



Reference manual

| Ref: CESAR-MU(3D)-v2021.0.1-EN

Table of content

1. Presentation of the user interface	9
Introduction	9
The interface	9
The "FILE" tab	10
The "General tools" bar	11
The "Display and selection tools" bar	11
Vertical "colour palette"	12
The tabs – The various project stages	13
The grids – "Apply / View" type dialogue boxes	13
Model tree diagrams	16
Data tree diagram	16
2. File management	19
Files names	19
Files generated by CESAR-LCPC	19
3. Conventions and notations	21
"Drawing" points and lines included in the geometric model ("Sketch")	21
The "geometric model": A set of bodies	21
The "models"	24
Mode "G" (Geometry) vs. mode "M" (Mesh)	25
4. Selections	27
Possible selections	28
Selection actions	30
5. Definition of model geometry	31
 Working plane definition	31
 Assigning points	33
 Lines	34
 Circular arcs	36
 Elliptical arcs	38
 Spline curves	38
 Information and editing of the length and angle of a line	39
 Shoulder fillets	42

	Line intersection	42
	Line cut	42
	Simplification of edges	43
	Translation, Symmetry, Rotation	44
	Creation of "1D bodies"	47
	Creation of "surface " bodies	48
	Rotating surface body	50
	Coons surfaces	52
	Definition of a "B-spline surface" type of surface bodies with imposed interior points	54
	Creation of "surface" type model elements from a "volume" type model element	55
	Creation of a surface body from the outer faces of volume bodies	55
	Creation of predefined volume bodies	56
	« Super-element » body (hexaedron, pentaedron, tetraedron)	57
	Definition of an "any volume" type of volume body defined by its exterior surfaces	58
	Extrusion operations	58
	Operations on bodies: Translation, Symmetry, Rotation	59
	Intersections between volumes or surfaces	62
	Merging volumes or surfaces	64
	Intersection of volumes by a user plane	64
	Partition of a surface body using lines	65
	Merge bodies	65
	Explode bodies	65
	Definition of anchors	66
	1D body connections	66
	Creation of interfaces	67
	Manual element creation: "Multi-point" bodies	67
	Attachment of anchors	68

	Body/wall attachment	71
	Scripts	73
	Delete	74
	Undo	74
	Redo	74
	Colors	75
	Model properties	75
	Body-related information	75
	Correction of the geometry	75
6. Mesh definition		77
	Constant density	77
	Fixed length density	78
	Progressive-density	78
	Mesh of 1D-bodies	80
	1D element orientation	80
	Surface meshing	81
	Volume meshing	85
	Surface body orientation	86
	Mesh properties	86
	Body-related information	87
	Correction of the mesh	87
7. Calculation modules and elementary physical problems		89
	Library of elementary physical problems	89
	Content of each elementary physical problem	90
	Compatibility of "physics"	90
8. Model definition		91
	"Model" management	91
	Properties of a model	93
	"Sharing" of sets (properties, boundary conditions, loads) among several models	95
	Construction stages process in geotechnical analysis	97
	Initial stress field for geotechnical analysis	99

9. Definition of model properties	103
Definition of the property tables	103
 "Graphical" application of properties	107
 Assign/view of properties in a table	107
 Definition of the Active / Inactive attribute	108
10. Initialisations of parameters	109
Introduction	109
 Initialization of displacements by rigid body motion	111
 Initialization of displacements by file input	111
 Initialization of speeds by rigid body motion	111
 Initialization of speeds by file input	112
 Uniform initial stresses by group	112
 Initial geostatic stresses	112
 Initialization of stresses by file input	113
 Uniform initialization of loads at any point	114
 Position-based load initialization	114
 Uniform initial load by body	115
 Initialization of loads by file input	115
 Uniform initialization of temperatures at any point	115
 Uniform initial temperature by body	116
 Initialization of temperatures by file input	116
 Uniform initialization of pressures at any point	117
 Position-based pressure initialization	117
 Uniform initial pressure by body	118
 Initialization of pressures by file input	118
11. Definition of a set of boundary conditions	119
Introduction	119
Creation of a new set of boundary conditions	122
 Mechanics: Sides and bottom ($w=0$) supports	122

	Mechanics: Sides and bottom ($u=v=w=0$) supports	123
	Mechanics: Sides, top and bottom ($w=0$) supports	123
	Mechanics: General definition of imposed displacements	124
	Mechanics: Coordinate system changes	126
	Hydrogeology: Imposed loads	127
	Hydrogeology: Imposed loads varying linearly with depth	127
	Hydrogeology: Seepage conditions	127
	Hydrogeology: Exchange conditions	128
	Thermics: Imposed temperatures	128
	Heat transfer: Exchange conditions	129
	Pressure: Imposed pressures	130
	Pressure: Exchange conditions	130

12. Definition of a load sets

133

	Introduction	133
	Creation of a new load case	136
	Mechanics: Nodal forces	137
	Mechanics: Uniformly-distributed pressure	138
	Mechanics: Non-uniform distributed pressure	139
	Mechanics: Hydrostatic pressure	140
	Mechanics: Surface forces	141
	Mechanics: Excavation forces	142
	Mechanics: Gravitational forces	145
	Mechanics: Constant stresses	146
	Mechanics: stresses input from a file	146
	Mechanics: Initial thermal stresses	147
	Mechanics: long-term effects	148
	Change in water table position	150
	Definition of the bodies assigned due to the change in water table	152

	Uniformly-distributed flow	152
	Point flow rates	152
	Volumic flow rates	153
13. Definition of "computation module" data		155
<hr/>		
	Introduction	155
	Accessing the computation settings	155
	The LINE module	156
	The MCNL module	157
	The TCNL module	161
	The DYN1 module	163
	The FLAM module	167
	The LINC module	169
	The LINH module	171
	The MODE module	172
	The SUMO module	174
	The DTNL module	177
	The NSAT module	182
	The TEXO module	186
	The MEXO module	192
	The CSNL module	195
	The MPNL module	200
14. Computation management		205
<hr/>		
	Introduction	205
	General analysis options	205
	Analysis manager	206
	Interrupting the computations	207
	Editing the data file	207
	Editing the "listing"	207
	Save, Save as, etc.	207
	Search, etc.	207
	Show the mesh of the model	207
	Mesh display options	207
15. Display of results		209
<hr/>		
	Introduction	209
	Type of results to display	210
	Envelope of iso-surface results	211

	Choice of entity	212
	Mesh settings	213
	Deformed mesh settings	216
	Contour plot settings	217
	Isosurfaces	220
	Vector options	221
	Tensors settings	222
	Beam results settings	224
	Contact state display options	225
	Legend	226
	Cutting plane	227
	Plane section analysis	230
	Information	231
	Results displayed by group	232
	Listing of results	232

16. Charts of results **233**

Introduction	233
 Points	234
 Set of points	234
 Line	235
 Line set	235
 Reverse the line set orientation	235
 Charts for a line set	236
 Charts for a set of points	237
 Parametric charts	238
 Chart settings	239
 Export charts data	240

A1. Elements library	241
A2. Constitutive models	243
Constitutive laws for the element families used in mechanics and geomechanics.	243
Toolbox for user constitutive model definition (only for families 1 and 2)	244
Constitutive laws for the element families used in Coupled analysis.	245
A3. Identification of scalar results	247
Volume bodies	247
1D bodies (bars, beams)	250
Shells	250
Vectors	251
Tensors	252

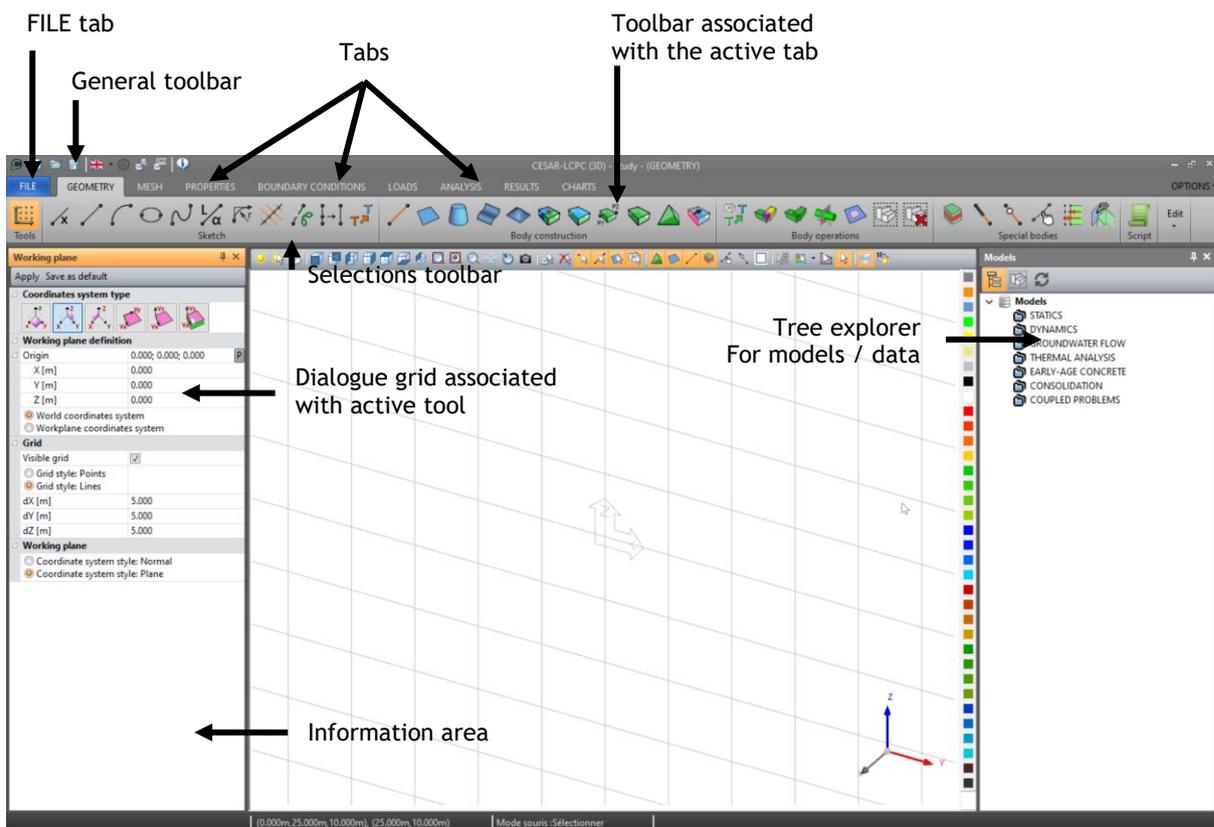
1. Presentation of the user interface

Introduction

This chapter provides the general operating and use instructions for the software. Moreover, it describes in particular the various elements composing the "interface". You'll discover in this chapter the guidelines for working with menus, tabs, tools and dialogue boxes, and start to become familiar with the overall approach to using this application.

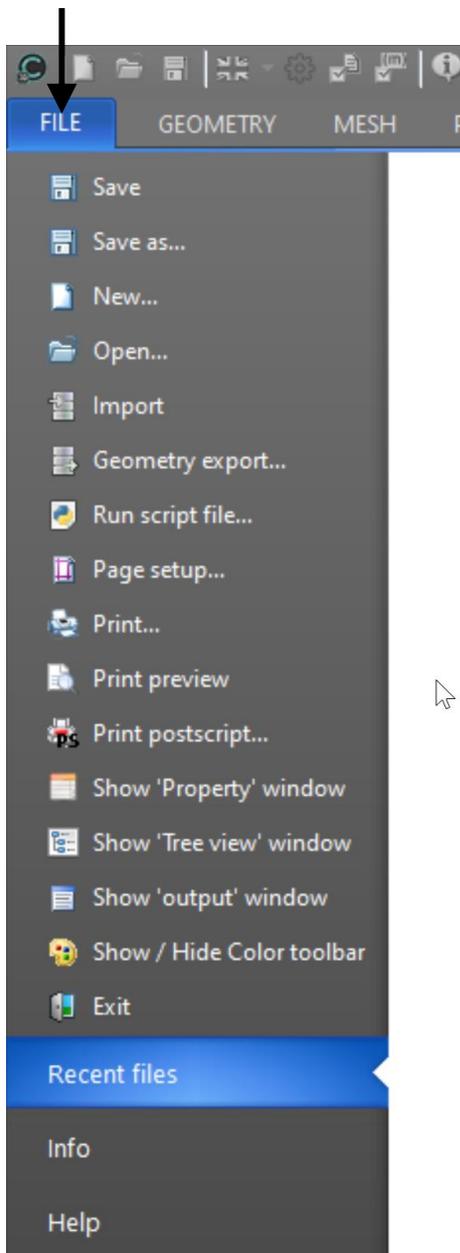
The interface

The figure below displays the software interface along with its primary components.



The "FILE" tab

The first user interface component is the FILE tab. Activating it displays a drop-down menu that provides access to most most conventional functions.



Let's begin by highlighting the following basic commands:

-  New: Opens a new project
-  Open: Opens an existing project
-  Save: Saves the current project
- Save as: Saves the current project by assigning a new project name
-  Exit: Closes the application

The "General tools" bar

The "General tools" bar allows for rapid and permanent access to a set of very commonly used functionalities in carrying out a project.



This toolbar contains the following functions.

- Files management
 - New: create a new study
 - Open: Opens an existing project
 - Save: Saves the current project
- Settings
 - Choice of interface language
 - Application settings: definition of general application preferences of the application
 - Study settings: definition of preferences for the current study
 - Definition of common units

The "Display and selection tools" bar

The "Display and selection toolbar" allows for rapid and permanent access to a set of very commonly used functionalities in carrying out a project. This permanent toolbar is updated with the entities available through the model process.



This toolbar notably contains the following functions.

- Displays
 - The selected elements are hidden (no longer visible)
 - Only the selected elements are visible
 - All model elements once again become visible
- Views
 - Predefined views (front, back, left, right, top, bottom, perspective)
 - Full-screen display of the structural representation
 - Full-screen display of the structural representation, with updated view from the "focus" point
 - Close-up: Expanded view of the image projected onto the screen
 - Displacement of the proposed structural image on the screen (panoramic effect)
 - Rotation of the structural view
 - Work space screen capture
- Selections of entities
 - Selection option settings
 - No entity may be selected
 - Activation of the point selection attribute
 - Activation of the line selection attribute
 - Activation of the surface selection attribute

-  Activation of the "body" selection attribute
- Selections of bodies
 -  Selection/activation of volume bodies
 -  Selection/activation of surface bodies
 -  Selection/activation of 1D bodies
 -  Selection/activation of interface bodies
 -  Selection/activation of user defined bodies
 -  Selection/activation of link bodies
 -  No selection/activation of bodies
- Other selections
 -  Selection per type of mesh (only at step "MESH")
 -  Selection by colour
 -  Selection by broken line
 -  Selection tool
- Other displays
 -  Transparency (intensity is set in "Study settings" > "Displays")
 -  Display of the names of the bodies (set in "Application settings > "Fonts").

Vertical "colour palette"

It is possible to associate a colour to any body of the model. This definition is done with the vertical "Colour palette", located on the right side of the working window.



Colour modification of the selected entities

The fact to click with mouse left-button on one of the colours affects this chosen colour on all the selected entities.

Selection of entities with a specific colour

The fact to click with mouse right-button on one of the colour selects all the entities having the specified colour.

The tabs – The various project stages

The interface then offers the user a series of "tabs" corresponding to the various project execution stages.



Generally speaking, activating a tab triggers the display of a specialized toolbar that allows performing the particular stage under consideration (e.g. Geometry, Mesh, Properties).

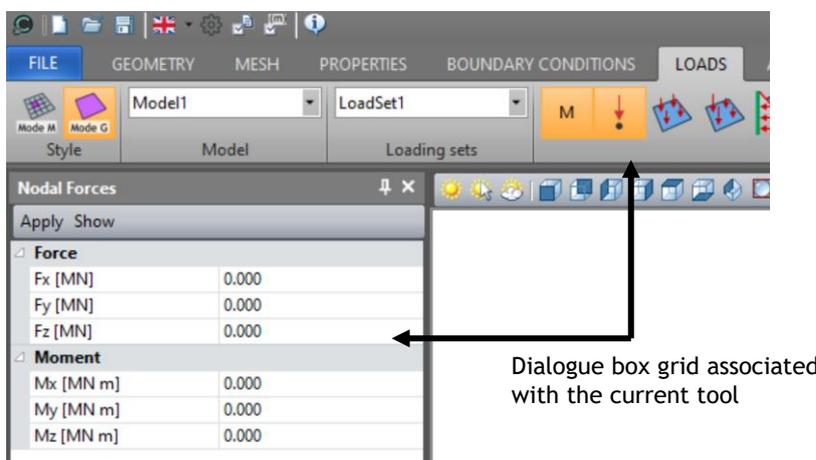
As a default, CESAR 3D proposes the following tabs:

- Geometry: Definition of the structural geometry.
- Mesh: Definition of the structural mesh.
- Properties: Definition of the model "properties" (materials, cross-sections, etc.).
- Boundary conditions: Definition of the sets of support conditions.
- Loads: Definition of all sets of loads.
- Analysis: Execution of the computations.
- Results: Graphical visualization of results.
- Curves: Visualization of results in the form of curves.

A detailed description of the functionalities associated with the tabs listed above is provided in individual chapters.

