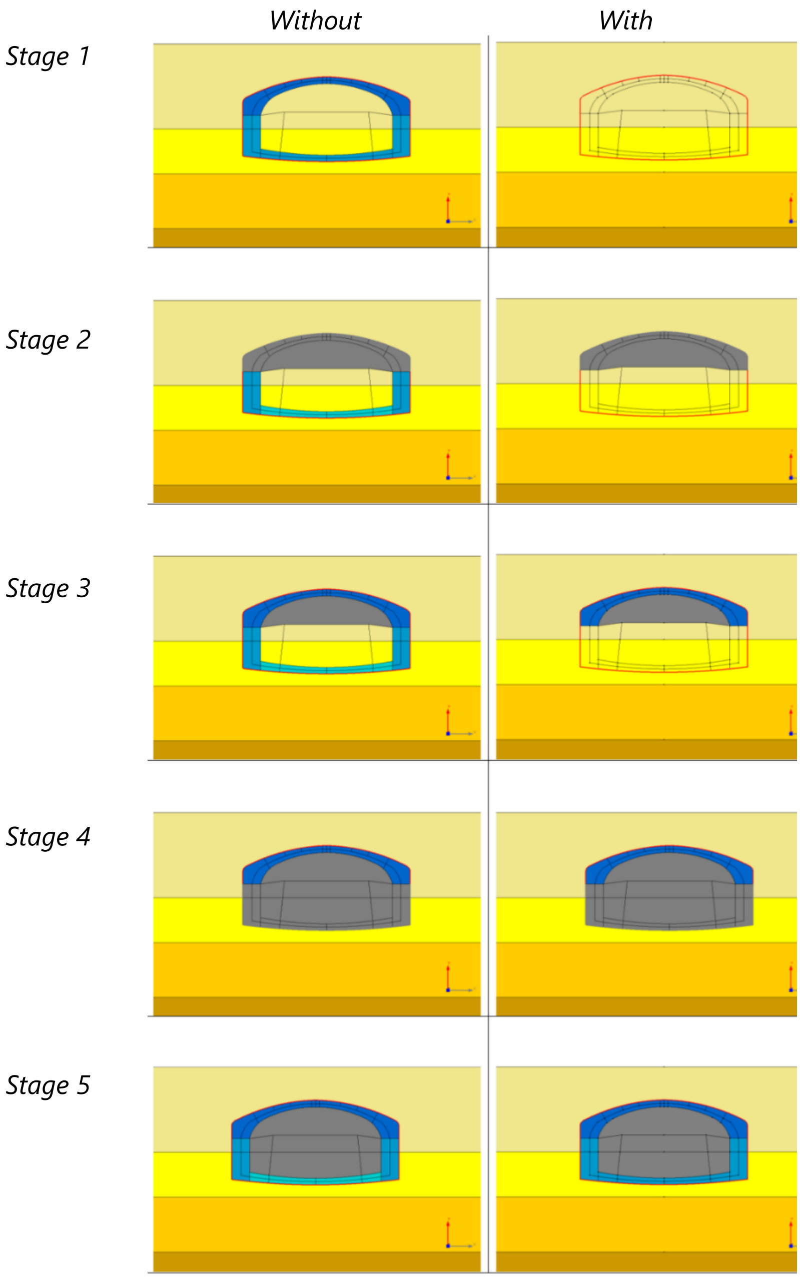
In the process of "stage construction" process, the user is assisted by assigning a colour per property set (surface and 1D bodies in 2D; surface, volume and 1D bodies in 3D).
The “colour” option has been added in the property definition toolbox. The selected colour is associated with the name of the property set.
The option is then activated by an “On/Off” button on the display toolbar.





This allows user to visually check the assignment of properties but also to follow the evolution of the stages in a more intuitive way.

