Release note

Version 2026.0

Table of contents

1. Material properties

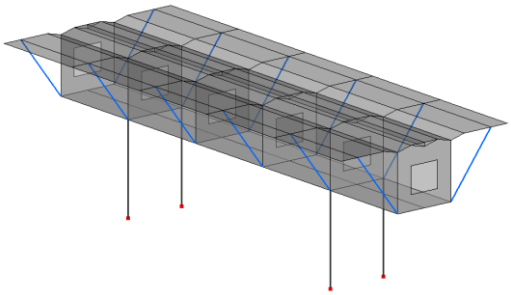
1.1. Non-linear supports

This new type of elements is limited to the support of 1D bodies, disconnected from any surface body (2D and 3D) or volume body (3D).
The support behaviour will be one of the followings:

1. Linear elasticity
2. Linear elasticity + perfect plasticity
3. Linear elasticity + perfect plasticity + brittle failure

Example in 2D

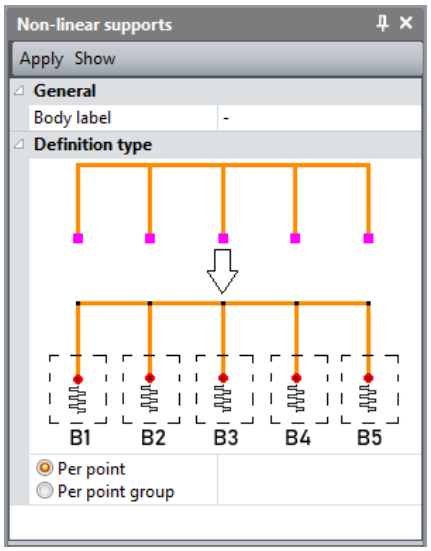
Example in 3D



Generation of non-linear supports

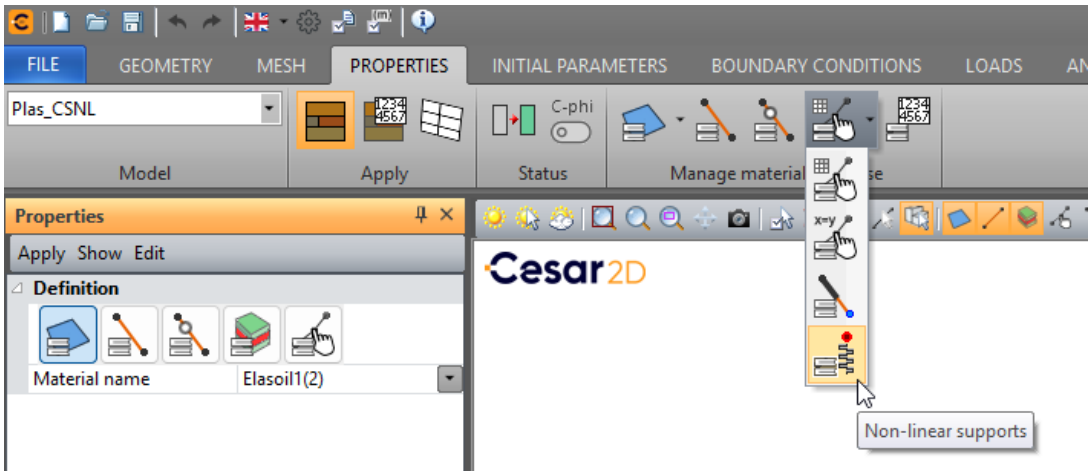
Step1: geometry

1. Under GEOMETRY, select one node, located at the end of a 1D-body.
2. Activate tool  **Nonlinear supports**. It displays the following edition grid.
   - Give a label to the body (facultative): if no label is given, a default one is given by the program.
   - Select one of the definition types:
     - Per point: each selected end-point will become an independent nonlinear support with its own set of properties
     - Per points group: all the end-points are merged in a group with a single set of properties.
3. Click "Apply".

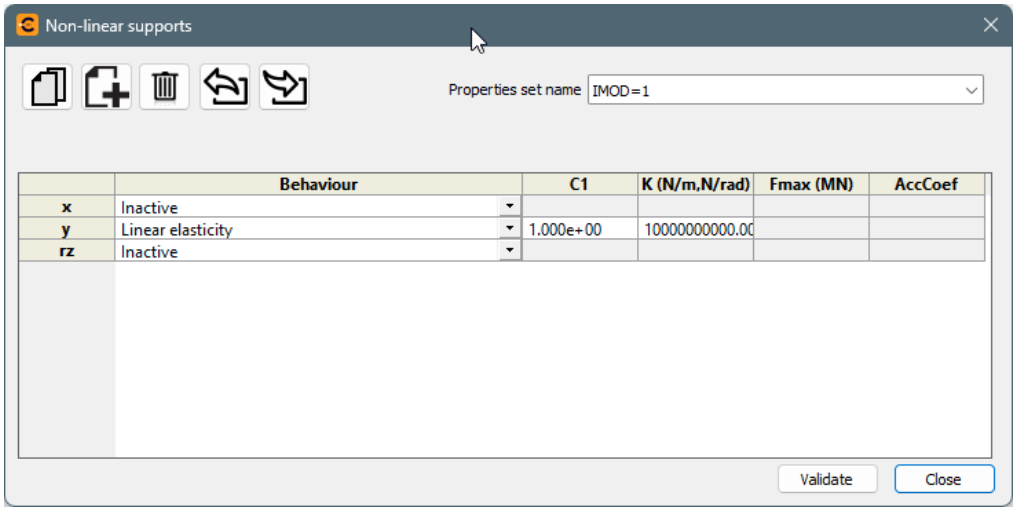


Step 2: properties

1. Under PROPERTIES, select "Supports"  in the list of "Properties of multinodal bodies". A new toolbox is displayed for the edition.
   For each of the degree of freedom (3 in 2D, 6 in 3D), define the type of behaviour:
   - Inactive
   - Linear elasticity
   - Linear elasticity + perfect plasticity
   - Linear elasticity + perfect plasticity + fragile
2. For each selection, edit the related parameters.
3. Click "Validate".