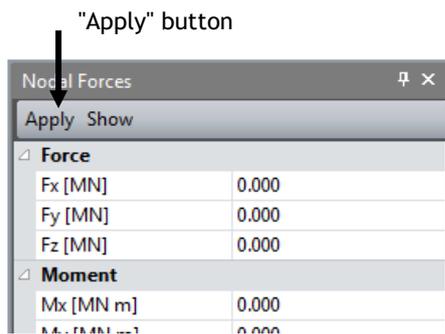
The grids – "Apply / View" type dialogue boxes

Activating a tool in the bar associated with the current tab most often launches the display of a grid dialogue box on the left side of the screen, as indicated in the next figure.



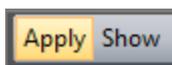
The above dialogue box serves, for example, to assign uniform pressures on a beam element. From a generic standpoint, a property is assigned to an entity selected from the model by following the standard approach given below:

Assigning properties (Button: Apply)



The property assignment step is characterized by the following sequence of operations:

- Activate the tool that enables the action
- Define the set of associated properties
- Select the entities (surfaces, bodies, etc.) featuring the given property
- Click the "Apply" button in the dialogue box to assign the properties previously defined to the selected entities.



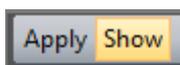
The "**Apply**" button serves to **assign the properties** defined in the dialogue box to the set of **selected entities**.

Recovering the properties assigned to an entity (information mode using a right-button click of the mouse)

It is generally possible to "recover" the properties assigned to an entity by simply **right-clicking** on this entity. The assigned properties will then be displayed in the dialogue box grid.

Visualizing those entities possessing the given properties (Button: Show)

Most interactive dialogue boxes contain a "View" button to allow visualizing the set of all entities possessing the property or properties defined in the dialogue box.

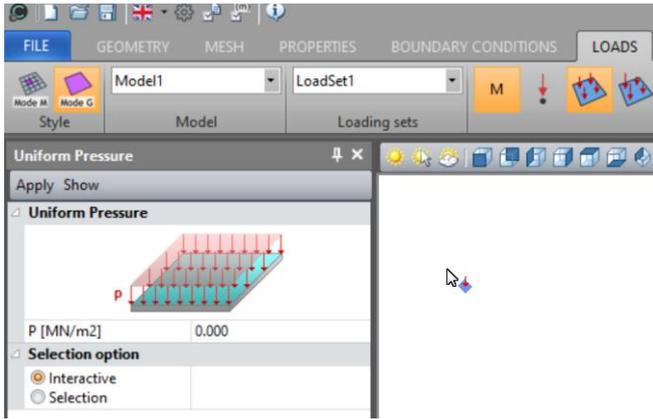


The "**Show**" button makes it possible to **select those entities** (and thus to **visualize** them) featuring the **properties** defined in the dialogue box.

The grids – Dialogue boxes with "direct assignment"

In some cases, the ability is needed to directly assign a property by clicking on the given entity (as opposed to a preliminary selection). Let's take the example of the dialogue box for defining surface pressures.

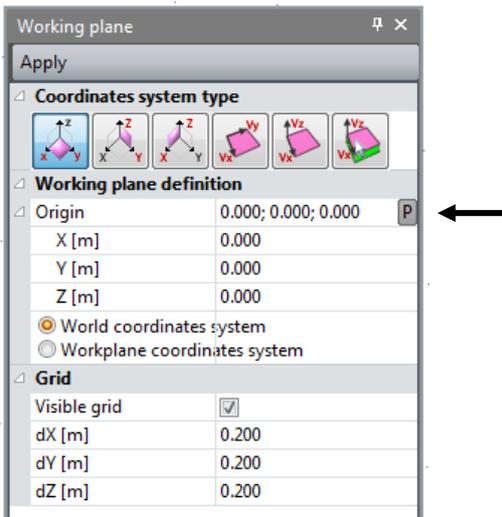
It can be seen here that the given dialogue box does not contain an "Apply" button since the assignment step is in fact performed by clicking directly on the desired surface (this operation is used to define the loaded "side" of the particular surface).



Recovering points or vectors in the grids – Dialogue boxes

In many cases, it is necessary to define within a dialogue box the coordinates of a point or vector in order to characterize an operation (Working plane definition, translation operation, etc.). In such cases, it is often worthwhile to "recover" the point or vector coordinates using the geometric model that has already been partially defined.

This operation becomes possible thanks to the "**P**" "**V**" buttons placed in the appropriate dialogue boxes. The capture below provides an example of the special case of a dialogue box that allows defining the working plane.



To recover the coordinates of a model point, simply follow the two-step sequence laid out below:

1. Click on the **P** button found in the dialogue box.
2. Click on the specific point in the geometric model. The coordinates of the indicated point are now displayed in the dialogue box.

To recover the coordinates of a vector using model points, simply follow the three-step sequence presented below:

1. Click on the **V** button in the dialogue box.
2. Click on a point in the geometric model in order to define the "starting point" of the vector.
3. Next, click on a second point to establish the vector end point. The coordinates of the vector so defined are now displayed in the dialogue box.

Model tree diagrams

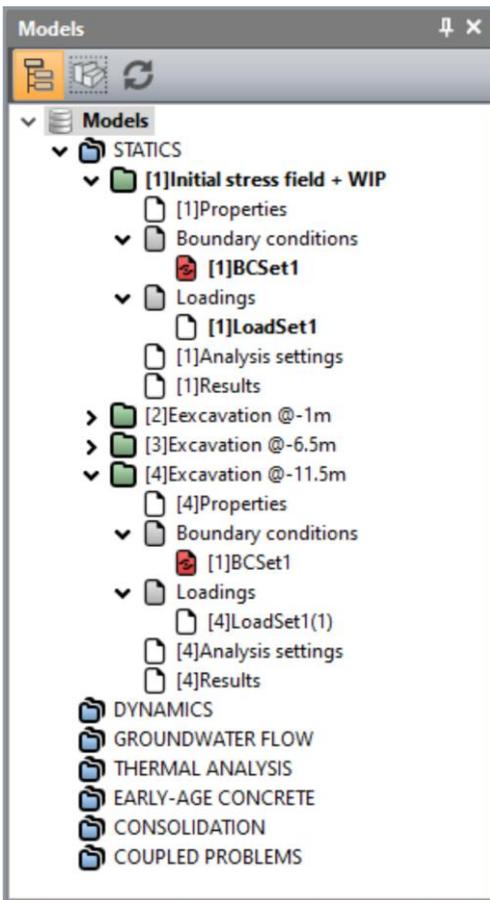
This work space serves to visualize in real time the set of models created the set of models created as well as their contents.

Presented in a Windows Explorer format, this space makes it possible to know at any time which is the active entity (load case, properties, boundary conditions, etc.), as it is always displayed in bold.

Double-clicking on an entity (e.g. load case) automatically updates the work space display so that it now corresponds to this choice.

With this tree diagram layout, it is possible to modify the assigned default names by using the [F2] key.

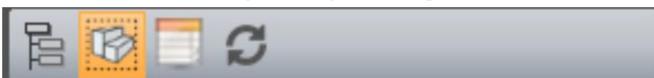
The following diagram lists 4 STATICS models (stages #1 to #5).



Data tree diagram

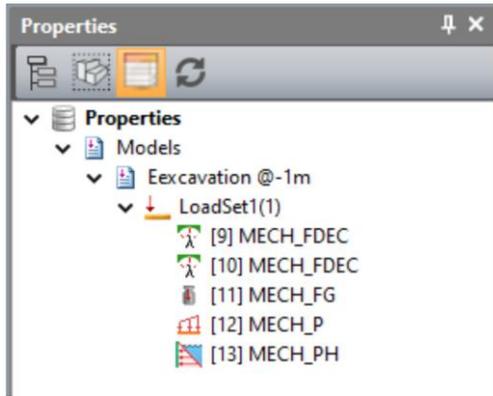
A single click shifts from the model tree diagram to the data tree diagram.

The data tree diagram provides access to the list of bodies composing the model. Select any one of these listed bodies by simply clicking on it.



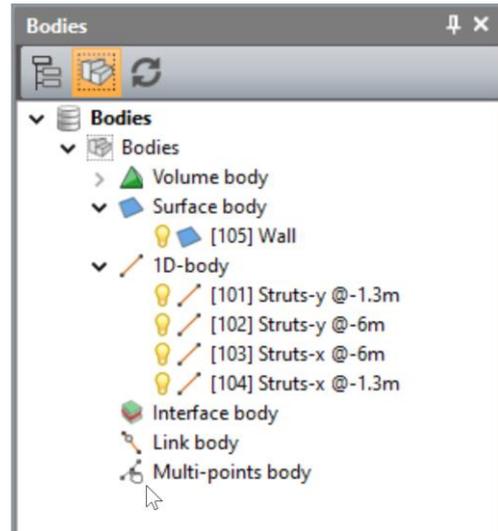
For each entity (load case, boundary conditions, etc.), the list of data the user has assigned to the entity can be called up.





Example 1: List of loadings associated with the "LoadSet1" load case:

- Excavation load (MECH_FDEC),
- uniform pressure type of mechanical loading (MECH_P),
- hydrostatic pressure type of mechanical loading (MECH_PH),
- gravity force type of mechanical loading (MECH_FG)



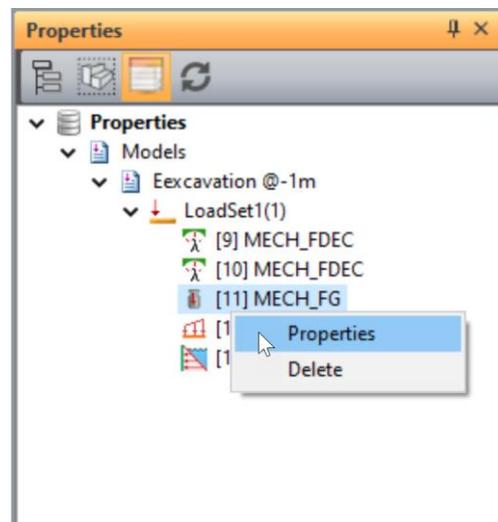
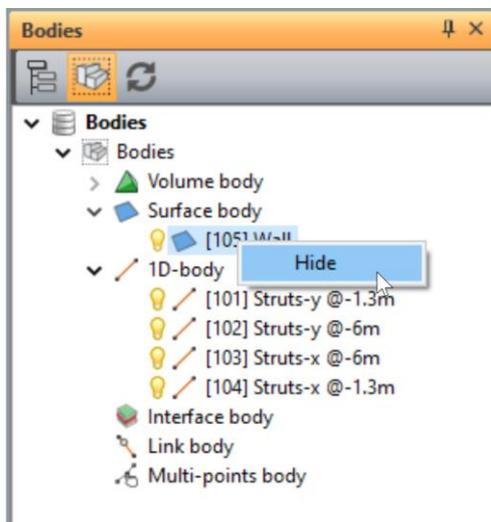
Example 2: List of the model component bodies:

- volume bodies,
- 1 surface body,
- 1D body.

The data tree helps the user to manage his model data. Bodies, , are shown or hidden.

About data affected to the model, , it is possible:

- To show the part of the model where they are affected. A click on the selected entity in the data tree activates the part of the model in the working space.
- To access the details: loads, boundary conditions material properties... A right click on the entity of the data tree activates « Properties ». The dialogue grid is then updated.



2. File management

Files names

The name of the study shall not:

- start with a number (but can contain one);
- contain one of these figures: "_", " ", "/", "\»; "(" or ")";
- exceed a given number of characters (default value is 20, set in the file SOLVCES.MTRL).

Files generated by CESAR-LCPC

All along the process of generation and analysis of the model, CESAR-LCPC will generate several types of files.

We take as example a study named TEST.

Files of the study:

- TEST.cleo26 for CLEO2D.
- TEST.cleo36 for CLEO3D.

This file contains all the information about the model (mesh, properties and results). It is the link between the other files described hereafter.

Data file for the kernel:

This ASCII file is automatically generated by the 2D or 3D GUI. It is the translation of all the model information requested for the calculation. As one study file can contain several calculation models Mi, the generated file will be named: TEST_Mi.data.

Results files

- TEST_Mi.rsv4, binary file. It contains the results of model Mi ;
- TEST_Mi.list, ASCII file. It contains the model data, the results and the calculation process.
- TEST_mail.resu, binary file. It contains the mesh data.

Storage files:

During staged construction, the restart/initialisations are linked to stored stress fields. These are stored in file with « rst » as extension. Thus, the calculation of the file TEST_Mi.data will generate the file TEST_Mi.rst.

It is important for the user to store these files as they may be useful in further calculations.

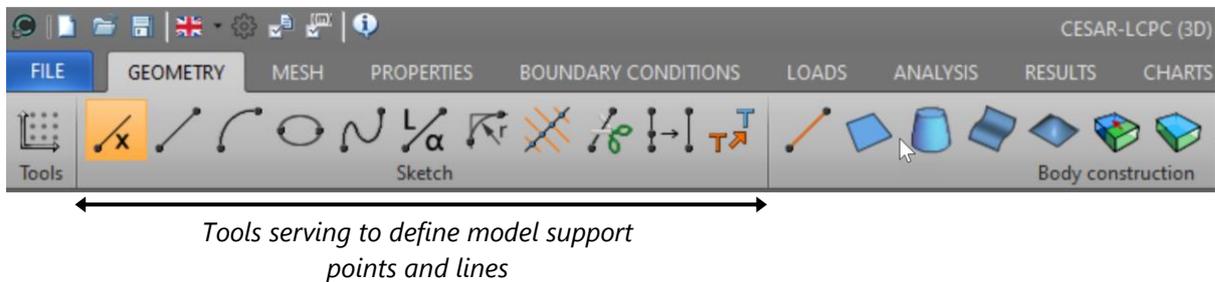
3. Conventions and notations

The objective of this chapter is to define the set of conventions and notations used as part of the CESAR application in general and within this manual in particular. This preliminary step is considered key to avoiding any confusion or misunderstanding of the explanations provided.

"Drawing" points and lines included in the geometric model ("Sketch")

To define the geometry of the particular structure, a set of both **points** and **lines** most often need to be defined for the purpose of supporting the geometric model to be built.

Defining these points and lines involves use of the tools found in the "Drawing" section of the "Geometry" toolbar.



It's important to mention here that these points and lines are merely the entities used to assist in building the geometric model. In this capacity, they are not therefore an integral part of the structure's "geometric model" and, for this reason, may not be "meshed".

The sole entities composing the given "geometric model" are what we will refer to generically herein as "**bodies**". It will be shown that for a point or line to be "meshed" (and thus belong to the geometric model), said point or line must be transformed into either a "point body" or a "1D body".

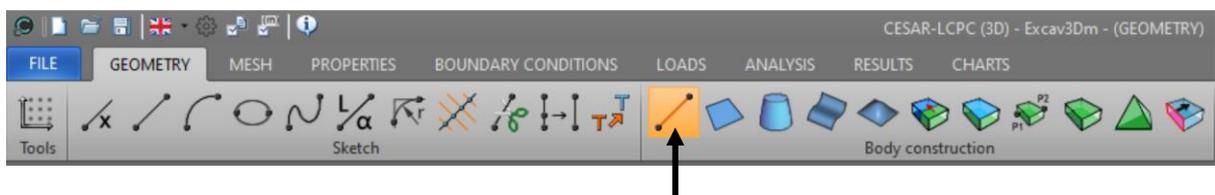
The "geometric model": A set of bodies

The "geometric model" of the given structure in reality consists of a set of "bodies". For starters, the following types of standard bodies shall be distinguished:

- 1D-Bodies: After meshing, these bodies will for example become "bar" or "beam" type elements.
- "Surface" bodies: After meshing, these bodies will for example become plate or of shell elements.
- "Volume" bodies: After meshing, these bodies will become finite volume elements.

All "surfaces" and "volumes" created in the "Geometry" module are by default transformed into "surface" and "volume" bodies, respectively.

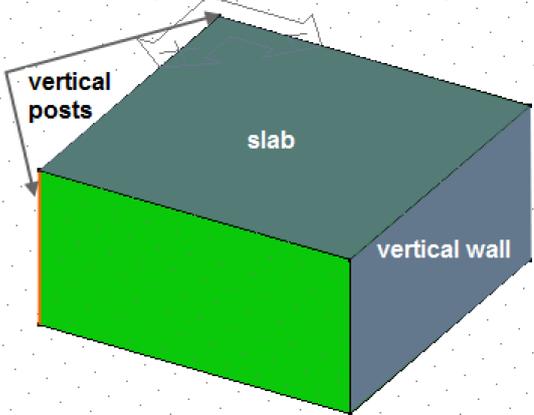
For a line to be considered as a "1D body" however, it must be explicitly declared as such using the tool below:



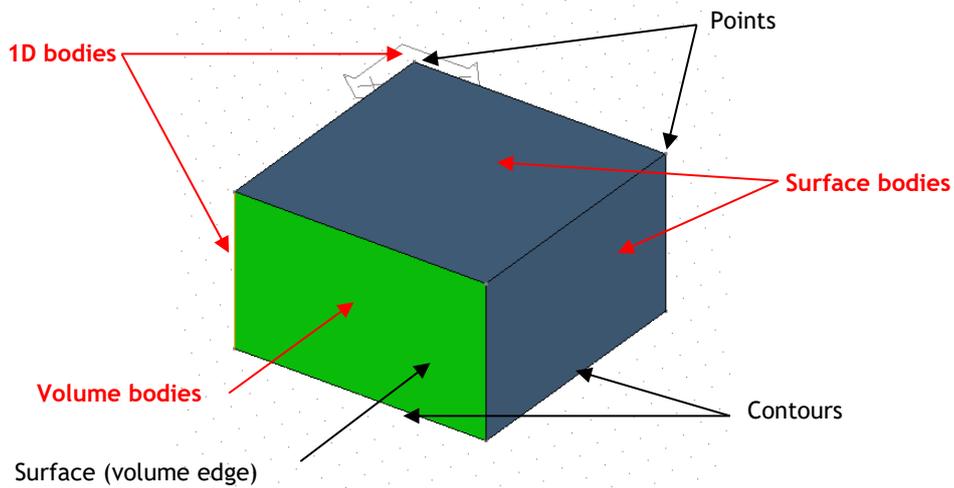
To illustrate this notion of "body", let's consider the various steps leading to the creation of a geometric model composed of:

- a parallelepiped ground volume,
- a plate (slab) on the upper part of this volume,
- two columns (piles) and a vertical wall.