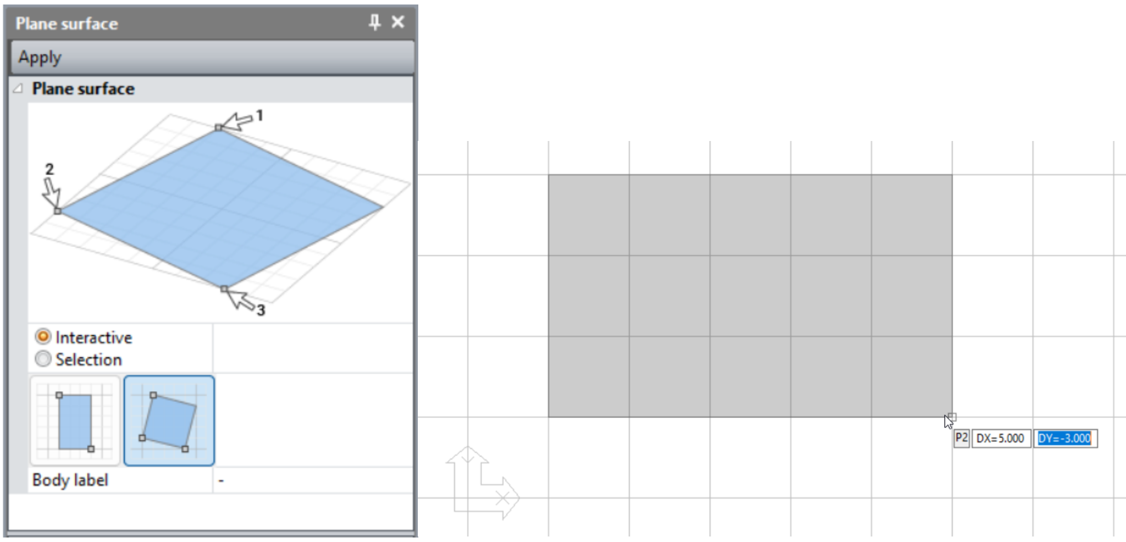
The example below illustrates the evolution of the stages with or without activation of the feature.



2. Geometry edition

2.1. Interactive rectangle – surface

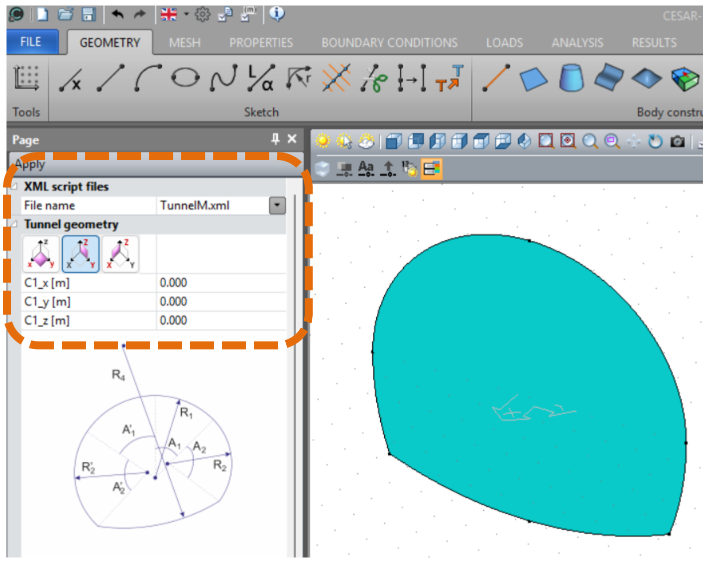
In GEOMETRY, use the ***Plane surface*** tool.
This tool is available in 2D and 3D to facilitate the generation of rectangular surfaces (e.g. 2D ground layers or 3D slabs).
The user can interactively define a rectangular surface by giving 2 or 3 points. These points are on the working grid or given by the interactive coordinates on the working space.
In 3D, a button makes it possible to toggle between the interactive generation and the usual "Selection" option: the user selects the contours (contained in the same plane) that close a surface.