Example of non-linear support edition



1.2. Improved data edition for unsaturated porous materials

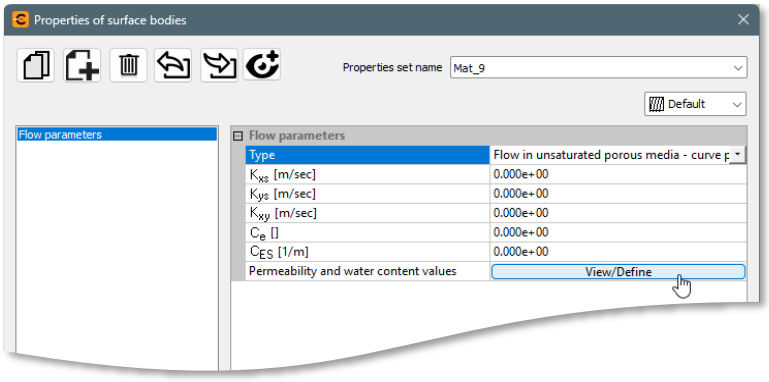
For unsaturated soils, CESAR offers various models for the flow in porous media.
The permeability and saturation laws included are:

- Preprogrammed curves based on works of Vauclin et Vachaud (1979),
- Van Genuchten and Gardner.

As the preprogrammed curves can also be edited point by point, the version 2026 allows a more comfortable edition.

Editing data for "Flow in unsaturated porous media – curve point by point"

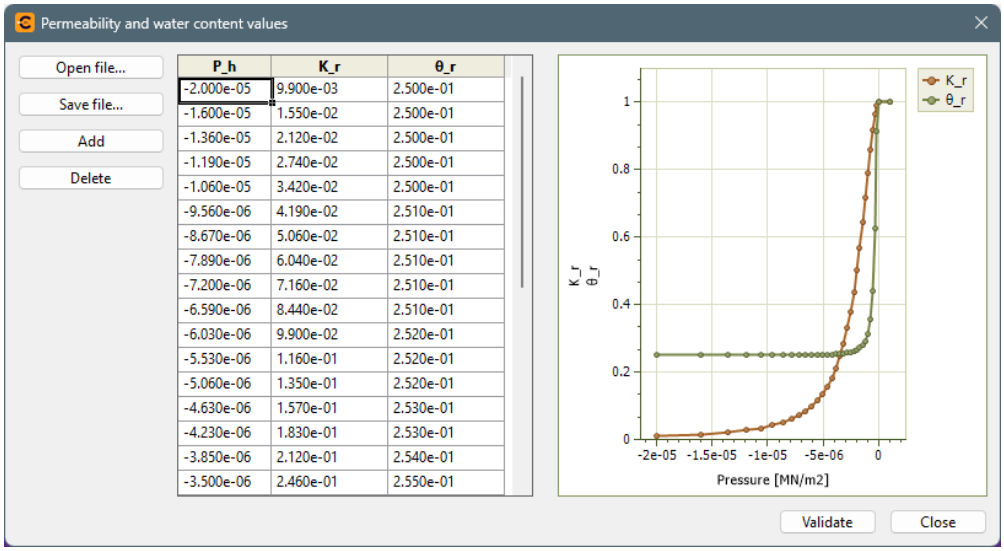
1. Under "PROPERTIES, select "Properties of surface bodies".
2. In the list of models for "Flow parameters", select "Flow in unsaturated porous media – curve point by point".
3. After the edition of the initial permeability coefficients, click on "View/Define" for the edition of the permeability and saturation curves.



Use the following features to edit the data:

- "Open file": import of data from a csv formatted file.
- "Save file": export of data in a csv formatted file.
- "Add": a line of data is added after the last one edited or after the last selected.
- "Delete": deletes the last line of data or the last selected one.

Example for display of permeability and saturation curves



Reference: Vauclin M., Khanji D., Vachaud D. (1979) Numerical Study of a Transient, Two-Dimensional Unsaturated-Saturated Water Table Recharge Problem, Water resources research, 15(5) : 1089-1101.