Step 1: Constructing the 4 points and lines of the upper surface	
	<p>At this stage of the construction, we are thus given:</p> <p>A working drawing comprising:</p> <ul style="list-style-type: none"> - four points - four contours <p>to be used as a support in building the upper surface of the volume</p>
Step 2: Defining the surface	
	<p>At this stage of the construction, we now have a geometric model composed of:</p> <ul style="list-style-type: none"> - one "surface body" (the surface created is set by default as a body)
Step 3: Defining the volume by extrusion	
	<p>During the extrusion operation, the application "asks" whether it is preferred to retain the surface body as the basis for extrusion or, on the other hand, if the body should be deleted (default option). In the present case, the body will be kept in order to model the upper slab.</p> <p>At this stage of the construction, we now have a geometric model composed of:</p> <ul style="list-style-type: none"> - one "surface body" (upper slab) - one "volume body"
Step 4: Defining the vertical wall	
	<p>The surfaces constituting the lateral edges of the volume are not bodies by default. To model a vertical wall (using plate elements), it is required to explicitly indicate that the given vertical surface is indeed a "surface body".</p> <p>To proceed, the "Creation of a surface body on the outer volume surface" tool shall be used.</p> <p>At this stage of the construction, we therefore have a geometric model composed of:</p>

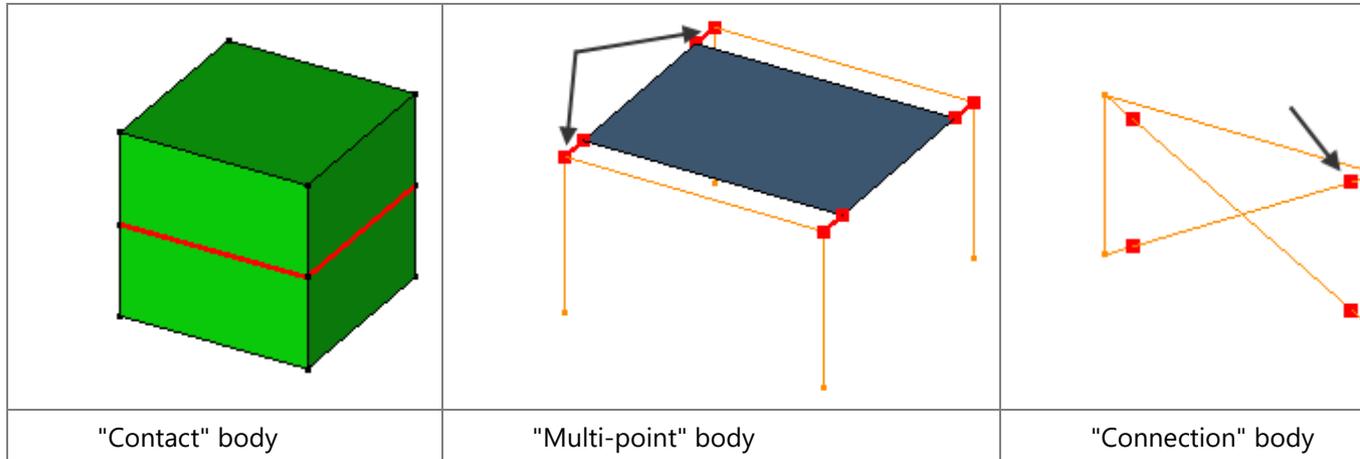
	<ul style="list-style-type: none"> - two "surface bodies" (upper slab and vertical wall) - one volume body
<p>Step 5: Defining the two vertical columns</p>	
 <p>The diagram shows a 3D model of a structure. A green rectangular volume is labeled 'vertical posts'. Above it is a grey rectangular slab labeled 'slab'. To the right is a blue rectangular wall labeled 'vertical wall'.</p>	<p>The two "1D bodies" making up the two columns visualized in the opposite figure will now be defined by applying the "Creation of 1D bodies" tool.</p>  <p>We therefore now have a geometric model composed of:</p> <ul style="list-style-type: none"> - two "surface bodies" (upper slab and vertical wall) - one "volume body" (the ground) - two "1D bodies" (the posts)

The following figure summarizes the various notation conventions used in the above example:



To complete the geometric model, three additional types of bodies will be used, namely:

- "Multi-point" body: To model, for example, either the linear relations between points or "special elements" containing user matrices
- "Contact" body: To define a contact zone between two solids
- "Connection" body: To characterize connections at the ends of beam elements



The "models"

On the basis of the structural "geometric model", it becomes possible to set forth several "models". A "model" contains the set of information that allows conducting an "analysis":

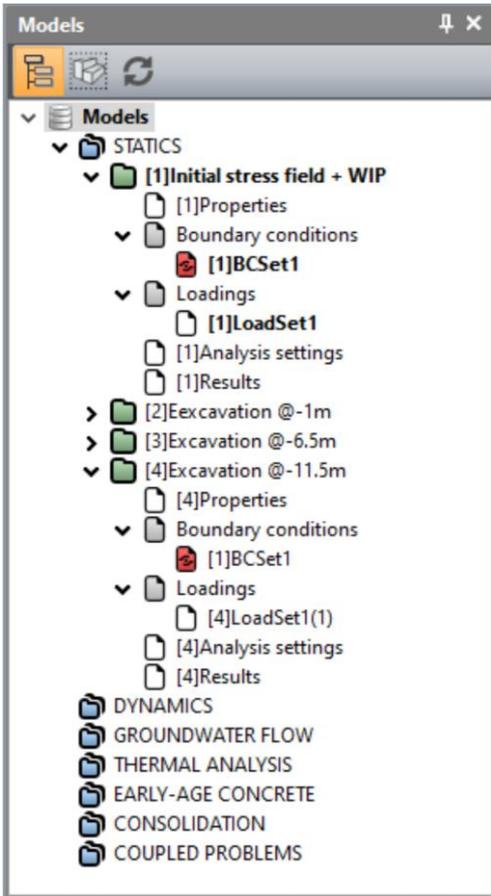
- Properties (materials, geometric properties, etc.)
- Support conditions
- Load conditions

Each model will be categorized into one of the following "domains":

- Statics
- Dynamics
- Hydrogeology
- Heat exchange
- Early-age concrete
- Consolidation
- Coupled problem

The models are created through use of the tree diagram displayed in the right-hand part of the user interface. By right-clicking the mouse over one of the proposed application domains, a drop-down menu opens offering "Add a model" as a possible option.

For purposes of illustration, the following figure shows the "Models" tree diagram after creating four models (four stages) within the statics domain and another model within the hydrogeology domain.



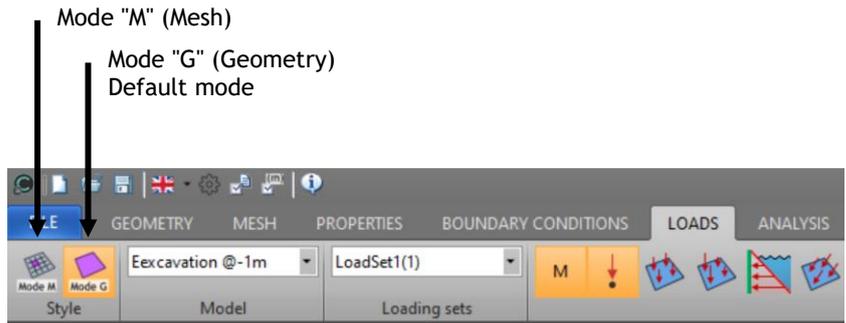
A more detailed analysis of this model creation procedure will be provided in a subsequent chapter on "Model definition".

Mode "G" (Geometry) vs. mode "M" (Mesh)

As a default (Mode "G"), the set of "properties" characterizing a model are assigned on entities belonging to the geometric model and are thus completely independent of the mesh associated with this model. A pressure can then be applied, for example, onto a surface of the geometric model and not onto the plane surfaces of a mesh. Since the assigned properties are independent of the mesh, they will be conserved regardless of the mesh modifications introduced. This feature constitutes the primary advantage of the mode referred to herein as "G".

In some cases, it might still be worthwhile however to be able to assign properties on mesh elements (plane surface, node, etc.). To satisfy these specific needs, CESAR 3D offers the user the possibility of working in mode "M" (for Mesh).

The transition from one mode to the other is performed via the two buttons shown below (example drawn from the "Loads" toolbar):

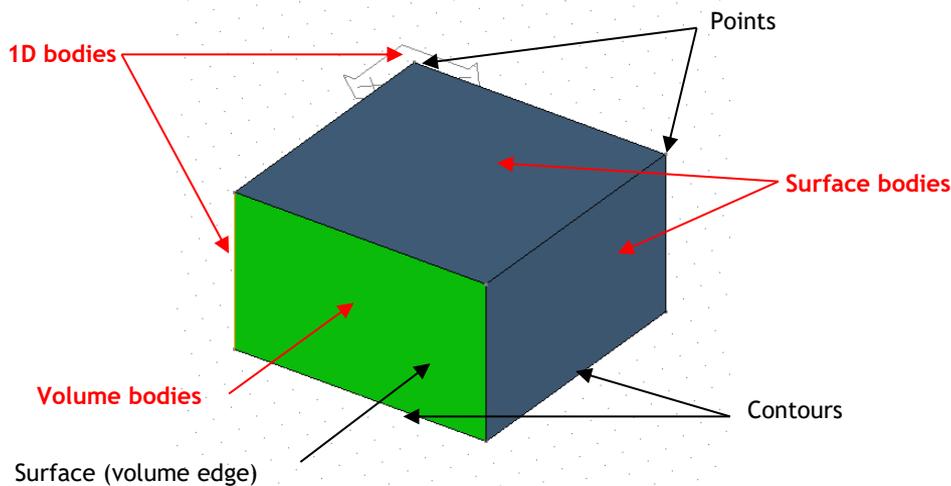


<p>Mode "G": Application of a pressure onto the outer surface of a volume (independent of the mesh).</p>	<p>Mode "M": Application of a pressure onto the plane surfaces of the volume mesh elements. The applied pressures will be deleted should the mesh be modified.</p>

4. Selections

Various entities may be selected depending on the progress made towards model generation. The user needs to be able to accurately select one entity versus another since some functionalities require selection of a specific entity (e.g. extrusion requires selecting a "surface" as opposed to a "surface body").

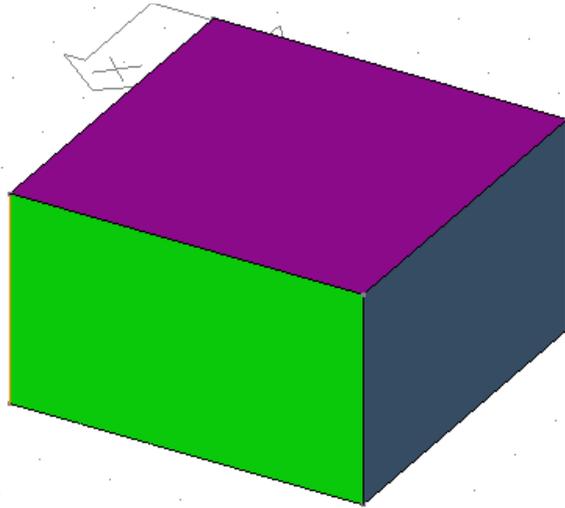
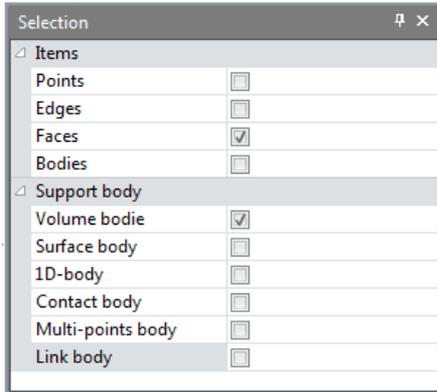
The previous section indicated the notation conventions employed, and the following example will provide a recap:



According to the general selection guidelines, the selectable elements ("Entities") are to be defined on the support bodies.

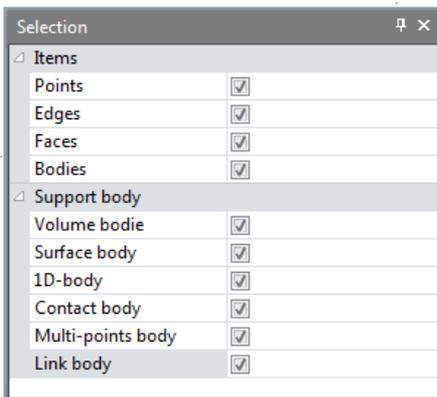
- Selections of entities
 - Selection option settings
 - No entity may be selected
 - Activation of the point selection attribute
 - Activation of the line selection attribute
 - Activation of the surface selection attribute
 - Activation of the "body" selection attribute
- Selections of bodies
 - Selection/activation of volume bodies
 - Selection/activation of surface bodies
 - Selection/activation of 1D bodies
 - Selection/activation of interface bodies
 - Selection/activation of user defined bodies
 - Selection/activation of link bodies
 - No selection/activation of bodies

As an example, to select a face on a volume body, the user will have needed to select "Faces" () in the "Entities" section and then "Volume bodies" () in "Support body", as indicated below.

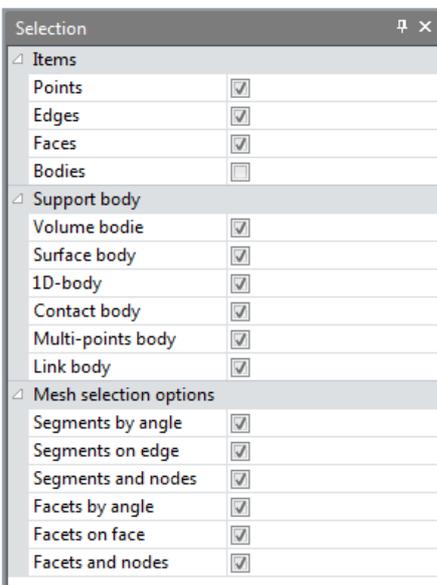


Possible selections

for "Geometry", "Mesh", "Properties", "Initializations"

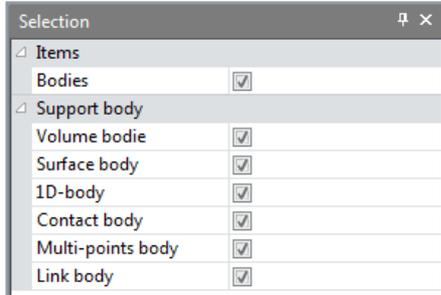


for "Boundary conditions", "Loadings"

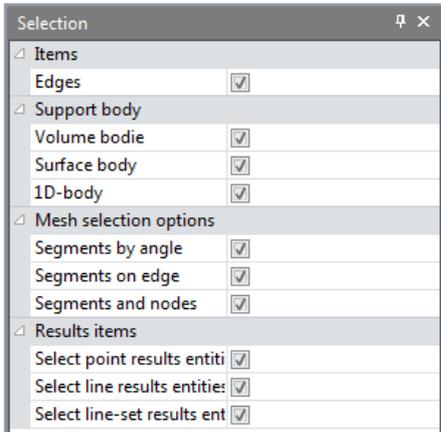


 The selection option "by angle" allows the user to select a set of entities matching the selection angle criterion, which is defined under "Preferences > Accuracy".

for "Results"



for "Charts"



Selection actions

Direct selection

Activating this option makes it possible to select model elements (points, parts). The selection step is carried out either by a single left-mouse click on the target entity or else by "drawing a region".

To add one or more entities to the list of selected entities, hold down the SHIFT [⇧] button on the keyboard during the selection.

With the "Selection" option activated, deselecting all entities requires clicking far away from any potentially selectable entity.

To "deselect" an entity, click on the specific entity while holding down the SHIFT button on the keyboard.

Selection by color

Activating this function triggers display of the available color palette. Choosing a single color automatically selects the set of entities with this same color as a property.

Selection by broken line

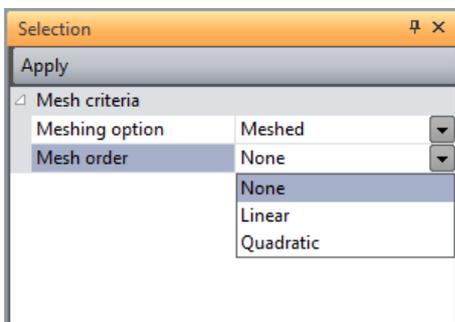
Activating this option serves to identify using the mouse a selection lying within a region delimited by successive clicks.

Selection by criterion

This option is only available at the "Mesh" stage.