2.2. Scripts > Tunnel sections

Use the ***Script*** and then select the "TunnelM.xml" file.

It is now possible to define coordinates of the centre of the tunnel section.
In 3D, it is also possible to specify in which plane the section will be drawn.

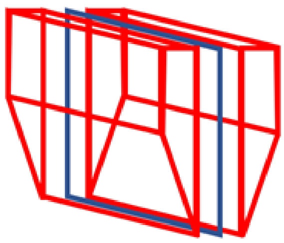


3. Mesh

3.1. Generation of interfaces at the end points

When a structural element (1D block or shell) immersed in a massif is surrounded by interface blocks (joints), we propose a better modelling of the ends.
The interface body generation process:
- detects the end of the interface
- at this end, merges the nodes of the "classic" interface. As a reminder, these elements have 6 nodes in 2D et 12 or 16 nodes in 3D.

Example for a 1D body



Example for a shell

4. Loads

4.1. New options for the definition of the water level in 2D and 3D

The definition of the initial water table (first stage of a staged calculation) or the variation of the water table between two successive stages of a staged calculation has been updated to be more flexible.

4.1.1. Method #1

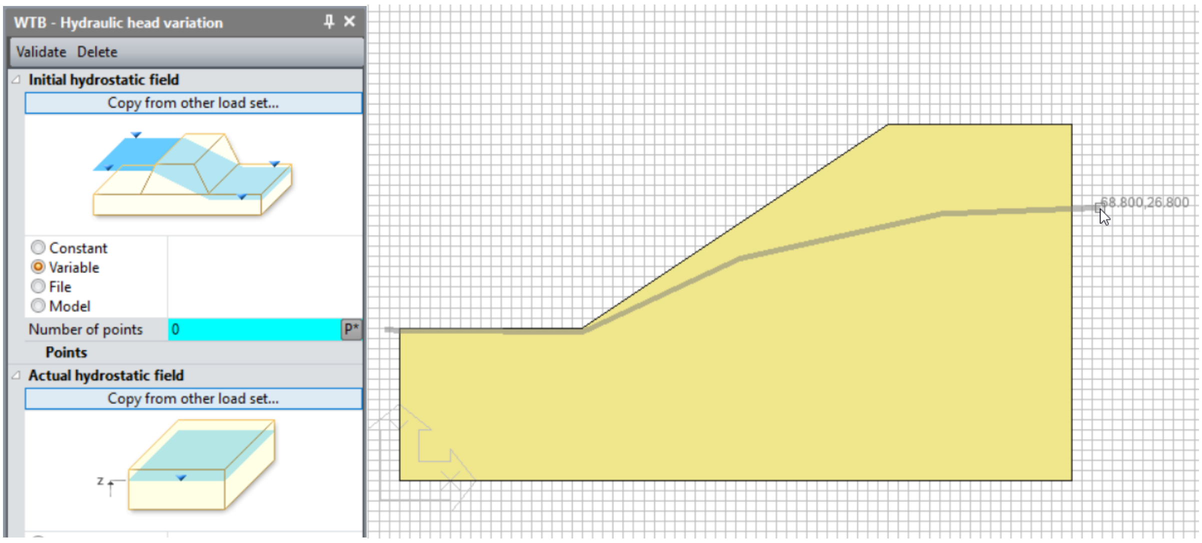
First, we have the possibility to define the free surface by successive mouse clicks on the model. In 3D, we first select one of the two vertical planes for this edition.

Step 1:

- Click "Variable"
- Click on [P*] to define the points by successive clicks. The user will then have an interactive representation of the coordinates of the points successively clicked on a grid.

Step 2:

- Validate. The number of points and their coordinates are then entered in the grid.