2. Loads

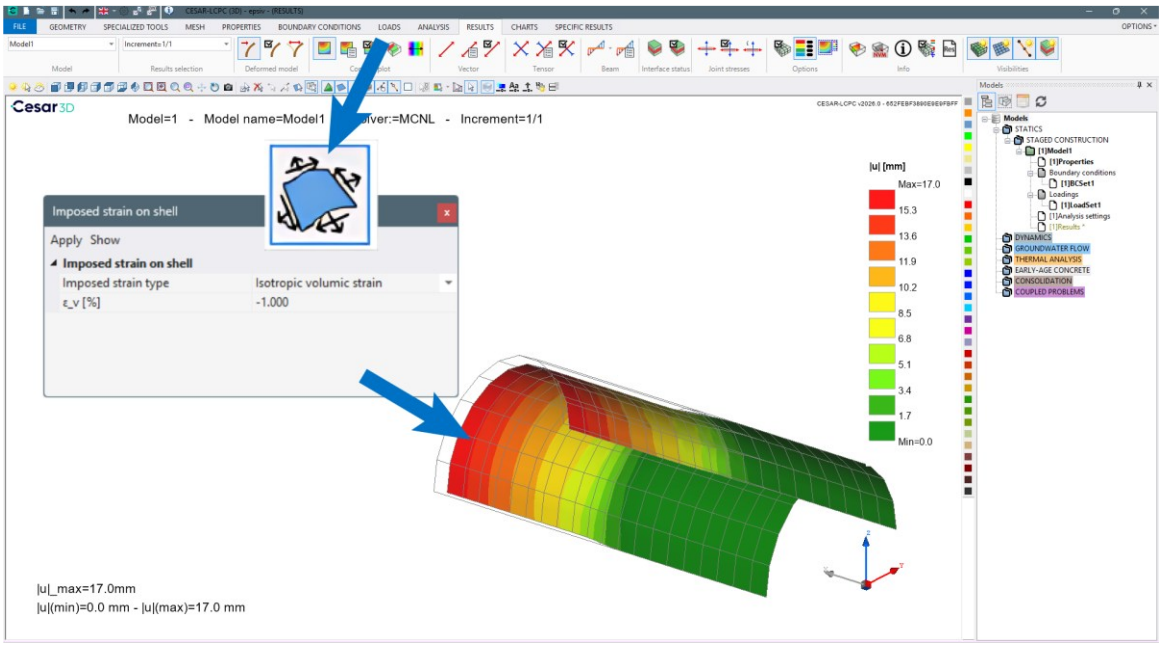
2.1. Imposed strain on shell - surface bodies (3D)

This new load allows to assign an imposed strain on a shell.
3 types of prescribed strain are proposed:

- Isotropic strain,
- Along an axis,
- Perpendicular to an axis.

A typical use of this feature is found in the domain of underground construction when one wants to model the shrinking of the lining due to the TBM advance.

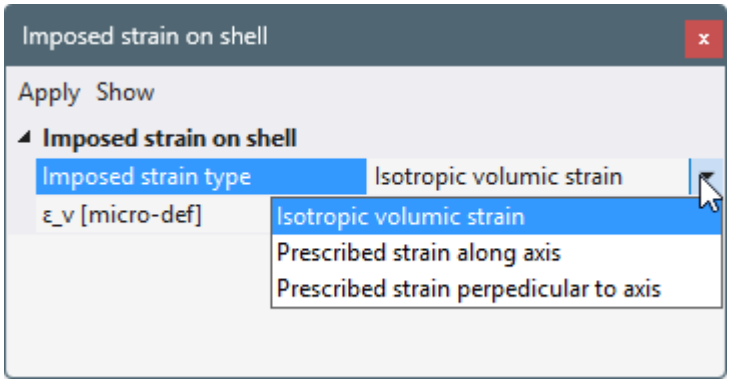
Example of imposed isotropic strain on a shell



Activate the tool  **Imposed strain on shell**

1. Select the type of imposed strain in the proposed list.
2. Input the strain value.

Note: The unit of strain is either [micro-def] – for μdef – or [%] ; it can be fixed in "Units" of the program.



2.2. Thermal loading on soils with HSM constitutive model (2D & 3D)

This existing load feature is improved so that it is now possible to define a thermal loading on soil bodies with the Hardening Soil Model; it works also for the other constitutive models with a non-linear elasticity.

For these types of models, as the elastic moduli depends on the stress path, it requires to provide a reference model where to read them.

Activate the tool  **Thermal loading**

1. Define the reference model with current Young modulus.
   Select the model in the proposed list.
2. Input the dilatation coefficient.
3. Define the reference temperature (constant).
4. Define the current temperature field among the 3 options:
   a. Constant temperature
   b. Model: selected in the list of models with thermal type.
   c. File: input the name of a file where the temperature field has been stored after a previous thermal calculation.

2.3. Thermal loading on 1D bodies (2D & 3D)

This new type of load allows the definition of a thermal load on a 1D-body (beam or bar).

Activate the tool  **Thermal loading on 1D-body**

Edition with a uniform temperature

1. Input the dilatation coefficient
2. Define the reference temperature (constant).
3. Define the current temperature
4. Activate "Uniform temperature"

Edition with a uniform temperature and a gradient

1. Input the dilatation coefficient
2. Define the reference temperature (constant).
3. Define the current temperature
4. Activate "Uniform temperature with gradient"

- Input value dT/dX
- Input value dT/dY

2.4. Surface forces (2D)

This load that already existed in 3D is now available in 2D.
It is a variation of "uniform pressure" where user can set a horizontal and a vertical component of a pressure at the surface of a surface body.