It allows the user to indicate selection settings according to the 2 following options:

- Bodies
 - o Meshed
 - o Non-meshed
 - o No criterion
- Type of mesh
 - o Quadratic
 - o Linear
 - o No criterion



5. Definition of model geometry



Working plane definition

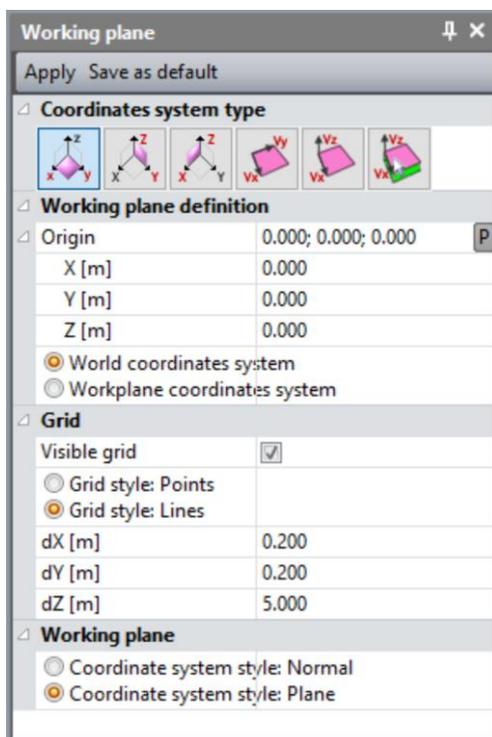
Defining a geometry requires the existence of a "current working plane". The XY horizontal plane is set as the default working plane. This plane is introduced in particular to execute the following operations:

- Direct definition of points using the mouse. Points created directly with the mouse (for the operations "lines", "arcs", etc.) are as a prerequisite contained in the working plane.
- Definition of circle arcs. The arc is contained in either the working plane or a plane parallel to it.
- If applicable, definition of the current working coordinate system used to identify point coordinates (X', Y' of the current coordinate system contained in the working plane).

The "Working plane definition" tool thus enables defining the following elements:

- The current working plane
- The characteristics of the current global coordinate system
- If applicable, the characteristics of a grid of points represented in the working plane and serving to assist in building the model. If such a grid is visible, its component points offer preferred benchmarks during the interactive line construction step.

Activating this tool displays the dialogue box shown below.



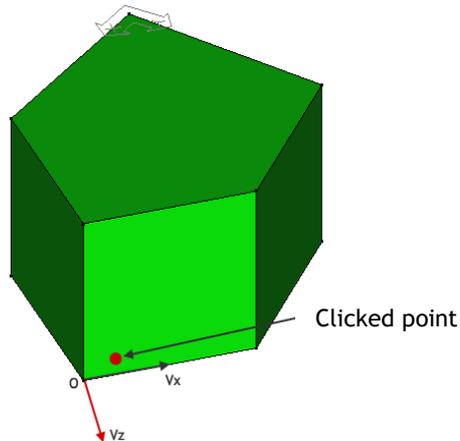
This dialogue box offers the user the six following possibilities for defining the working plane:

- Horizontal plane XY
- Vertical plane YZ

- Vertical plane XZ
- Any working plane defined by two vectors V_x and V_y
- Any working plane defined by two vectors V_x and V_z
- A working plane defined by a surface

Defining the working plane and grid characteristics

1. Choose the type of current working plane from among the six types proposed in the dialogue box.
2. If the current working plane is of the type XY, XZ, YZ, then identify the origin point of the current working coordinate system.
3. If the current working plane is of the type "Vectors V_x , V_y ", then define successively: i) the origin point of the current working coordinate system, ii) the vector V_x used to identify the X' axis of the working plane, and iii) the vector V_y used to identify the Y' axis of the working plane.
4. If the current working plane is of the type "Vectors V_x , V_z ", then define successively: i) the origin point of the current working coordinate system, ii) the vector V_x used to identify the X' axis of the working plane, and iii) the vector V_z used to identify the Z' axis of the working plane.
5. If the current working plane is of the type "Surface", then simply click on a point of the given surface. The origin will be identified by the coordinates of the surface point closest to the clicked spot. The vector V_x will be identified by the tangent to the origin of the line included in the surface contour closest to the clicked point. The vector V_z will be identified by the normal to the surface at the origin.



6. Check the box labeled "Working plane system" if it is preferred to use the current working coordinate system to **define coordinates**. If the box "Global coordinate system" is checked instead, the coordinates will continue to be defined in the global coordinate system regardless of the characteristics of the current working coordinate system. The data input will still be used however to define the "working plane".
7. Check the "Visible grid" box if it is preferred to use the assistance function offered by points on the grid.
8. Specify the steps dX and dY representing the distances between two lines of consecutive points in both the X and Y directions.
9. Click on the "Apply" button to record the settings defined in this sequence.



The software will automatically name the coordinate system once the vectors V_i have been defined.



Assigning points

This tool basically serves to generate points by means of keyboard-entered coordinate data.



These coordinates are to be provided in the "current reference" (see: Definition of the working plane).

Creating points through coordinate data entry

1. Enter the coordinates of the first point to be created.
2. Click the "Apply" button to create the point. All points created are represented on the screen by a spherical symbol. The point will only be created if the coordinates entered do not lie near those of a preexisting point (see: Proximity).
3. Repeat steps 1 and 2 for the entire set of points to be defined.



To move from one control parameter to the next within the dialog box, it is possible to use the "Tab" key.

Using the "Linked points" option to create line segments

The "Linked points" option enables creating line segments that connect the sequence of points defined using the dialog box.

1. Establish the coordinates of the "starting point". The "Linked points" option can only be activated once the starting point of the segment to be created has been defined. As a default, the starting point will correspond to the last point entered by clicking the "Apply" button.
2. Check the "Linked points" box.
3. Specify the coordinates of the "ending point".
4. Now click the "Apply" button to create both the ending point and the corresponding segment. This newly-created point automatically becomes the starting point of the next segment.

Specifying coordinates in relative mode

The "Relative" option serves to define the coordinates of a point by means of inputting its relative coordinates with respect to a reference point.

1. Indicate the coordinates of the reference point. The "Relative" option may be activated once a reference point has been entered. As a default, the "reference point" corresponds to the last point created using the "Apply" button.
2. Tick the box marked "Relative".
3. Input the relative coordinates of the new point to be created.
4. Click the "Apply" button to generate the point that will automatically become the new reference point.

Identifying the coordinates of a given point

1. Clicking the right mouse button when the cursor is positioned on a previously-specified point generates a dialog box containing the coordinates of the given point. These coordinates are displayed in both the currently-defined reference and coordinate system.



Lines

This tool allows creating broken lines using the mouse.

Creating a line as link of two points

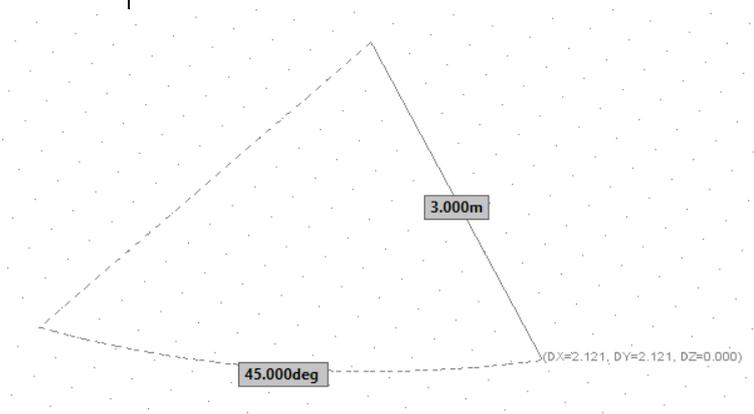
2. Set the starting point with the left mouse button. If the selected point lies close to an existing point, the existing point will be considered as the starting point. Otherwise, a point is created in the working plane at the selected position.
3. Define the ending point of the segment in an analogous fashion. Since this creation procedure is continuous, the ending point automatically becomes the starting point of the next segment.
4. Halt the segment-creation procedure. This procedure for creating line segments may be halted by one of the three following methods:
5. - Double-click on an ending point;
6. - Press the "Esc" key on the keyboard;
7. - Activate another tool or menu item.
- 8.



If the selected point is not close to an existing point and if the "Magnetic grid" option has been activated (see: Grid option in the Working plane definition), the point created in the working plane will automatically be a point on the grid.

Interactive creating of a line

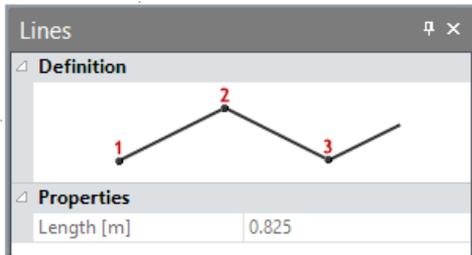
1. Set the starting point with the left mouse button. If the selected point lies close to an existing point, the existing point will be snapped and considered as the starting point. Otherwise, a point is created in the working plane at the selected position.
2. Move the mouse for a second point definition, a segment and an arc are displayed with segment dimension and angle orientation as labels.
3. End the segment :
 - Method #1: classically, click on a point of the working place
 - Method #2: specify a length and an angle in the labels. Validate the values using [Tab] (labels become grey). Press [Enter] for creating the segment.



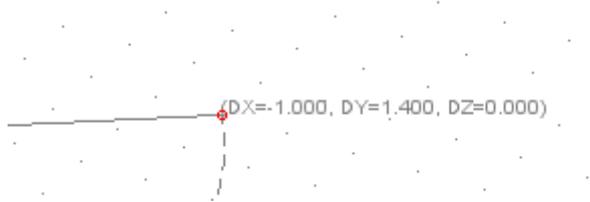
Interactive information

When moving the mouse, information will be displayed interactively:

- Inside the grid, the segment length,



- On the work space, the relative displacement along the Ox and Oy axes.



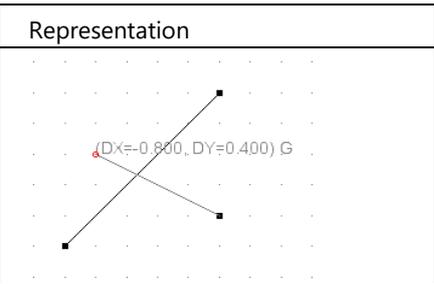
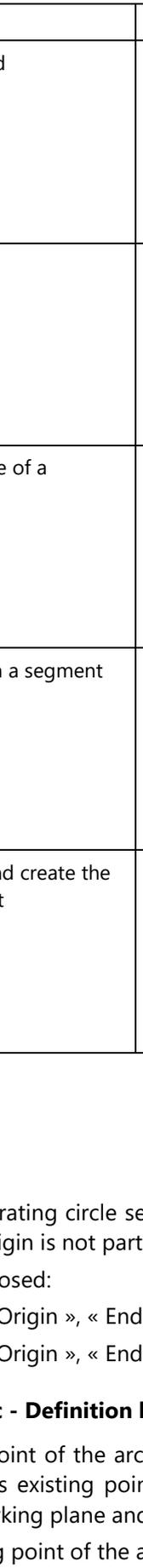
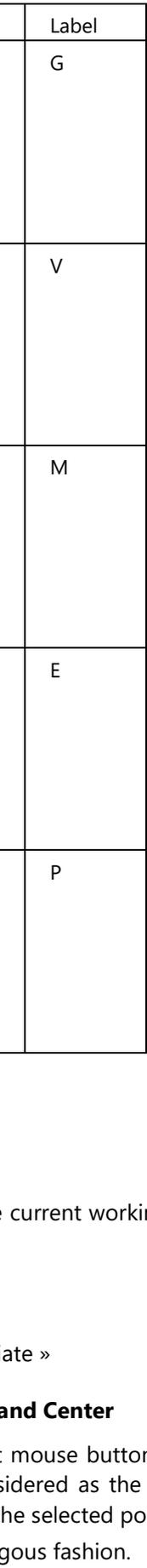
Interactive "snapping" of the departure point

When running the tool, specifically for identifying the first segment point, the cursor moved by the user will activate a symbol on the work space for all actions listed in the following table.

Action	Symbol	Representation
Snapping onto the grid	red circle	
Snapping on a point	red rectangle around the point	
Snapping in the middle of a segment	red cross	
Snapping anywhere on a segment	red cross	

Interactive "snapping" of the arrival point

Once the departure point has been established, the user will define the segment arrival point by means of moving the cursor on the work space, which will then display a symbol for the snapping actions listed in the table below. Additional information presented as labels will also indicate the type of snapping action in specific cases.

Action	Symbol	Label	Representation
Snapping onto the grid	red circle	G	
Snapping on a point	red rectangle	V	
Snapping at the middle of a segment	red cross	M	
Snapping anywhere on a segment	red cross	E	
Snapping to a point and create the perpendicular segment	croix rouge	P	



Circular arcs

This tool enables generating circle segments in the current working plane or in a plane parallel to the working plane if the origin is not part of it.

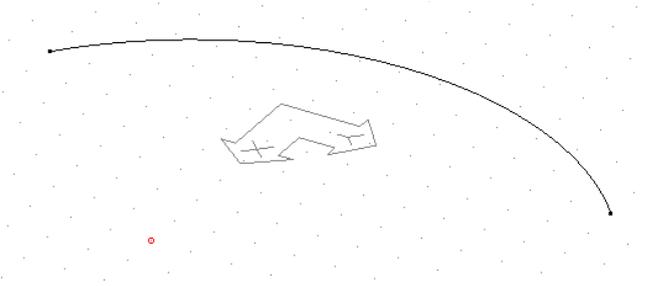
Two methods are proposed:

- Definition by « Origin », « End », « Center »
- Definition by « Origin », « End », « Intermediate »

Creating a circular arc - Definition by Start, End and Center

1. Set the starting point of the arc using the left mouse button. If the selected point is close to an existing point, this existing point will be considered as the starting point. Otherwise, a point is created in the working plane and assigned to the selected position.
2. Specify the ending point of the arc in an analogous fashion.
3. Define the position of the circle center using the left mouse button.

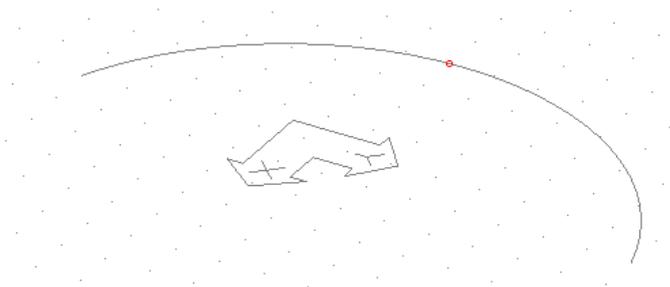
4. This arc-creation procedure may be halted by pressing the "Esc" key on the keyboard or any item of the Menu toolbar.



Creating a circular arc - Definition by Start, End and Intermediate

1. Set the starting point of the arc using the left mouse button. If the selected point is close to an existing point, this existing point will be considered as the starting point. Otherwise, a point is created in the working plane and assigned to the selected position.
2. Specify the ending point of the arc in an analogous fashion.
3. Specify the intermediate point of the arc in an analogous fashion.

i The radius of the circular arc under construction is displayed dynamically in the grid.



Creating a full circle

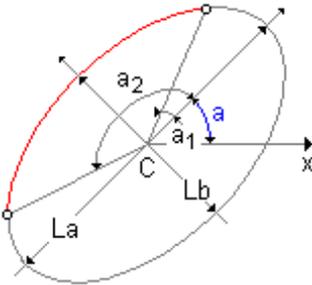
Full circle is generated when Start and End points overlap.

1. Set the starting point and the ending point on the same point.
2. Define the position of the circle center using the left mouse button.



Elliptical arcs

Activating this tool generates a dialog box that enables creating elliptical arcs located in a plane parallel to the current working plane.



An elliptical arc is characterized by the following data:

- X_c, Y_c, Z_c : Coordinates of the ellipse center
- L_a : Length of the first ellipse diameter
- L_b : Length of the second ellipse diameter
- a : Angle between the x -axis of the working plane reference and the first ellipse diameter
- a_1, a_2 : Angles specifying the starting and ending points of the elliptical arc. If these two angle values are identical, the entire ellipse will be generated. A characteristic point will then be positioned at the defined angle.

Creating an elliptical arc

1. Specify the characteristics of the ellipse defined above.
2. Use the "Apply" button in order to create the given ellipse.

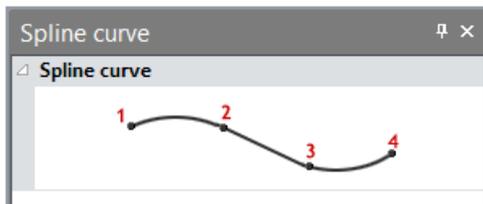


The elliptical arc is created in the plane running parallel to the working plane and passing through the center C.



Spline curves

Activating this tool serves to create cubic spline curves characterized by N points. The spline curve is defined interactively by sequentially pointing on the N component points using the left mouse button. The last component point in the spline curve definition is indicated by a double-click.



Creating a spline curve

1. Specify the sequence of points characterizing the spline curve using the left mouse button. If the selected point lies close to an existing point, this existing point will be included. Otherwise, a point is created in the working plane at the selected position.
2. Complete the spline curve definition by double-clicking on the last point.