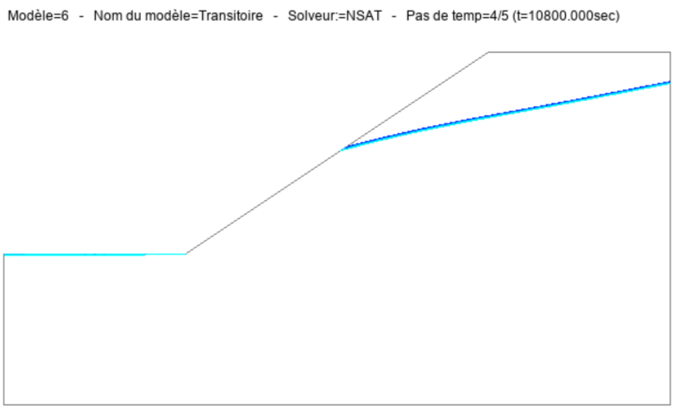


4.1.2. Method #2

The pressure field can also be recovered from a flow calculation (HYDROGEOLOGICAL modules) in steady state or transient mode.

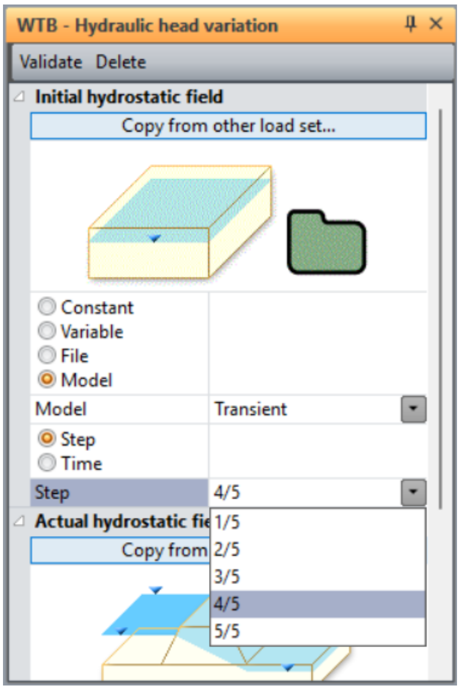
Step 1: hydrogeological calculation.

Example below, free surface (p=0) at the end of a transient NSAT calculation (5 time steps).



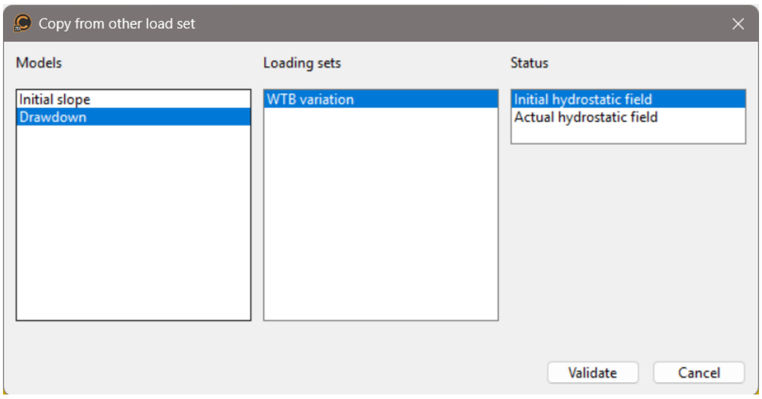
Step 2: In the mechanical model, use the "WTB – Water table variation" tool.
- Selection of the model
- Selection of the time step from those identified (if transient calculation)

Example: use of the previous NSAT model "Transient" and selection of time step 4/5.



4.1.3. Copy of the WTB loads

In order to ease the data input between stages, it is now possible to copy a water table state from a previous calculation.
In the editing window of the "WTB - Water table variation" tool, clicking on "Copy another load case" displays the window box below.
It lists:
- All models with at least one WTB type load,
- The list of WTB loads by model,
- The water table state retained from the selected WTB load: initial state or final state.