Activate the tool  **Surface forces**

1. Input the value of pressure component along x-axis, Px.
2. Input the value of pressure component along y-axis, Py.

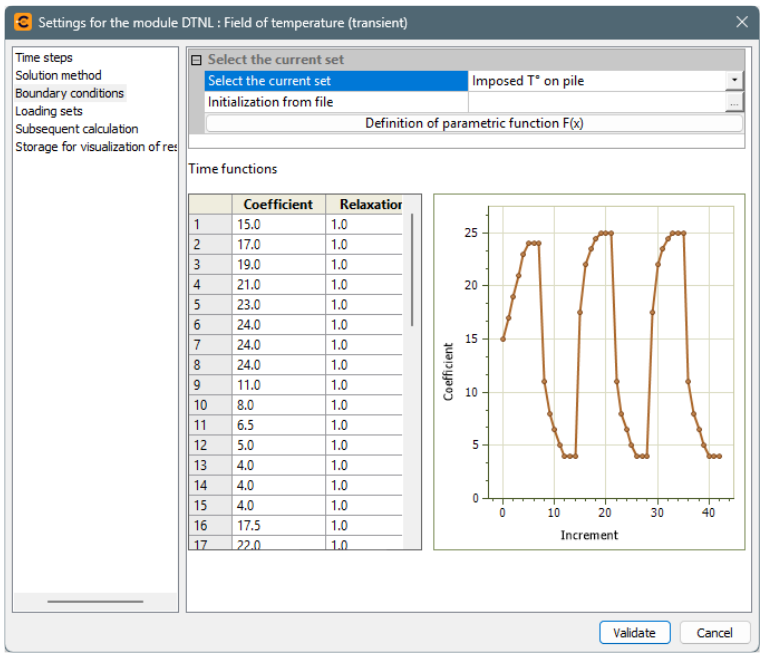
3. Calculation settings

3.1. Display of load increments or times steps

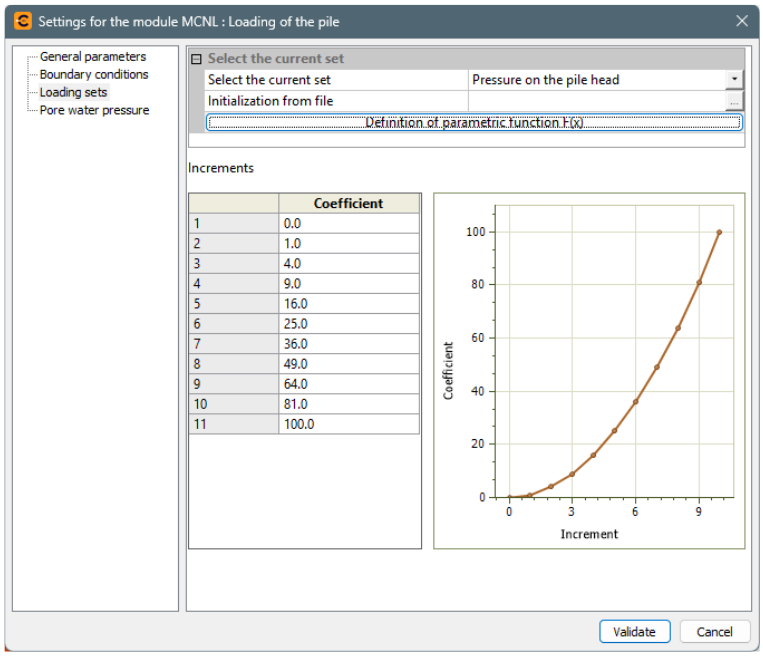
In mechanical algorithms (MCNL, MEXO), one can set a number of increments to pilot the load sets and/or the boundary conditions sets. When performing hydrogeology, dynamics or thermal analysis, one can define the time sequence.

With the new version, it is now possible to have the real-time display of these edited fractions of load or time.

Example for a thermal calculation



Example for a mechanical calculation

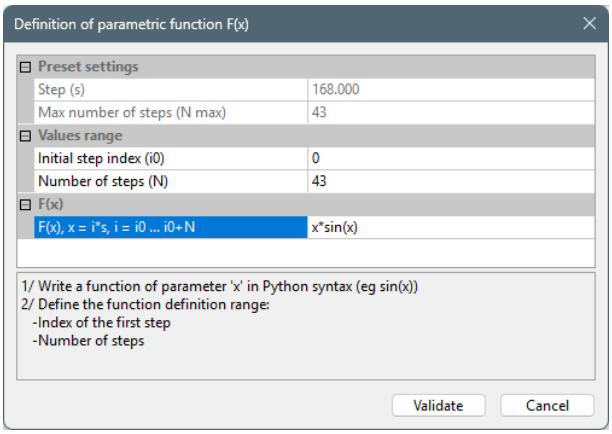


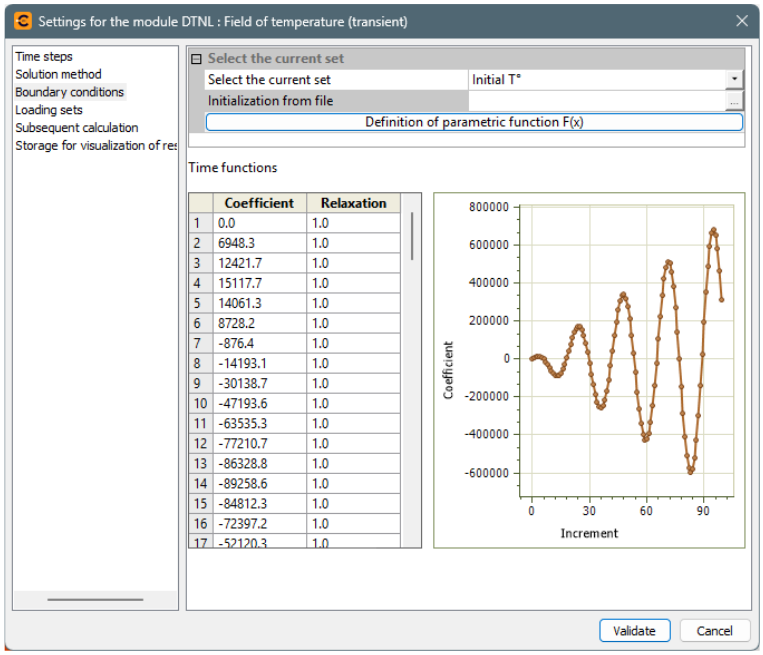
3.2. Editing a parametric function of load increments or times steps

When the piloting signal is derived from a mathematical function, the user can define it using Python pre-programmed functions.