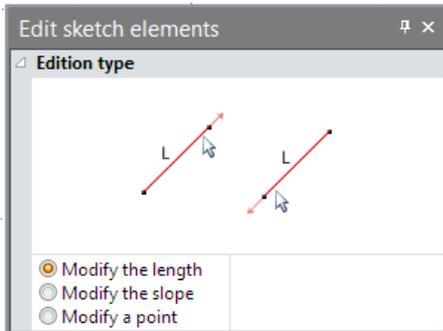
This spline curve-creation procedure may be halted by pressing the "Esc" key on the keyboard.



Information and editing of the length and angle of a line

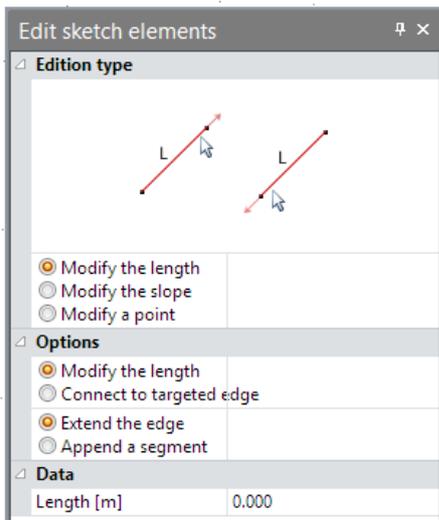
This tool serves to modify the length or angle of a line segment with respect to a horizontal plane or to modify the coordinates of a point on the sketch.

Activating this tool displays the grid dialogue box shown below.



Edit the length of a line

Several options are available.



1st method: the only line is considered.

1. (Optional) Right-click on the line segment whose length is to be modified. The length of this segment is displayed in the dialogue box.
2. Tick "Modify the length".
3. Tick the option "Extend the edge".
4. Now set the new length of the given edge.
5. Click on the line whose length is to be modified on the **side of the point intended to be moved**.

In order to modify the length, a segment can also be appended:

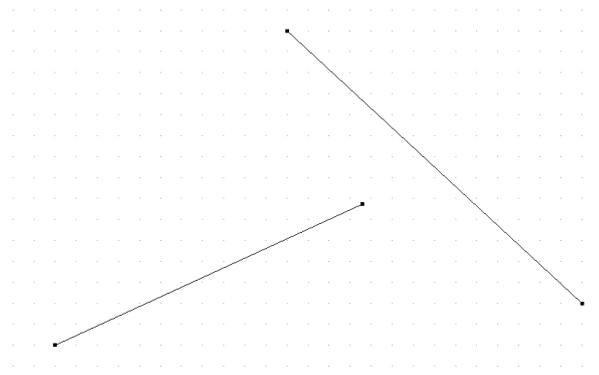
6. Tick "Modify the length".
7. Tick the option "Append a segment".
8. Now set the new length of the given edge.
9. Click on the line whose length is to be modified on the **side of the point intended to be moved**.

2nde method: the line is considered for connexion to another line.

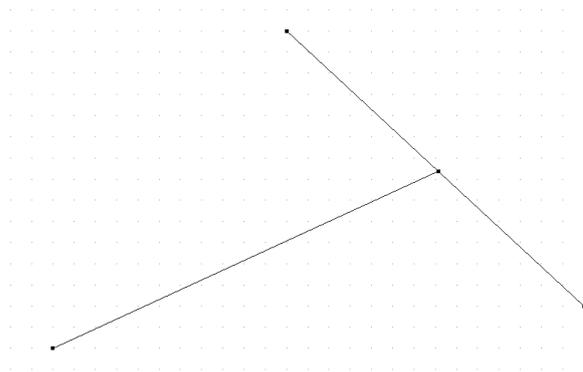
1. Tick "Connect to the targeted edge". In this case, no length has to be specified.
2. Tick the option "Extend the edge".
3. Click on the edge to be modified and next click on the one to be linked to.

For this operation, a segment can also be appended:

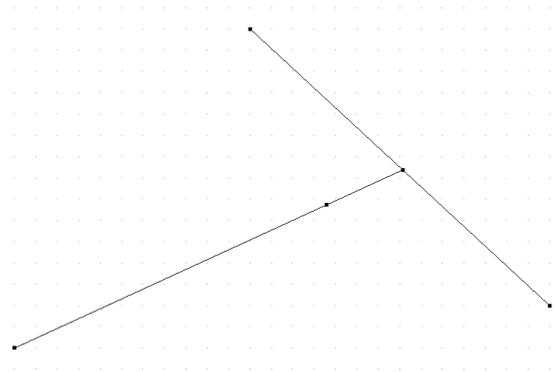
4. Tick "Connect to the targeted edge".
5. Tick the option "Append a segment".
6. Click on the edge to be modified and next click on the one to be linked to.



Initial sketch

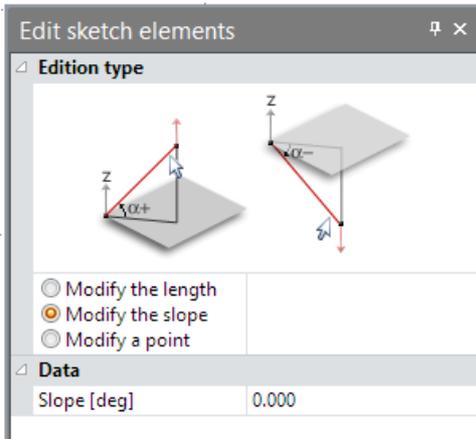


Extension of the line to the targeted one



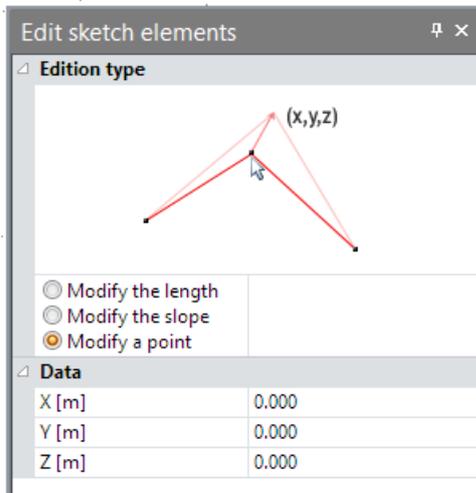
Extension of the line with addition of a segment

Edit the slope of a line



1. (Optional) Right-click on the line segment whose slope is to be modified. The length and slope of this segment with respect to a horizontal plane are displayed in the dialogue box.
2. Click on the "Slope" option.
3. Set the new slope of the given line.
4. Click on the line whose slope is to be modified on the side of the point intended to be moved.

Edit the coordinates of a point



1. (Optional) Right-click on the point whose coordinates have to be modified. These coordinates are displayed in the dialogue box.
2. Click on the "Modify a point" option.
3. Set the new coordinates.
4. Click on the point to be modified.

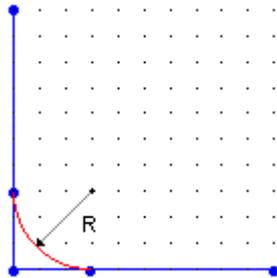


If the modified point is linked to lines or surfaces, then these elements are also automatically modified.



Shoulder fillets

This tool serves to define shoulder fillets between concurrent line segments.



Defining a shoulder fillet

1. Set the radius R of the shoulder fillet in the dialog box.
2. Click on the first of the two concurrent line segments.
3. Click next on the second segment. It should be noted that the plot of this shoulder fillet can be visualized dynamically on the screen during the creation procedure whenever the mouse cursor moves near an admissible line segment.



Line intersection

This tool serves to identify the intersection point or points between any two lines (line segment, arc of a circle, ellipse, spline, etc.).

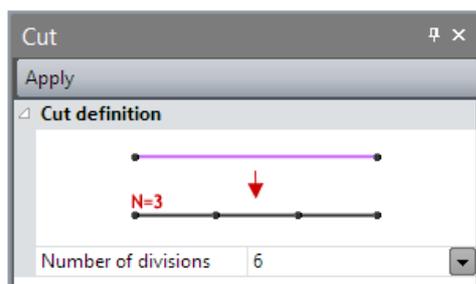
Defining the intersection between two lines

1. Activate the "Line intersection" tool.
2. Select whether it is preferred to only cut the first line (object) or else both lines (object and "tool") in using the proposed option.
3. Left-click on the first of the two lines (object).
4. Next, click on the second line (tool). The intersection point is generated.



Line cut

This tool serves to cut the selected lin(s) in several segments of same length.



Cut a line in several segments

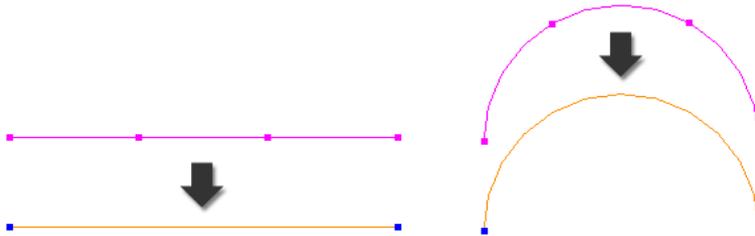
1. Select the lines to be modified.

2. Define the number of divisions in the combo list.
 3. « Apply » to get the number of segments.
-  Use keys [Ctrl] or [Shift] for a multi-selection.



Simplification of edges

This tool serves to delete intermediate points on the set of selected edges having the same mathematical definition curve (line segment, arc circle or spline).



Simplifying

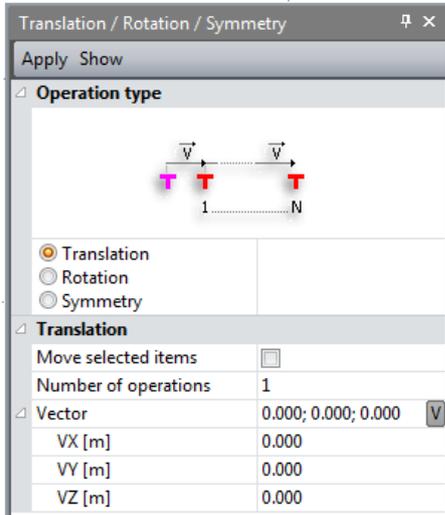
1. Activate the tool « Simplification of edges ».
2. Select the edges to « simplify ».
3. Click « Apply ».



Translation, Symmetry, Rotation

This tool serves to duplicate, translate or rotate geometrical entities or bodies.

Translation



Duplicating points and boundaries via translation

1. Select the set of points and boundaries to be duplicated by translation.
2. Activate the "Translation" tool.
3. Verify that the box labeled "Move selected items" has not been ticked.
4. Specify the number of translation operations by inputting a value in cell "Number of operations".
5. Define the translation vector components within the current reference working plane.
6. Use the "Apply" button to perform the translation operation.



User may collect the translation vector components following these operations : i) click the « **V** » button located on the « Vector » line ; ii) using left mouse button, click on a point as starting point of the vector ; iii) next, click on a point as end of the translation vector.

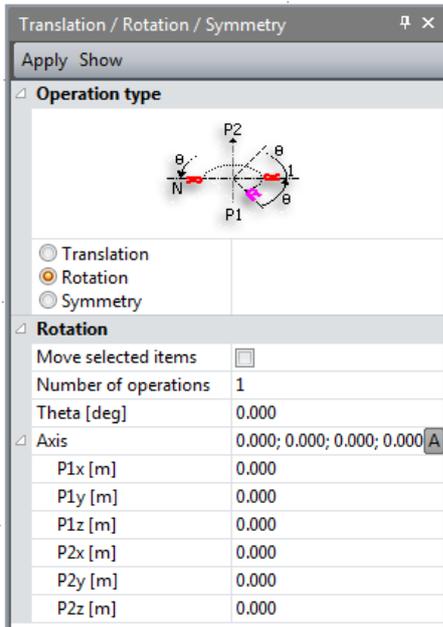
Moving points and boundaries via translation

1. Select the set of points and boundaries to be moved.
2. Activate the "Translation" tool.
3. Tick the box labeled "Move selected items ".
4. Define the translation vector components within the current reference (see: Coordinate system - Working plane).
5. Use the "Apply" button to perform the translation operation.



The boundaries may be selected in any order either before or after activating the "Translation" tool.

Rotation



Duplicating points and boundaries via rotation

1. Select the set of points and boundaries to be duplicated by rotation.
2. Activate the "Rotation" tool.
3. Verify that the box labeled "Move selected items" has not been ticked.
4. Specify the angle characterizing each rotation operation by inputting a value for "Theta" in the appropriate cell.
5. Indicate the number of rotation operations by inputting a value in the cell marked "Number of operations".
6. Define the current reference coordinates of points 1 and 2 that serve to identify the axis of rotation.
7. Use the "Apply" button to perform the rotation operation.

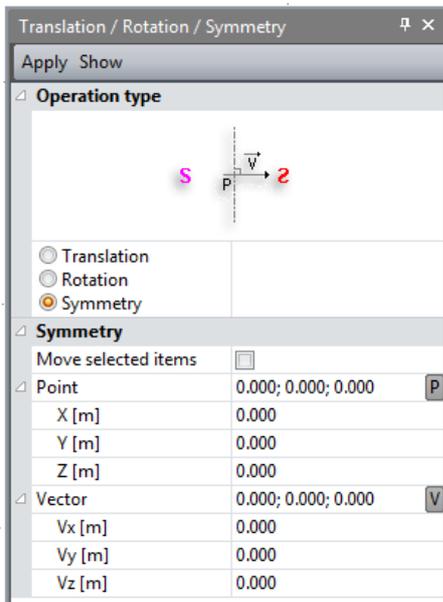
Moving points and boundaries via rotation

1. Select the set of points and boundaries to be moved by rotation.
2. Activate the "Rotation" tool.
3. Define the current reference coordinates of points 1 and 2 that serve to identify the axis of rotation.
4. Specify the angle characterizing the particular rotation operation by inputting a value in the "Theta" cell.
5. Tick the box labeled "Move selected items".
6. Use the "Apply" button to perform the rotation operation.

i The coordinates of an existing point may be "recovered" according to the following procedure carried out using the right mouse button: i) Position the cursor on the dialog box in either the X, Y or Z field of the point whose coordinates are to be recovered; ii) Right click on the given point of the model.

i Boundaries may be selected in any order either before or after activating the "Rotation" tool.

Symmetry



Duplicating points and boundaries via symmetry

1. Select the set of points and boundaries to be duplicated by symmetry.
2. Activate the "Symmetry" tool.
3. Verify that the box labeled "Move selected items" has not been ticked.
4. Specify the coordinates of a point lying in the plane of symmetry within the current reference.
5. Specify the components of a vector perpendicular to the plane of symmetry within the current reference.
6. Use the "Apply" button to perform the symmetry operation.

Moving points and boundaries via symmetry

1. Select the set of points and boundaries to be moved by symmetry.
2. Activate the "Symmetry" tool.
3. Tick the box labeled "Move selected items".
4. Specify the coordinates of a point lying in the plane of symmetry within the current reference. The coordinates of this point may be "recovered" by right clicking on a point of the model contained in the plane of symmetry.
5. Specify the components of a vector perpendicular to the plane of symmetry within the current reference.
6. Use the "Apply" button to perform the symmetry operation.



The boundaries may be selected in any order either before or after activating the "Symmetry" tool.



User may collect the vector components following these operations : i) click the « **V** » button located on the « Vector » line ; ii) using left mouse button, click on a point as starting point of the vector ; iii) next, click on a point as end of the translation vector.

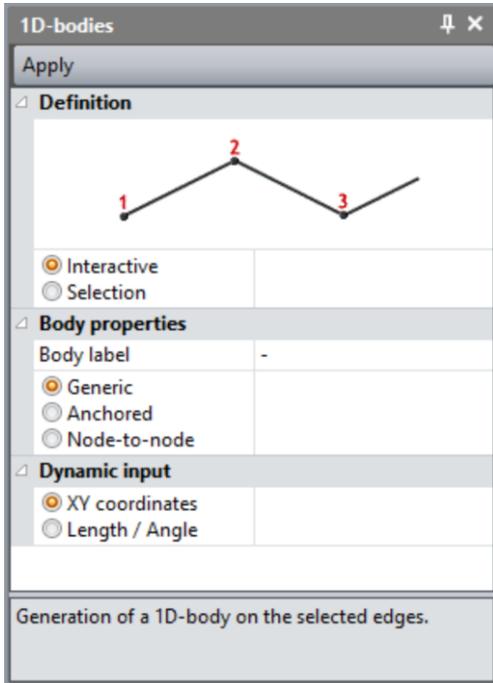


User may collect the coordinates of the point "**P**" may be "recovered" by right clicking on a point of the model contained in the plane of symmetry.