5. Calculation settings

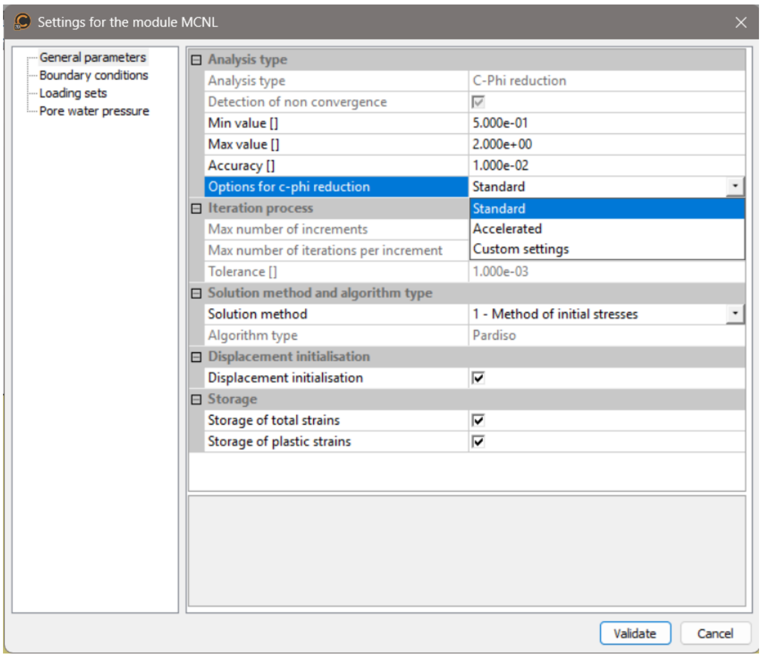
5.1. Accelerated c-phi reduction process

The “c-phi reduction” calculation procedure for calculating the safety coefficient on the stability of structures has been enhanced with a faster method.

The user is therefore offered 3 methods:

- Method 1 (standard, by default):
    - Resolution method: initial stresses
    - Number of iterations: 500
    - Precision: 0.001
- Method 2 (accelerated):
    - Resolution method: initial stresses + secant method
    - Number of iterations: 200
    - Precision: 0.001
- Method 3 (custom settings):
    - Resolution method: “free”
    - Number of iterations: “free”
    - Precision: “free”

The “accelerated” method will go more quickly to the result of the safety coefficient. However, it will be less precise for the representation of the failure mode.

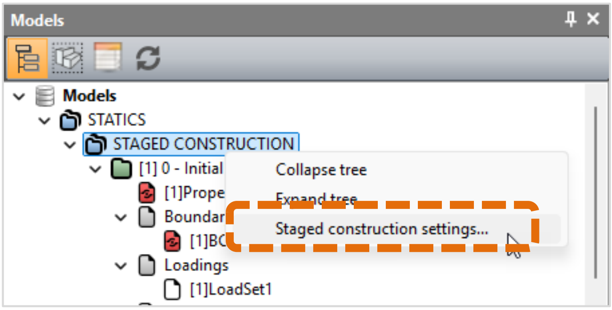


User can refer to recently published works of Bourgeois et Remaud:

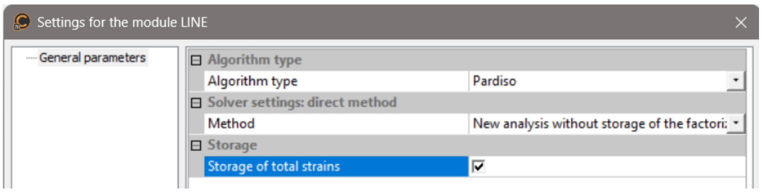
- Geotechnical structures stability: ULS calculations using the Finite Element Method, Proceedings ECSMGE 2024, Lisbonne.
- Une procédure numérique alternative pour les calculs de c-phi réduction dans CESAR-LCPC, Actes des JNGG 2024, Poitiers. (in French)

5.2. Edition of default settings for all construction stages

In the model tree, it is now possible to assign/modify the calculation settings for all the models defined with “Staged construction” type.
For example, this option is helpful to modify the precision of the iterative process or the number of iterations and affect them to all stages.