1. Open “Calculation settings”.
2. Activate the “Boundary conditions” or the “Loading sets” tab.
3. Select the current set (if more than 1 has been edited).
4. Click on “Definition of the parametric function F(x)”.
    a. Edit the initial step value “i_0”
    b. The number of steps “N”
    c. The function “F(x)”
    d. Click “Validate”

*Example of definition of a x*cos(x) function.*





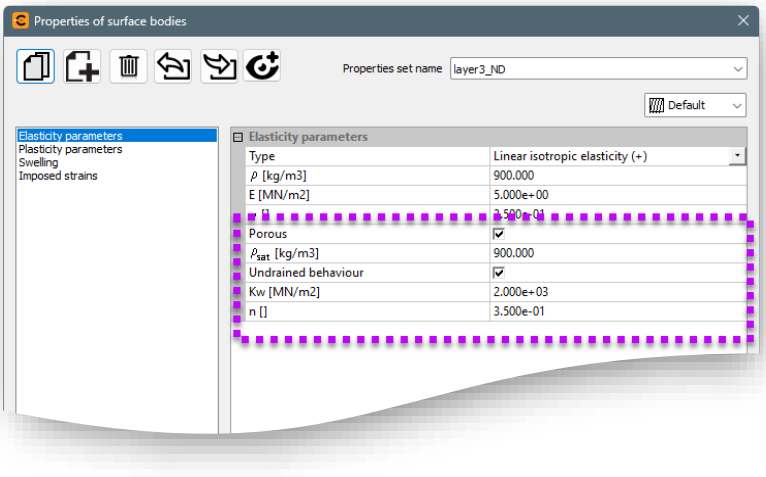
The list of mathematical functions can be found here: [math — Mathematical functions — Python 3.14.5 documentation].

4. Staged construction: staggered consolidation

New options have been integrated to get access to the excess pore pressure field generated by the mechanical loadings in the case of undrained evolutions of ground layers.
This allows now to edit a staggered analysis for simulating a consolidation, by mixing undrained analysis, transient flow and transient evolution of the water table.

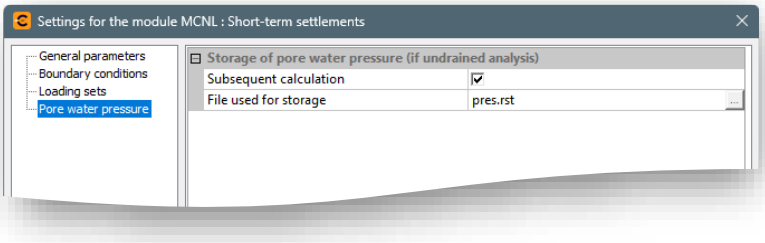
1. Generate a 1st stage (mechanical analysis) where we apply the load – weight of an embankment for example - with undrained behaviour for some of the soil layers.

Example of undrained properties



The excess pore pressure field is calculated and stored.

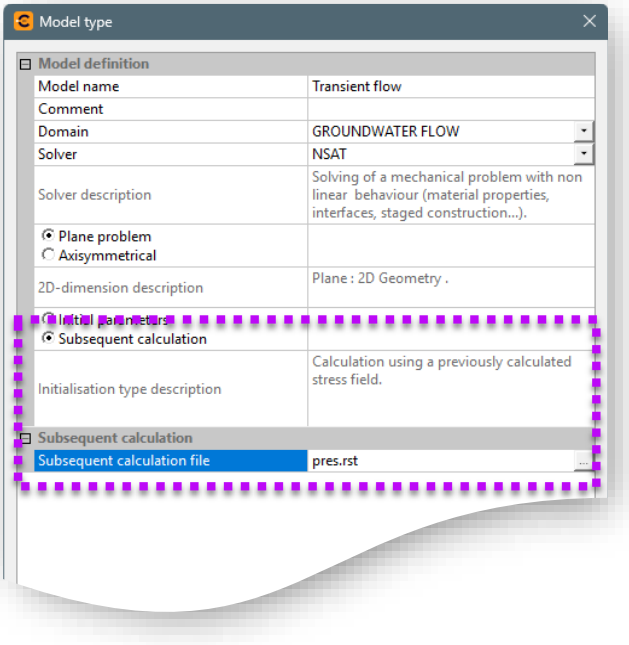
Example of analysis settings for undrained analysis