Creation of "1D bodies"

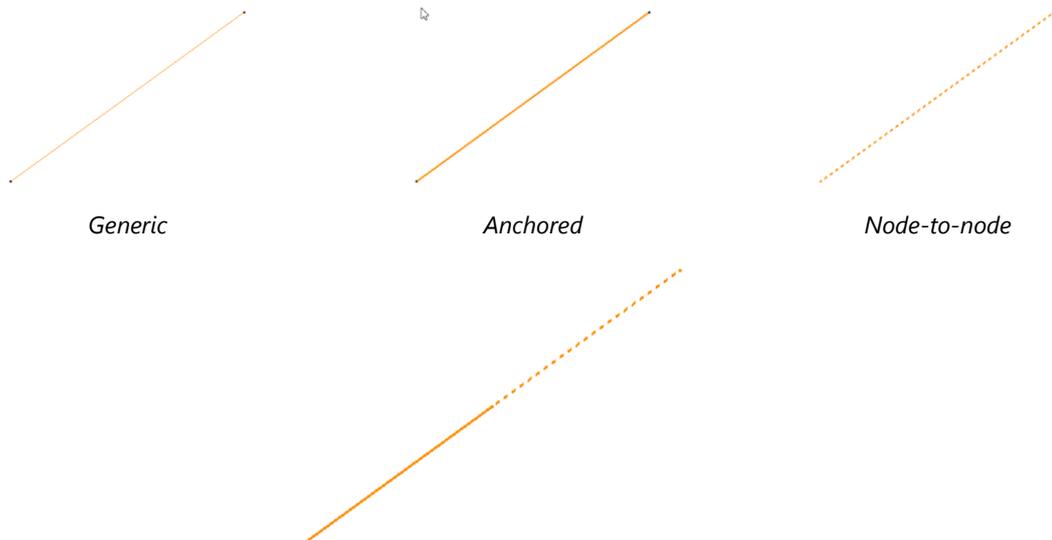
This tool serves to create a 1D body from the set of selected lines. Let's note that this operation is only possible provided none of the selected lines already belong to a "1D body".



3 types of 1D bodies are proposed:

- "Generic"; generic 1D body for the modelling of bar or beam elements. This body has no particular connexion with surrounding mesh. It is supported by the sketch.
- "Anchored"; if the 1D body is immersed in one or several volume bodies, when meshed, its nodes will be connected to those of the volume bodies. In other words, the volume mesher will be constrained to take into account the anchored 1D body.
- "Node-to-node"; if the 1D body is inside one or several volume bodies, only its ends will be detected by the volume mesher.

Each type of 1D body has its own graphical display.



Example of use for a ground anchor: bond length (anchored 1D body) and free length (node-to-node 1D body)

Creating a 1D bodies

1. Select the lines composing the 1D body.
2. Assign the body name in the proposed grid.
3. Click on the "Apply" button to create the 1D body.



The body created now appears in the data tree diagram under a generic name generated by the software (Bodies).

Dynamic creation of a 1D body

1. Define the starting point with the mouse left-click button. If the specified point is in the vicinity of a point of the grid or an existing point, this point will be considered as the starting point. If not, a new point is created.
2. Select a dynamic method:
 - Coordinates XY
 - Length/Angle
3. Move the mouse to define the second point of the segment:
 - If "Coordinates XY", a line and an arch are displayed with labels for the relative displacements in ΔX and ΔY in the working plane.
 - If " Length/Angle ", a line and an arch are displayed with labels for the length and the angle.

Ending the 1D body creation

1. Method 1: classic, click anywhere in the working space.
2. Method 2: interactive, specify the length and the angle in the labels. Validate these values using [Tab] key (labels get grey colour). Press [Enter] key to create the segment.

This tool serves to create a 1D body from the set of selected lines. Let's note that this operation is only possible provided none of the selected lines already belong to a "1D body".

Creating a 1D bodies

1. Select the lines composing the 1D body.
2. Assign the body name in the proposed grid.
3. Click on the "Apply" button to create the 1D body.



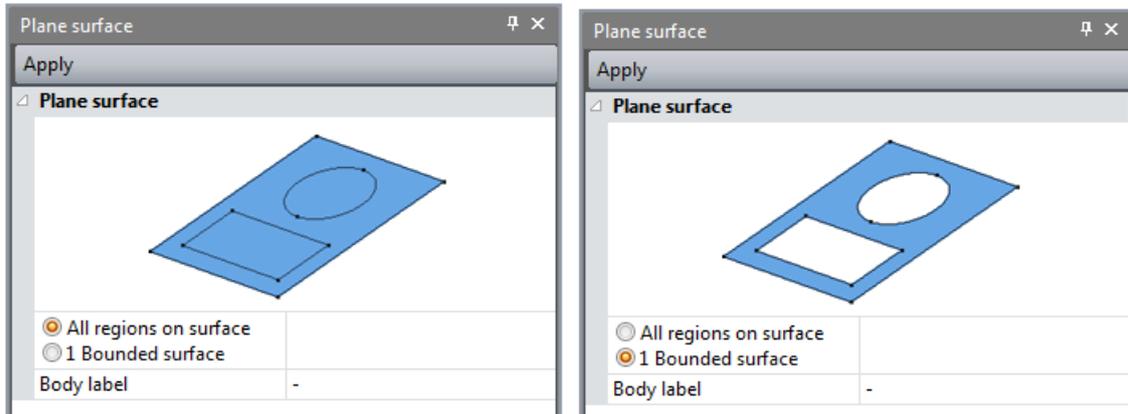
The body created now appears in the data tree diagram under a generic name generated by the software (Bodies).



Creation of "surface " bodies

This tool serves to create a "surface plane" type of surface body from the set of selected lines characterizing both the outer and "inner" contours.

Activating this tool displays a dialogue box that allows choosing between the two main options as follows:



- Definition of a body using just the "external" surface region,
- Definition of a body composed of all "external" and "internal" surface regions.

Defining a " plane surface " type of surface body

1. Select the lines making up the outer and inner contours of the surface regions composing the surface body.
2. Select the appropriate generation mode (i.e. external regions or all regions).
3. Assign the body name in the proposed grid.
4. Click on the "Apply" button to generate the surface body containing the newly created surface regions.

i The surface body will not be created if it is impossible to indicate a closed outer contour and closed inner contours with the selected lines.

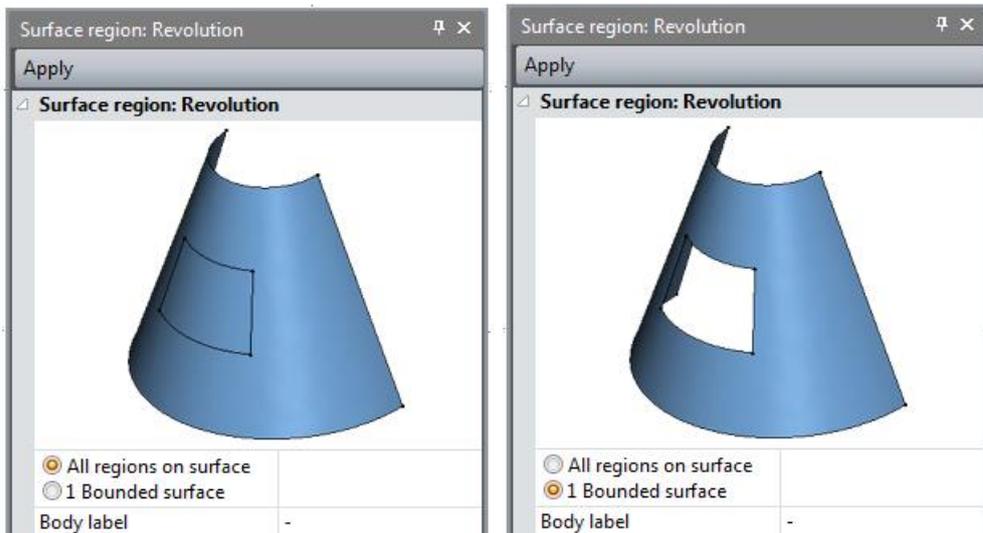


Rotating surface body

This tool allows defining a surface body of the "rotating" type by means of the set of lines characterizing the exterior contour and interior contours.

Activation of this tool displays a toolbox proposing a choice between two main following options:

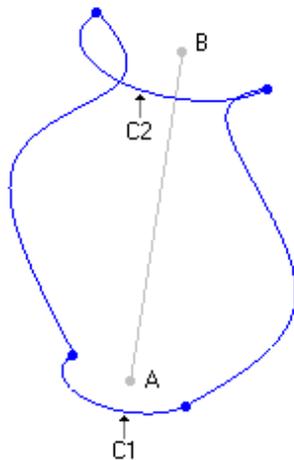
- Definition of a model part with external surface region only,
- Definition of a model part with all surface regions.



On the basis of the inputted selected lines, the software seeks to automatically determine the rotating surface on which the given region actually lies.

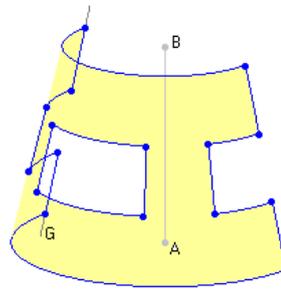
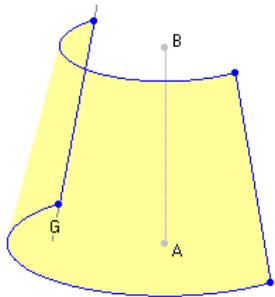
i This automated determination process can only be conducted provided the following conditions have been satisfied:

- **Condition 1:** Within the set of selected boundaries, at least one circular arc exists whose axis overlaps with that of the rotating surface. This condition, which is satisfied in a large majority of cases, relieves the user from having to supply additional information for defining this axis. In the example below, circular arcs C1 and C2 yield axis AB of the rotating surface.

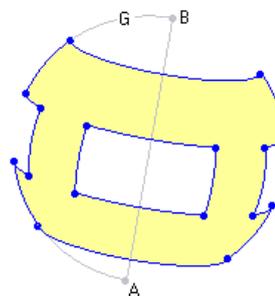
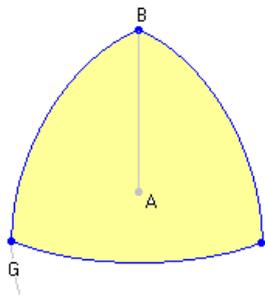


- **Condition 2:** Within the set of selected boundaries, at least one boundary is actually "carried" by a mathematical curve capable of serving as the surface generator element. The mathematical curve considered in this instance must belong to a plane that contains the axis of revolution of the surface.

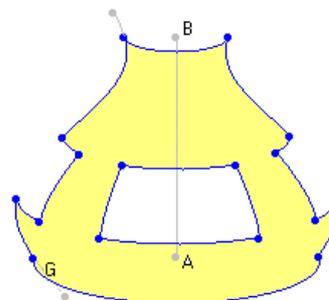
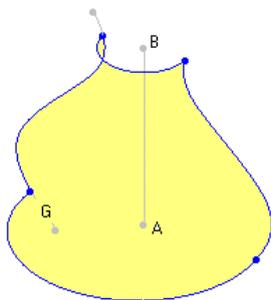
In what follows, we have provided several examples of acceptable rotating surface regions.



Rotating surface regions generated using straight lines

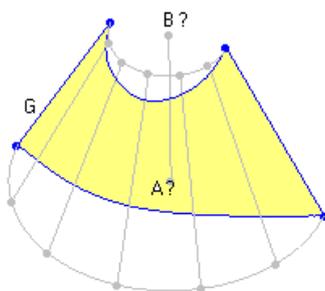


Rotating surface regions generated using circles

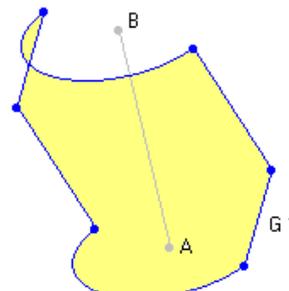


Rotating surface regions generated using spline curves

In contrast, those surface regions visualized in the following figure cannot be incorporated into the "Rotating surface region" tool described herein.



Condition 1 (unverified): The component boundaries do not contain any circle capable of defining the axis of revolution of the surface.



Condition 2 (unverified): It is impossible to establish a generating element with a single mathematical curve

Defining a rotating surface body

1. Select the lines characterizing the exterior and interior contours of the body to be defined.
2. Select one of the generation methods: all regions on surface or 1 bounded region.
3. Give a name to the body.
4. Click on "Apply" to generate the rotating surface body.

i In addition to the two conditions listed above, the surface region will not be created if it proves impossible to define a closed exterior contour and closed interior contours with the selected lines.

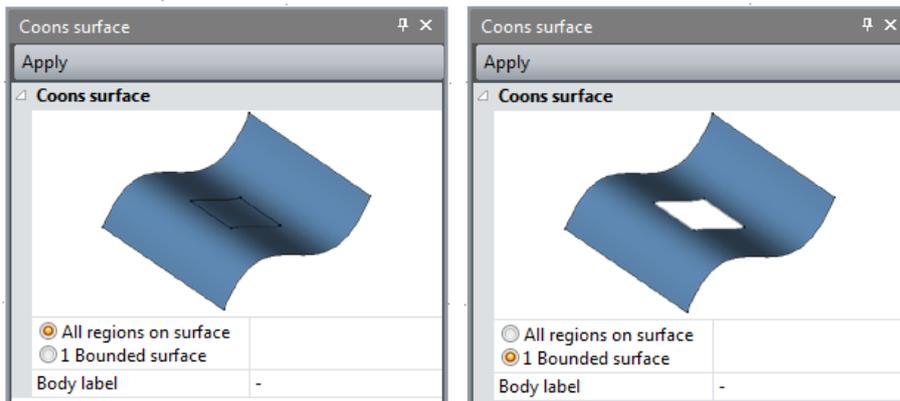


Coons surfaces

This tool allows defining a surface body of the "Coons" type by means of the set of lines characterizing the exterior contour and interior contours.

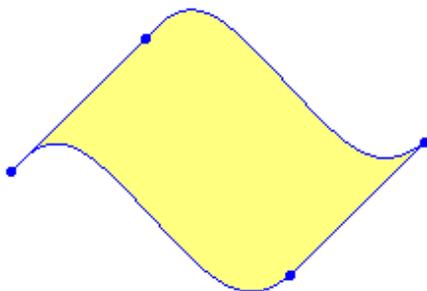
Activation of this tool displays a toolbox proposing a choice between two main following options:

- Definition of a model part with external surface region only,
- Definition of a model part with all surface external and internal regions.

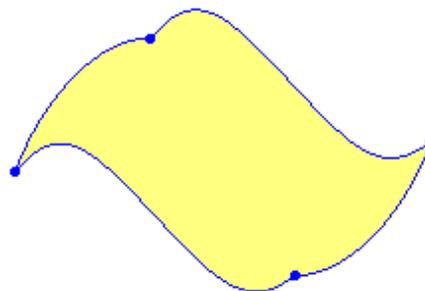


i This surface body will not be created if the selected boundaries do not enable determining an exterior contour composed of three or four boundaries.

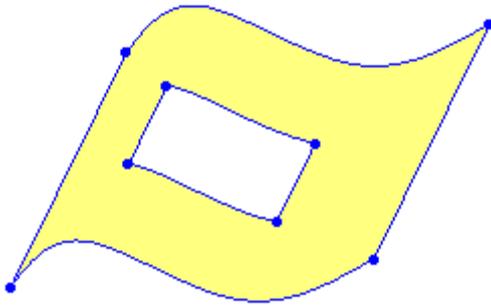
In what follows, we will provide a few examples of acceptable Coon's patch-type surface regions.



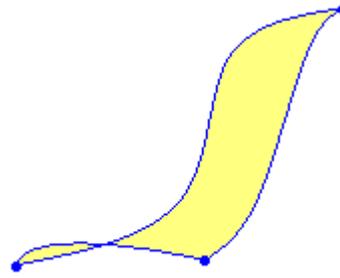
*4-boundary Coon's patch regions
- 2 spline curves and 2 straight lines -
(equivalent here to an adjusted surface)*



*4-boundary Coon's patch regions
- 2 spline curves and 2 circular arcs -*

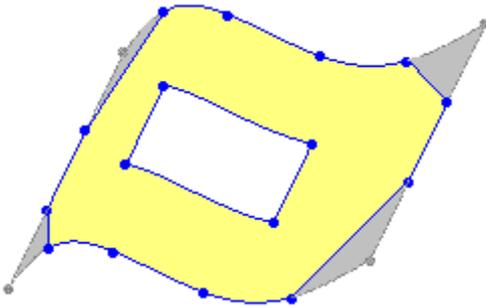


4-boundary Coon's patch regions with an interior contour



*3-boundary Coon's patch regions
- 2 spline curves and 1 circular arc -*

In contrast, the surface regions visualized in the following figure cannot be incorporated into the "Coon's patch-type surface region" tool.



*Condition 1 (unverified):
The exterior contour does not comprise
3 or 4 boundaries*

Defining a Coon's patch-type surface region

1. Select the lines characterizing the exterior and interior contours of the body to be defined.
2. Select one of the generation methods: all regions on surface or 1 bounded region.
3. Give a name to the body.
4. Click on "Apply" to generate the rotating surface body.