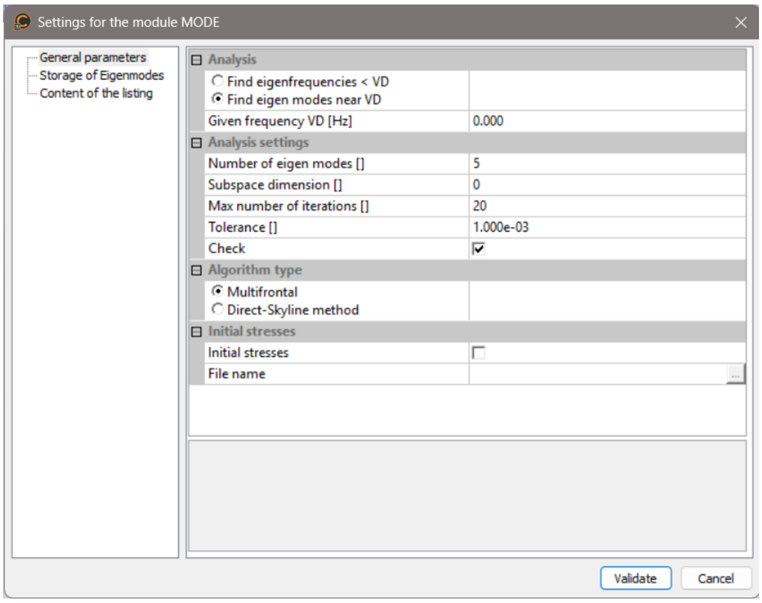


5.3. Storage of total strains for LINE / LINH



5.4. Multifrontal algorithm for MODE (eigen-values)

This option allows the faster computation of the eigen-values.

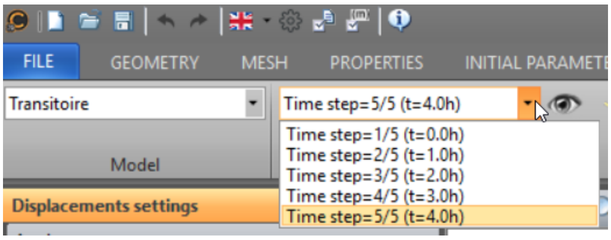


6. Results

6.1. Modification of the increment/time step

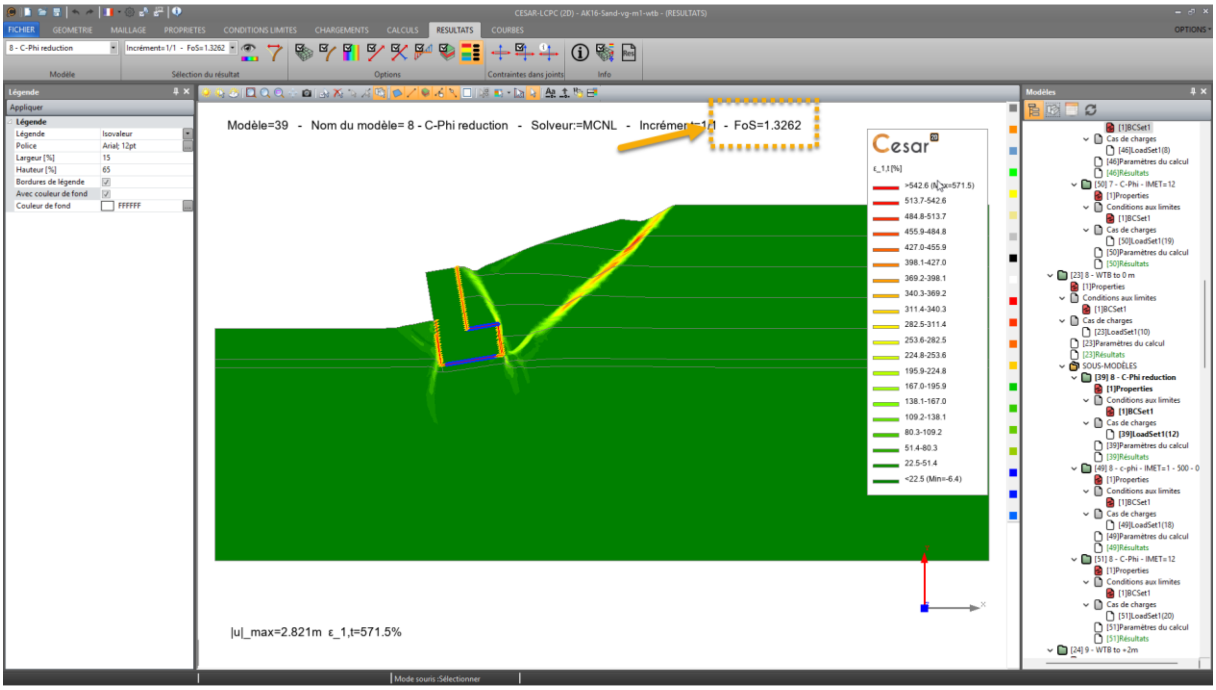
When the user accesses the RESULTS tab, the last increment (or time step) calculated is automatically displayed (if there are several increments or time steps calculated).