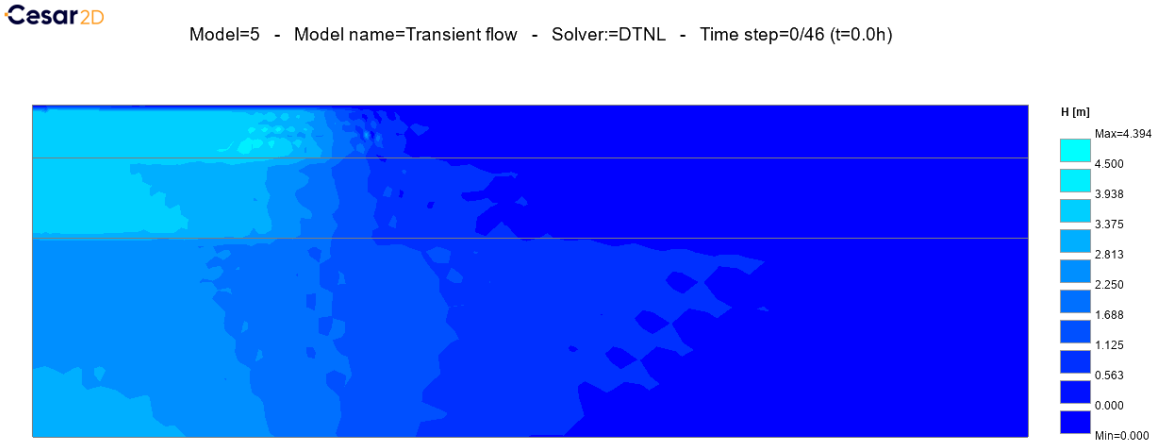
2. The evolution of the pore pressure field in the model is calculated with a transient hydrogeological analysis.
   This model is initialised with the previous head water field, calculated in the undrained analysis.

   Generate a GROUNDWATER FLOW calculation (DTNL – if saturated behaviour of the porous materials - or NSAT – if unsaturated behaviour).
   Define it as a "Subsequent calculation" with the initialisation from the previous stored file.

Example of initialisation of the water head before transient flow calculation



Example of initial pore pressure field, calculated at step #1

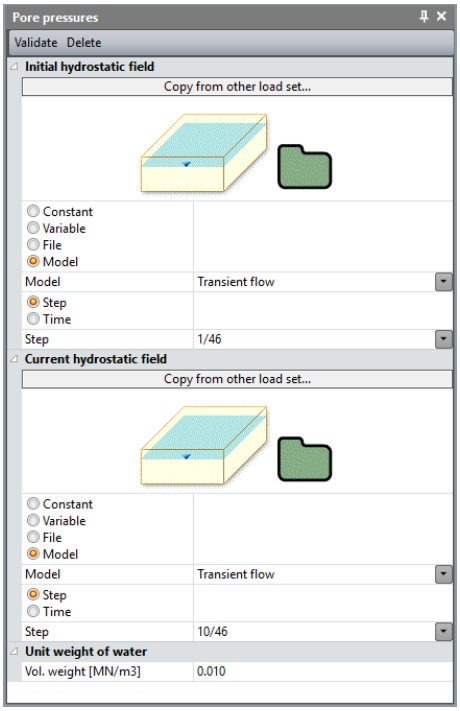


3. Generate a second stage after the initial undrained analysis.
   In this stage, we model the evolution of the pore pressure with several increments (considered as time steps).

   Under PROPERTIES, the porous materials have now a drained behaviour.

   As LOADS, we define the evolution of the pore pressure field in several time steps (identified time steps that reflects the behaviour of the soil mass).
   Activate the "Pore pressures" tool. Select the previous calculated model as reference, set step 1 as initial hydrostatic field and set another time step for the current hydrostatic field. The variation will generate a stress gradient that will load the soil mass.

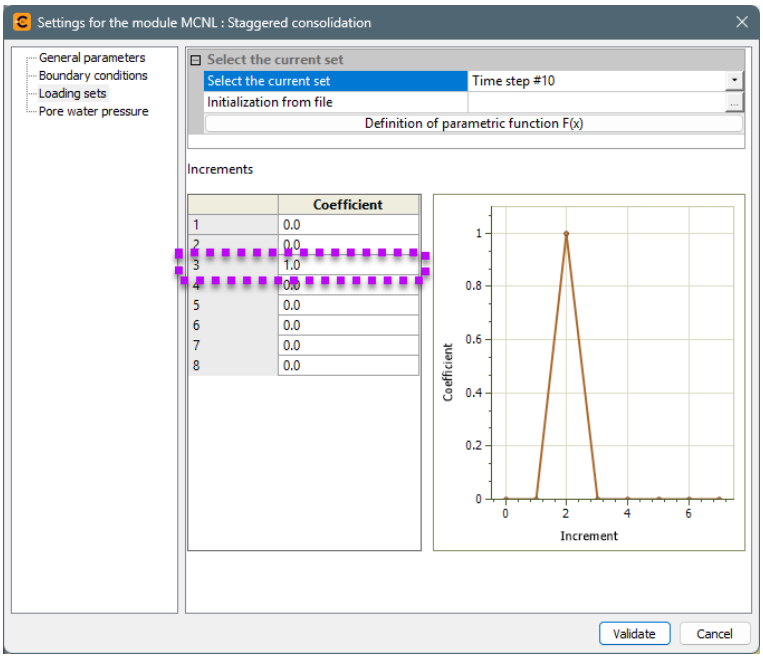
Example of edition variation of pore pressures between initial state and time step #10.



Repeat the previous operations for the desired number of time steps (increments).

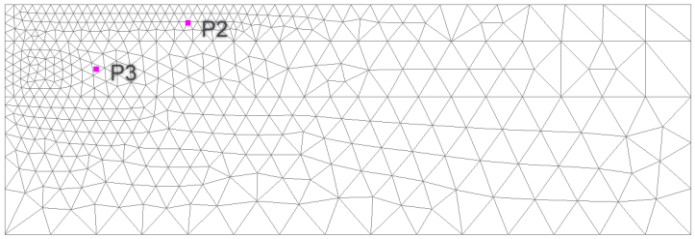
In analysis settings, define for each load its coefficient of activity through the various increments.