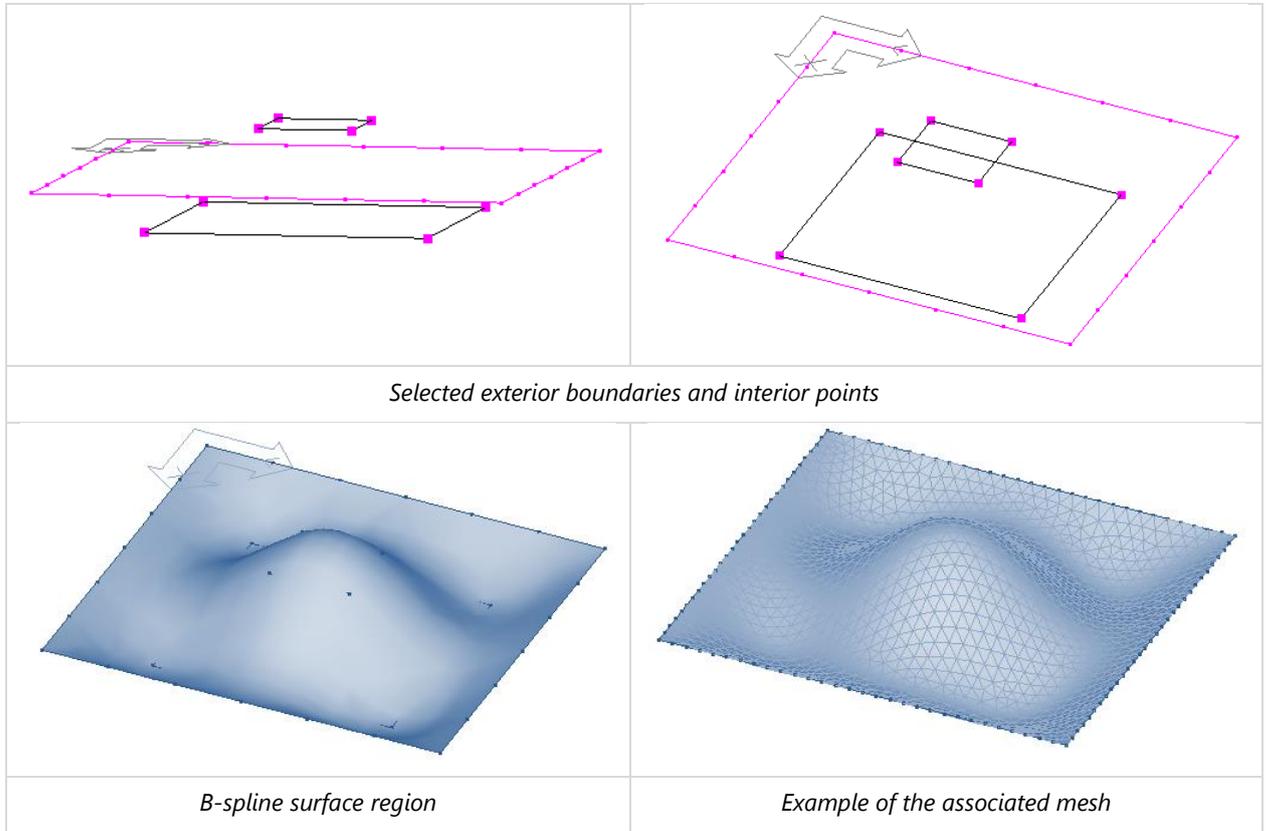
In addition to the conditions listed above, this surface region will not be created in the following cases:

- The selected lines do not all belong to the Coon's patch-type surface defined by the exterior contour;
- It is impossible to define a closed exterior contour and closed interior contours with the selected lines.



Definition of a "B-spline surface" type of surface bodies with imposed interior points

This tool serves to create surface bodies of the "B-spline surface" type with imposed interior points. Such a surface region is identified by boundaries of any number and type, forming the outer contour and, if applicable, by interior points that provide an equal number of governing points.



This tool may be used, for example, to represent the ground relief in a "large-scale" model (e.g. tunnel behavior).

Defining a "B-spline" type of surface body with imposed interior points

1. Select the lines composing the exterior boundaries and the subsequent interior points of the surface region delineating the surface body.
2. Assign the body name in the proposed grid.
3. Click on the "Apply" button to generate the surface body containing the newly created surface region.

i In order to obtain the B-spline surfaces with sufficient accuracy, it may be critical to define "intermediate points" (e.g. use the "Cutout" option) on the exterior boundaries of the surface region.



Creation of "surface" type model elements from a "volume" type model element

This tool serves to replace a "volume" type surface body by a "surface" type surface body that contains the set of surface regions making up the "outer skin" of the volume.



This operation causes the original volume body to be deleted.

Defining a surface body from a volume body

1. Select the volume body or bodies to be replaced by surface bodies.
2. Assign the surface body name in the proposed grid (e.g. MySurfaceBody). If several volume bodies are selected, this command will trigger the generation of the same number of surface bodies. These newly created surface bodies will automatically be assigned the names: MySurfaceBody, MySurfaceBody (1), MySurfaceBody (2), etc.
3. Click on the "Apply" button to generate the surface body or bodies. These newly created surface bodies are assigned the same default color as that given to the surface regions.



Creation of a surface body from the outer faces of volume bodies

This tool serves to create a surface body from the selected outer faces of one or more volume bodies.

Defining a surface body from the outer faces of volume bodies

1. Select the outer face or faces of the particular volume bodies.
2. Assign the body name in the proposed grid.
3. Click on the "Apply" button to generate the surface body containing the selected surfaces. This newly created surface is assigned the same default color as that ascribed to the surface regions.

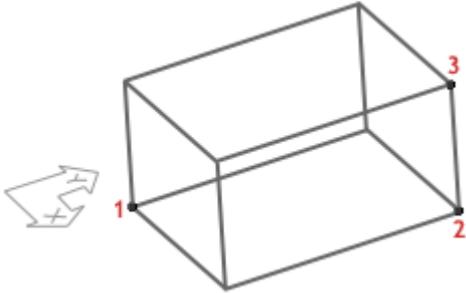


Creation of predefined volume bodies

This tool serves to create volume bodies according to the four following "predefined" types:

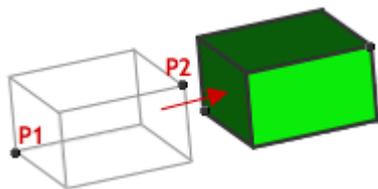
- Rectangular parallelepiped defined by three points
- Rectangular parallelepiped defined by two points (diagonal)
- Rectangular parallelepiped defined by one point and three dimensions
- Cylinder defined by a center and a radius.

Defining a parallelepiped volume body interactively by three points



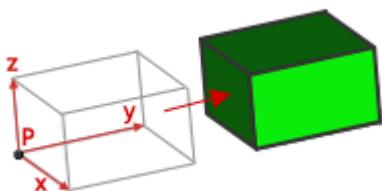
1. Assign the name of the volume body to be created.
2. Choose the "Parallelepiped by three points" option.
3. Click inside the graph zone in order to identify point 1 (either an existing point or nonexistent but lying in the working plane).
4. Click inside the graph zone to identify point 2. The base rectangle is then displayed interactively.
5. Click inside the graph zone again to identify point 3. An alternative solution consists of setting the height H of the volume in the dialogue box and then activating the "Apply" button. The newly defined volume body has now been generated.

Defining a parallelepiped volume body by two points



1. Name the volume body to be created.
2. Choose the "Parallelepiped by two points" option.
3. Set the coordinates of point 1.
4. Set the coordinates of point 2.
5. Press "Apply" to generate the volume body.

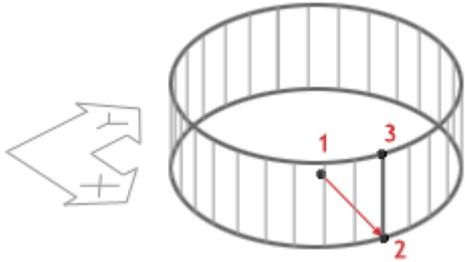
Defining a parallelepiped volume body by one point and three dimensions



1. Name the volume body to be created.
2. Choose the "Parallelepiped by point and dimensions" option.

3. Set the point coordinates.
4. Set the three lengths along the X, Y and Z axes, respectively.
5. Press "Apply" to generate the volume body.

Defining a cylindrical volume body interactively by the center and two points



1. Name the volume body to be created.
2. Choose the "Cylinder by center and two points" option.
3. Click inside the graph zone in order to identify the center point 1 (either an existing point or nonexistent but lying in the working plane).
4. Click inside the graph zone to identify point 2. The base circle is then displayed interactively.
5. Click inside the graph zone again to identify point 3. An alternative solution consists of setting the height H of the volume in the dialogue box and then activating the "Apply" button. The newly defined volume body has now been generated.

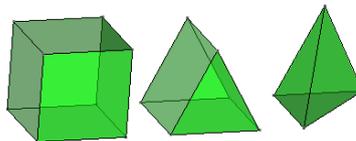
 Height H may be set as a negative value for a downward direction.



« Super-element » body (hexaedron, pentaedron, tetraedron)

This tool serves to generation of volume body of various types by means of selected edges:

- 12 edges for the hexaedron,
- 9 edges for the pentaedron,
- 6 edges for the tetraedron.



Creating a volume body of « super-element » type

1. Select appropriate number of edges fir the volume body to be generated.
2. Set a name in the grid.
3. Click on "Apply" to generate the super element volume body.



Definition of an "any volume" type of volume body defined by its exterior surfaces

This option allows generating an "any volume" type of volume body by means of selecting the set of surfaces "outside" the targeted volume.

Defining an "any volume" type of volume body

1. Select the surface regions composing the set of surfaces "outside" the given volume model element.
2. Name the model element in the proposed grid.
3. Click on the "Apply" button to generate the particular volume model element. The surface bodies used to derive this volume body are now deleted.



Extrusion operations

This tool serves to create bodies by extruding the selected geometric entities (points, lines, surfaces). Extruding points will lead to creating 1D type bodies, while extruding lines yields surface bodies and extruding surfaces will yield volume bodies. It is possible here to perform extrusions by both translation and rotation.

Performing an extrusion

1. Select the set of geometric entities to be extruded.
2. Select the type of "extrusion" (creation of 1D, surface or volume bodies). Only those selected elements of the associated type will actually be taken into account during the extrusion operation (e.g. lines will be selected if the type of extrusion = "creation of surface bodies").
3. Select either the "translation" or "rotation" type of operation.
4. Set the number of "translation" or "rotation" operations.
5. If translation: Indicate the V_x , V_y , V_z components of the translation vector in the structure's global coordinate system.



The translation-based extrusion of a surface cannot take place in the original surface.

6. If rotation: Identify the angle of rotation as well as the coordinates of two points P1 and P2 that define the axis of rotation in the structure's global coordinate system.



The axis of rotation must lie in the same plane as the element to be extruded (whether a surface or segment).

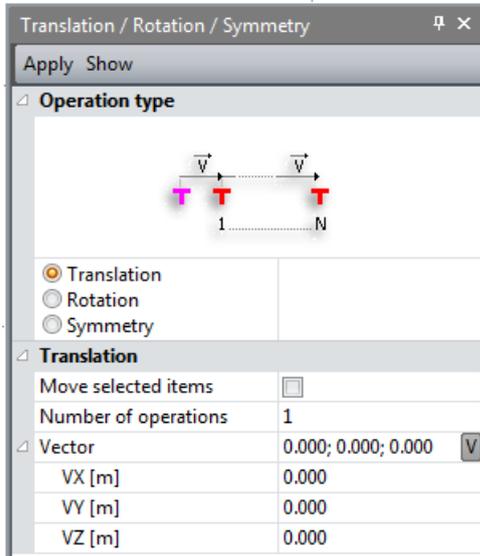
7. If volume bodies are being created: Choose the "Delete surface bodies" option should it be preferred for the model to only contain the newly created volume bodies and no longer the surface bodies used to support the extrusion operation.
8. Press "Apply" to execute the extrusion operation.



Operations on bodies: Translation, Symmetry, Rotation

This tool serves to duplicate, translate or rotate geometrical entities or bodies.

Translation of a body



Duplicating bodies via translation

1. Select the set of bodies to be duplicated by translation.
2. Activate the "Translation" tool.
3. Verify that the box labeled "Move selected items" has not been ticked.
4. Specify the number of translation operations by inputting a value in cell "Number of operations".
5. Define the translation vector components within the current reference working plane.
6. Use the "Apply" button to perform the translation operation.



User may collect the translation vector components following these operations : i) click the « **V** » button located on the « Vector » line ; ii) using left mouse button, click on a point as starting point of the vector ; iii) next, click on a point as end of the translation vector.

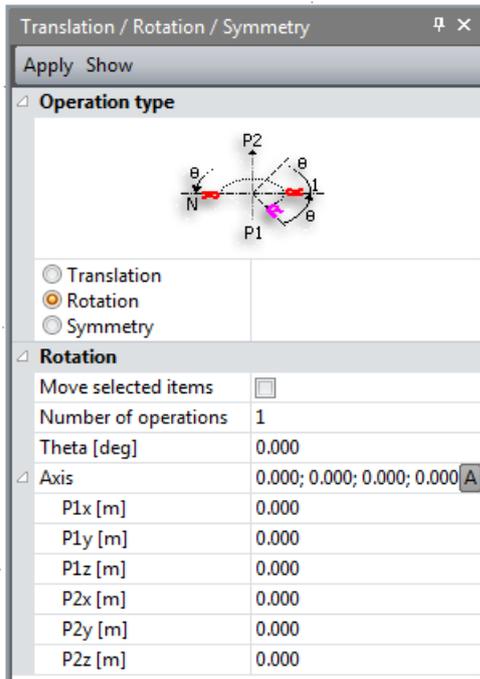
Moving bodies via translation

1. Select the set of bodies to be moved.
2. Activate the "Translation" tool.
3. Tick the box labeled "Move the selected entities".
4. Define the translation vector components within the current reference (see: Coordinate system - Working plane).
5. Use the "Apply" button to perform the translation operation.



The boundaries may be selected in any order either before or after activating the "Translation" tool.

Rotation of bodies



Duplicating bodies via rotation

1. Select the set of bodies to be duplicated by rotation.
2. Activate the "Rotation" tool.
3. Verify that the box labeled "Move selected items" has not been ticked.
4. Specify the angle characterizing each rotation operation by inputting a value for "Theta" in the appropriate cell.
5. Indicate the number of rotation operations.
6. Define the current reference coordinates of points 1 and 2 that serve to identify the axis of rotation.
7. Use the "Apply" button to perform the rotation operation.

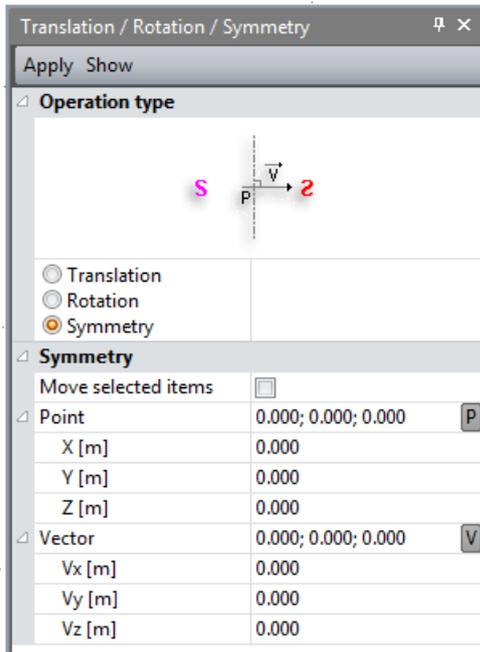
Moving bodies via rotation

1. Select the set of bodies to be moved by rotation.
2. Activate the "Rotation" tool.
3. Define the current reference coordinates of points 1 and 2 that serve to identify the axis of rotation.
4. Specify the angle characterizing the particular rotation operation by inputting a value in the "Theta" cell.
5. Tick the box labeled "Move selected items".
6. Use the "Apply" button to perform the rotation operation.

i The coordinates of an existing point may be "recovered" according to the following procedure carried out using the right mouse button: i) Position the cursor on the dialog box in either the X, Y or Z field of the point whose coordinates are to be recovered; ii) Right click on the given point of the model.

i Boundaries may be selected in any order either before or after activating the "Rotation" tool.

Symmetry of bodies



Duplicating bodies via symmetry

1. Select the set of bodies to be duplicated by symmetry.
2. Activate the "Symmetry" tool.
3. Verify that the box labeled "Move the selected entities" has not been ticked.
4. Specify the coordinates of a point lying in the plane of symmetry within the current reference.
5. Specify the components of a vector perpendicular to the plane of symmetry within the current reference.
6. Use the "Apply" button to perform the symmetry operation.

Moving bodies via symmetry

1. Select the set of bodies to be moved by symmetry.
2. Activate the "Symmetry" tool.
3. Tick the box labeled "Move the selected entities".
4. Specify the coordinates of a point lying in the plane of symmetry within the current reference. The coordinates of this point may be "recovered" by right clicking on a point of the model contained in the plane of symmetry.
5. Specify the components of a vector perpendicular to the plane of symmetry within the current reference.
6. Use the "Apply" button to perform the symmetry operation.



The boundaries may be selected in any order either before or after activating the "Symmetry" tool.



User may collect the vector components following these operations : i) click the « **V** » button located on the « Vector » line ; ii) using left mouse button, click on a point as starting point of the vector ; iii) next, click on a point as end of the translation vector.



User may collect the coordinates of the point "**P**" may be "recovered" by right clicking on a point of the model contained in the plane of symmetry.

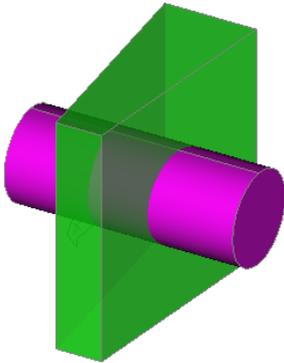


Intersections between volumes or surfaces

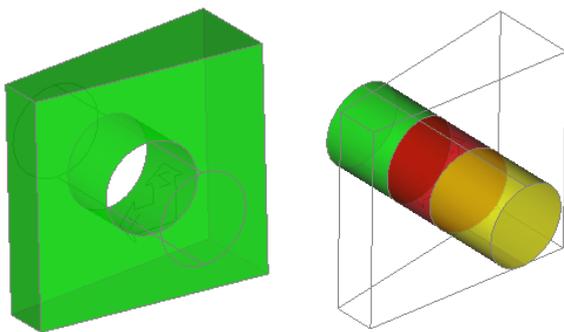
This powerful tool serves to search for intersections (partitions) between volume bodies or surface bodies.