In the case of time steps, the time value is displayed.



6.2. Display of the safety factor

After a calculation of "c-phi reduction" (on shear strengths parameters) of "safety factor" (on loads) type, the calculated safety factor is displayed as a label on the working window.

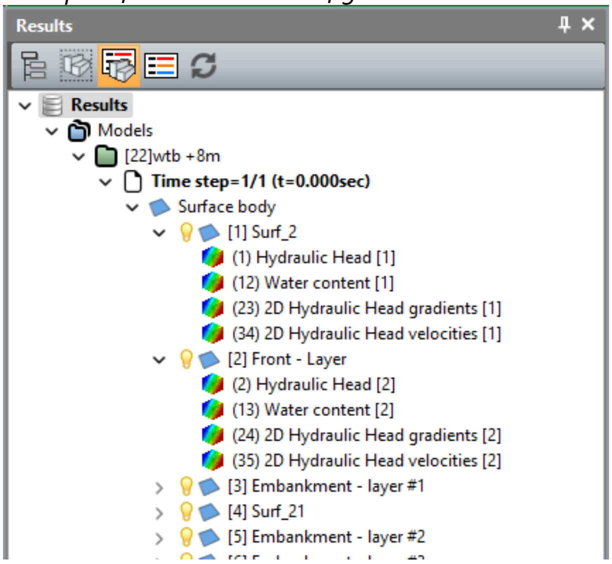


6.3. Modification of the tree of results

The results tree is modified so that it is easier to identify the type of result and the body in which we are looking for this result:

- User can identify the bodies by their name
- Results are sorted per body

Example of the new tree configuration



6.4. Additional scalars

`Volumic deformation`

When the deformation tensor is available in the results, the volumic deformation is proposed in the list of scalars to be displayed.

It is calculated as follows:

$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy}$	plane deformation
$\varepsilon_v = 2\,\varepsilon_{rr} + \varepsilon_{zz}$	axisymmetric
$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}$	plane stress
$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}$	3D

`Norm of the gradient of water head`

In NSAT (flow in unsaturated soils) and DTNL results.

$$|\Delta H| = \sqrt{\Delta H\_x^2+\Delta H\_y^2}$$

`Norm of the flow velocity`

In NSAT (flow in unsaturated soils) and DTNL results.

$$|vH| = \sqrt{vH\_x^2+vH\_y^2}$$

`Norm of the gradient of temperature`

$$|\Delta T°| = \sqrt{\Delta T°\_x^2+\Delta T°\_y^2}$$

`Norm of the temperature velocity`

$$|vT°| = \sqrt{v\,T°\_x^2+v\,T°\_y^2}$$