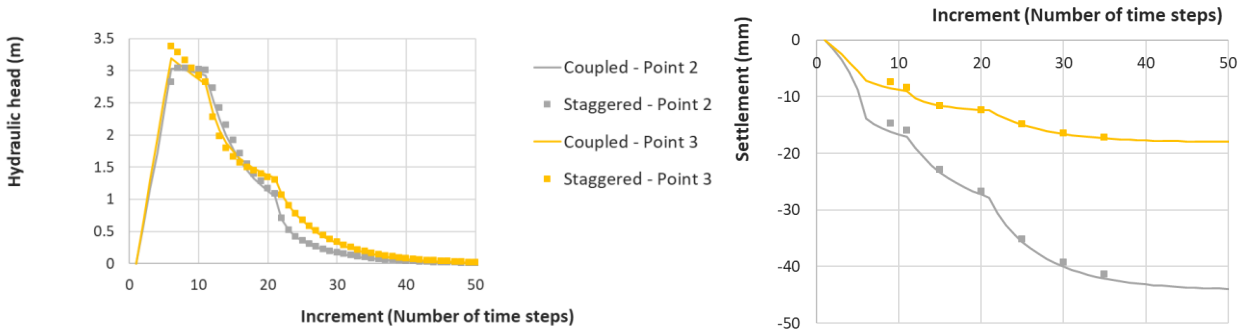
Example of load activity definition – Time step #10.



After the calculation, we can analyse evolution of water head inside the model and the evolution of the settlements with time.

Comparison of staggered analysis and coupled analysis (based on tutorial 2g.02).





This process of consolidation allows for the following analyses:

- Calculation of safety factors (c-phi reduction process, determination of a limit load)
- Integration of joints, structural blocks (1D blocks, shells, etc.).

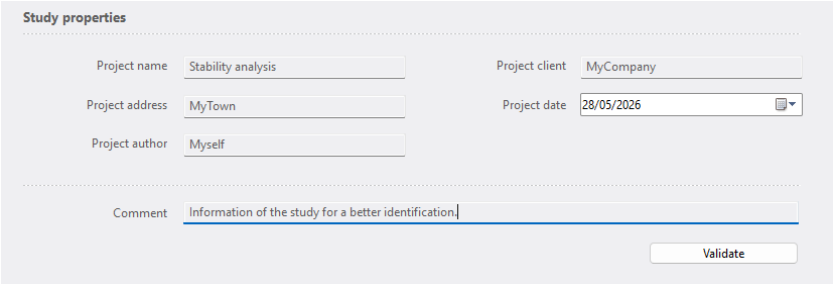
5. Python API

New features, new documentation.

6. Other GUI updates

6.1. Study information

Under the FILE menu, user can edit the information of the study.
This information is stored and is a useful for a quick overview of the study.



This information is integrated in the layout of the document when printed.

6.2. Dynamic analysis (DYNI): selection of time steps stored in the listing file

Dynamic calculations are sometimes demanding a hight number of increments (time steps) for piloting the load sets of the boundary conditions sets.
User can now define the ones to be stored :

- In the listing file
- In the results files.

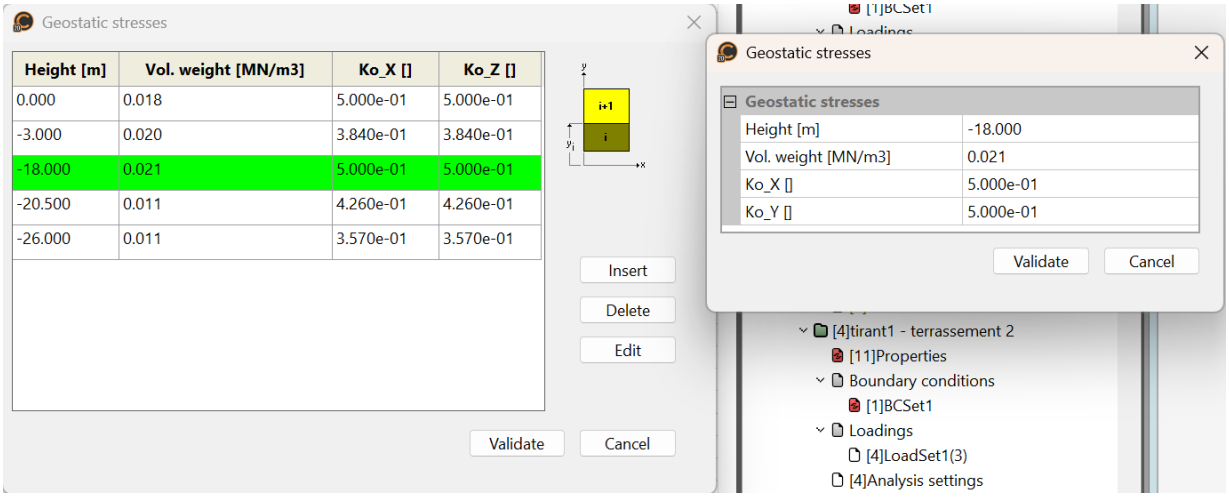
Hence, the size of the resulting files is optimized.

6.3. Tables edition

When edition requires tables, the user can insert a line of data. The new version is proposing a more intuitive method.
First, in the table, the selection of a line defined the next action :

- The line is edited if click on "Edit"
- The line is deleted if click on "Delete"

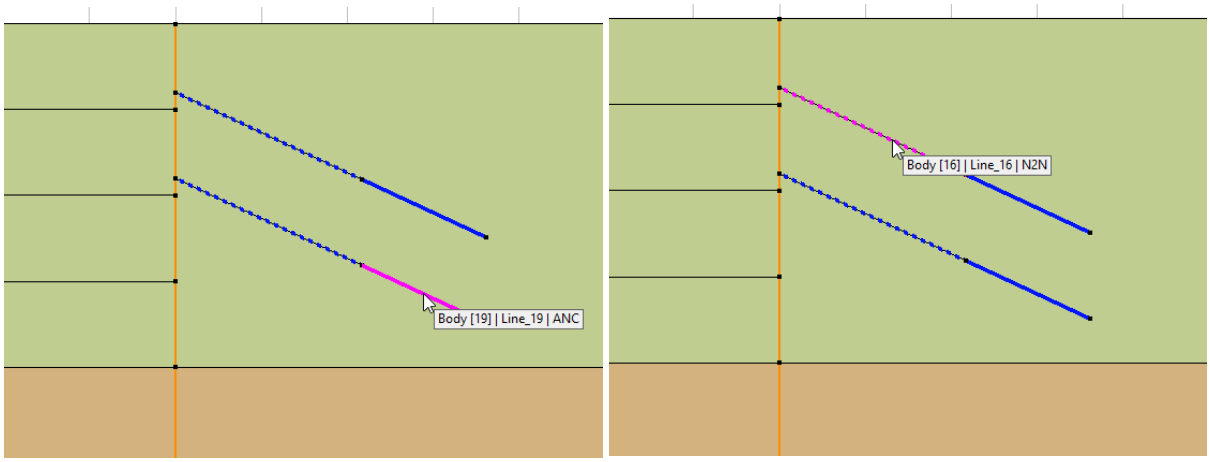
Example of table edition: geostatic stresses)



6.4. Improved object information (1d-body type, mesh info…)

On mouse over an object, the information about the object is displayed.

Example for 1D bodies : Anchored (ANC) or Node-to-Node (N2N)



6.5. Display of results

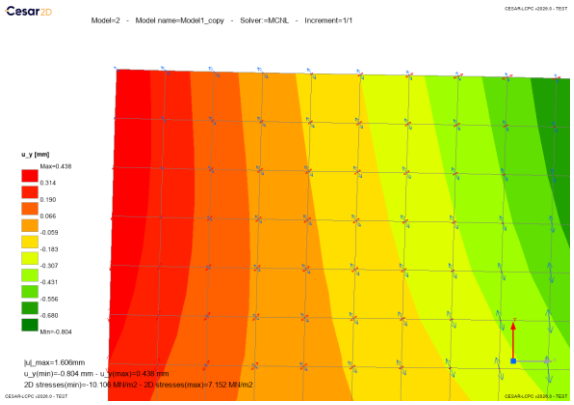
Several improvements:

- Colour legend: more compact, text between colour boxes
- The titles are displaying more information (safety factor...).
- Min/Max values are displayed in lower legend

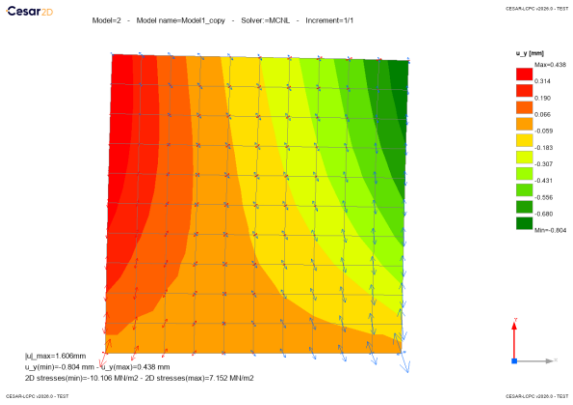
A new tool allows now to reorganize the view of the model and the legend in a default state. The default view also avoids to override details of calculations -et the top - and maximal values – at the bottom.

Click on to  to reset the display to the default state.

Initial view



Default view



6.6. Settings of graphs

Additional options have been integrated for a better edition of the charts.

- Display calculated model information in the Charts view.
- Optimize the definition of the primary curve grid so that the grid is not too dense.
- The font size adjustment tool is not available to control the size of the text in graphs.
- The colour palette for curves is modified to improve contrast with white background.
- For analyses function of time (NSAT, DYNI, CSNL, MPNL...), it is now possible to select the time steps values as abscissa, preferable to increment steps.
- Improve the display of increments in line-graphs property grid

6.7. Colour palette

The colour palette has been enriched.
Its behaviour has been improved. On mouse over one of the colour patches, the items of the model with the same colour are highlighted.

6.8. Import of dxf files

The import of has been improved.
The list of entities imported are sum up below.

DXF Entity	Notes
LINE	edge straight curve
CIRCLE	Full circle edge
ARC	Partial-circle edge
ELLIPSE	Ellipse edge
SPLINE	Spline edge
POLYLINE	Supports flags: Closed Line (**1**), 3D Polyline (**8**), 3D Polygon Mesh (**16**/**17**), Polyface (**64**)
LWPOLYLINE	Lightweight polyline; supports open (**0**/**128**) and closed (**1**) types
VERTEX	Sub-entity consumed while building POLYLINE / Polyface
POINT	Imported as vertex
3DFACE	Imported as face
INSERT	Block reference — exploded into its constituent entities before transfer

6.9. New default theme

User will notice a new default theme of colours for the graphical user interface.
This theme is named "MS Visual Basic 2015 Light".

If user wants to come back to the previous state, he activates OPTIONS at the top right of the GUI. Select here a new theme.

It's the opportunity to remind that user can also set here a new layout, display scheme or font.