Creating an intersection between volume bodies

This "intersections" tool option enables intersecting one volume body by another.

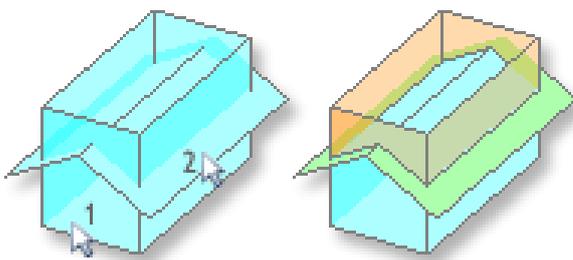


1. Choose the "Volume body intersection" option.
2. Click on the "B" button in the "Object entity" grid row.
3. Define the first volume body "to be partitioned" by clicking on the specific volume. The name of this volume body is then displayed in the grid.
4. Next, click on the "B" button of the "Tool entity" grid row.
5. Define the second "tool" volume body by clicking on the specific volume. The name of this volume body is then displayed in the grid.
6. Choose whether it is preferred to partition just the "object" entity or else both the "object and tool" entities using the option proposed in the grid.
7. Click on the "Apply" button. The intersecting lines and the volume bodies resulting from the intersection are now displayed.



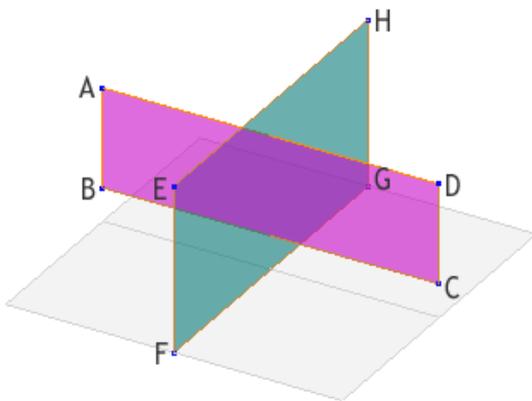
Creating an intersection between surface bodies

This option serves to define the intersection between two surface bodies.

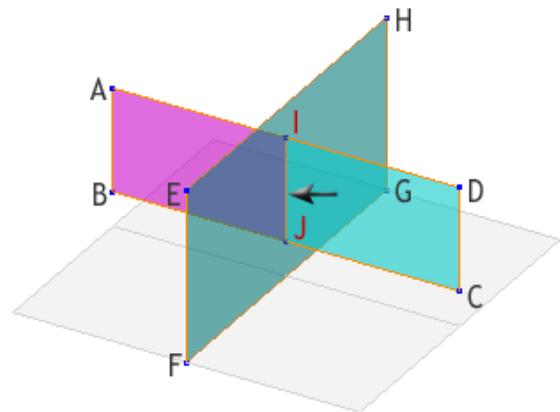


Defining the intersection between two surfaces

1. Choose the "Surface body intersection" option.
2. Click on the "B" button in the "Object entity" grid row.
3. Define the first surface body "to be partitioned" by clicking on the given surface. The name of this surface body is then displayed in the grid.
4. Next, click on the "B" button of the "Tool entity" grid row.
5. Define the second "tool" surface body by clicking on the particular surface. The name of this surface body is then displayed in the grid.
6. Choose whether it is preferred to partition just the "object" entity or else both the "object and tool" entities using the option proposed in the grid.
7. Click on the "Apply" button. The intersecting lines and the surface bodies produced from the intersection are now displayed.



Initial model configuration



The model after intersection of the two surfaces

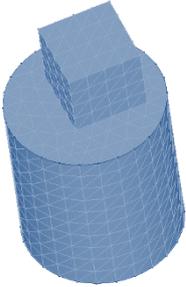
8. In the above example, we have considered the intersection of two surface bodies ABCD and EFGH. The right-hand figure shows the result obtained upon completion of the intersection operation. From an analytical standpoint, this case will yield the following elements after the intersection step:
 - Points I and J are created.
 - Line IJ positioned at the intersection of the two planes is created.
 - Line AD is now replaced by two lines AI and ID.
 - Line BC is now replaced by two lines BJ and JC.
 - Surface body ABCD is **entirely "cut"** by surface EFGH. Upon completion of this intersection operation, surface body ABCD is composed of two surface regions ABJI and IJCD.
 - Surface EFGH **is not entirely "cut"** by surface ABCD; it thus remains unchanged.



Merging volumes or surfaces

This tool serves to:

- create a volume model element by "merging" two selected volume model elements.
- create a surface model element by "merging" two selected surface model elements.



This operation can prove to be worthwhile in many cases. Let's consider for example the case of the above figure. The merging operation between the two solids makes it possible to simplify the geometry definition stage in that the cylindrical volume may be defined independently of the top parallelepiped. The merging operation will provide the necessary links and the creation of a single volume body.

Merging two volume bodies

1. Choose the "Merge volume bodies" option.
2. Click on the "B" button of the "Bodies 1" grid row.
3. Define the first volume body "to be merged" by clicking on the given volume. The name of this volume body is then displayed in the grid.
4. Next, click on the "B" button of the "Bodies 2" grid row.
5. Define the second "tool" volume body by clicking on the given volume. The name of this volume body is now displayed in the grid.
6. Click on the "Apply" button to execute the merging operation.



Intersection of volumes by a user plane

This tool serves to partition the volume bodies selected by a plane that has been defined by one point P and a normal vector.

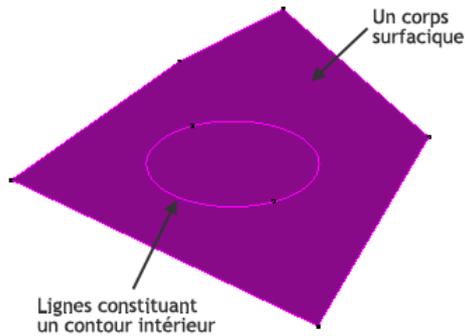
Creating an intersection of volumes by a plane

1. Select the volume bodies to be partitioned.
2. In the structure's global coordinate system, indicate the X, Y and Z coordinates of a point lying in the cutting plane.
3. In the structure's global coordinate system, record the components of a vector V normal to the cutting plane.
4. Press "Apply" to execute the partitioning operation.
- 5.

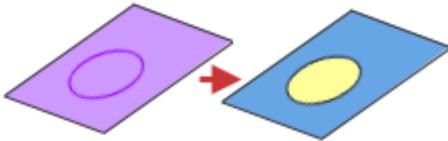


Partition of a surface body using lines

This tool serves to partition a surface body using lines that might constitute an inner contour, as indicated in the following figure.



Partitioning a surface body using lines



1. Select the surface body to be partitioned.
2. Select the lines used for this partitioning operation.
3. Press "Apply" to execute the partitioning operation.



Merge bodies

This tool merges selected bodies into a single body. It is different from the fusion tool as this operation is reversible (by using tool "Explode") whereas fusion operation is not.

Merge selected bodies

1. Choose the type of bodies to merge (1D, surface, volume).
2. Set the name of the new resulting body.
3. Select the bodies to be merged.
4. "Apply" to generate the merging operation.



Explode bodies

This tool explodes (unmerges) selected bodies that have been merged by previous operation.

Explode selected bodies

1. Select the bodies to be exploded.
2. "Apply" to generate the exploding operation.



Definition of anchors

This tool serves to assign the "anchored" attribute to the set of selected 1D bodies. All 1D bodies featuring this attribute will be used during meshing procedures carried out on volume bodies containing these same attributes. Each point of these 1D bodies will actually be considered as a "governing point" for the volume mesh. Such an integration step will preserve the "anchor in the volume" characteristic of the given 1D body.

Defining the anchors

1. Select the set of 1D bodies featuring the "Anchored" attribute.
2. Choose whether for these bodies it is preferable to assign or delete an anchor.
3. Press "Apply" to include the requested operation. The 1D bodies assigned an "Anchor" attribute are represented with a double-line thickness.



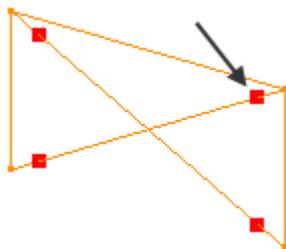
1D body connections

Activating this tool serves to generate two-node special elements positioned on one end of a 1D body. These elements are created by directly designating with the mouse the end to receive the body.

This operation proceeds by:

- creating a node featuring the same coordinates as the 1D element end node,
- creating a "special" element composed of the two nodes with identical coordinates ("connection" type bodies).

The newly created "connection" type bodies can be visualized graphically by a symbol placed at the indicated 1D body end.



Representation of "connection" type bodies



Elements of this type will mainly be used to model special connections (e.g. ball joints) between beam elements.

Creating "connection" type bodies

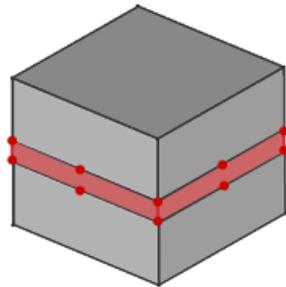
Run the "1D body connections" tool.

1. Name the body to be created.
2. Left-click on the ends of the 1D bodies targeted for the connection to be built.

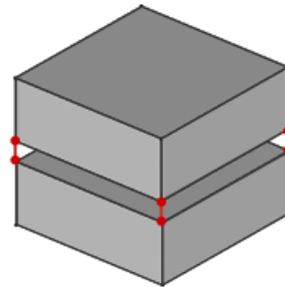


Creation of interfaces

This option serves to define a "interface type body" on a set of common surfaces between volume bodies.



Interface body of « contact » type



Interface body of « link » type

Generating an « interface » type body

1. Remove visibility of some volume bodies so as to visualize the surfaces on which the contact bodies are set to be defined.
2. Select these surfaces (selection tool )
3. Name the interface body to be generated.
4. Choose the element type used to model the interface (12- or 16-node contact elements or 2-node connections).
5. Press "Apply" to create the newly defined interface bodies. These bodies are now shown on the screen, as indicated in the previous figure.

Modifying an « interface » type body

1. Select the interface bodies to be modified (selection tool  + ).
2. Chose the new type of interface body interface (12- or 16-node contact elements or 2-node connections).
3. « Apply ».



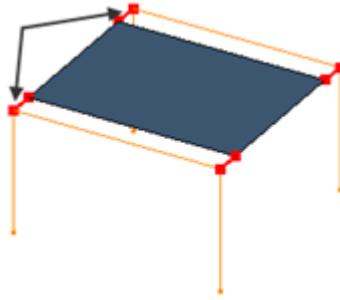
Manual element creation: "Multi-point" bodies

This tool serves to generate "multi-point body" type elements. The procedure followed to create such a body makes use of the sequence described below.

Creating a multipoint body

1. Activate the "Manual element creation" tool.
2. Name the body to be created.
3. Select a point of the model by a right-click of the mouse. The characteristics of the given point are now displayed in the dialogue box.
4. Press the "+" button to add the selected point to the points already making up the multi-point body. The number of the indicated point is then displayed in the "Element" section of the dialogue box.
5. Repeat steps 3 and 4 for all points composing the multi-point body.
6. Press "Apply" to generate the newly defined multi-point body, which is represented by a line connecting the targeted nodes along with a specific symbol positioned on each of these nodes.

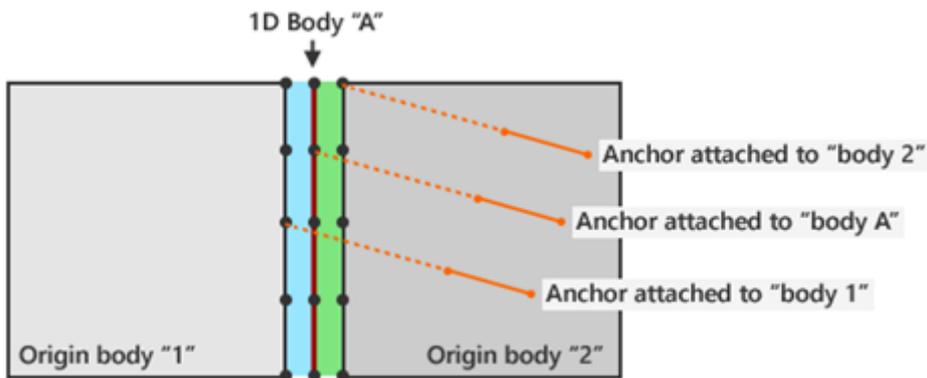
« Multipoint »-type bodies



Attachment of anchors

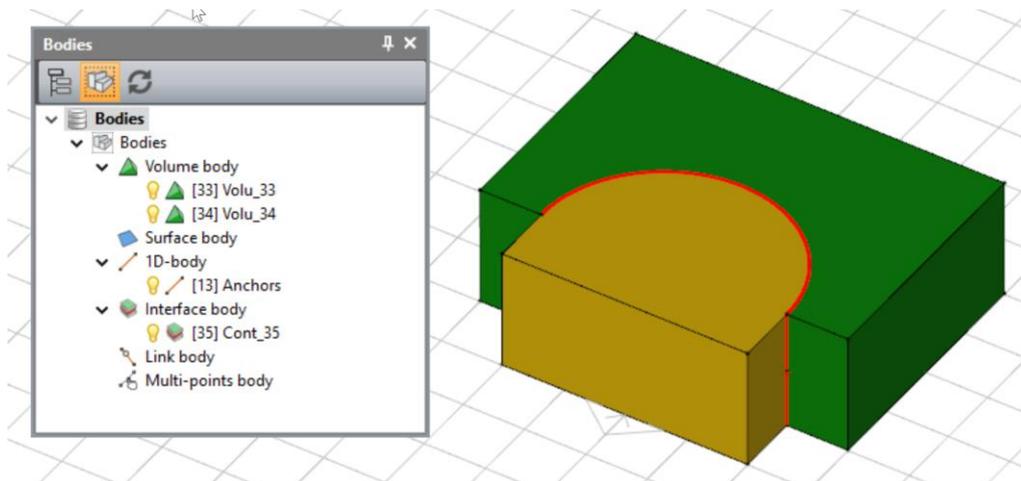
Default behaviour of bodies generation links the anchors (1D-bodies) to the wall (shell bodies), even if interface bodies are defined on this wall body.

In more complex situations, user may want to define where the anchor is attached as illustrated below.



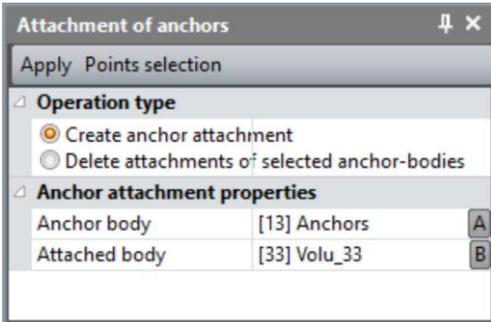
Definition of anchors attachment

It starts with this situation:



In order to attach the 1D-bodies "Anchors" to the volume body "Volu_33", user will proceed as follow:

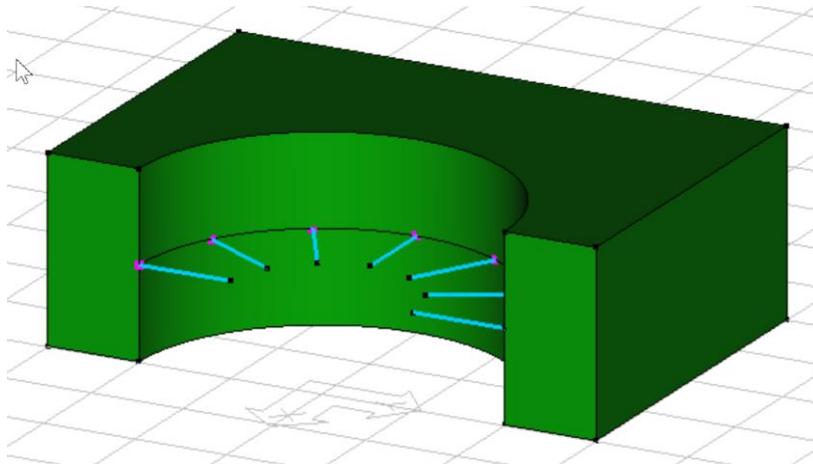
1. Activate tool "Attachment of anchors" 
2. In the property grid, press "A" in the item "Anchor body".
3. Select the body "Anchors"
4. In the property grid, press "B" in the item "Attached body".
5. Select the body "Volu_33"
6. After these actions, situation is as follow:



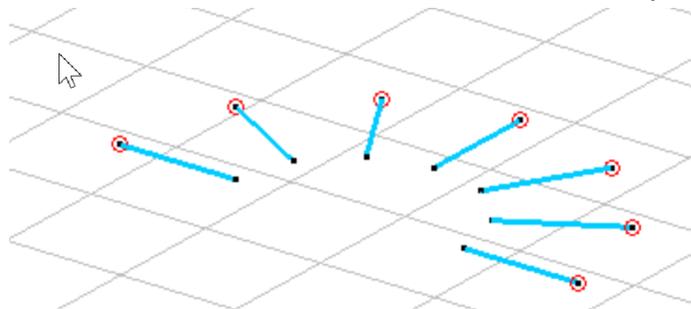
7. Press "points selection" in order to select the common points between "Anchors" and "Volu_33"

i This tool is useful to select ALL the points in common between the anchors and the targeted body. User may also select manually this set of points.

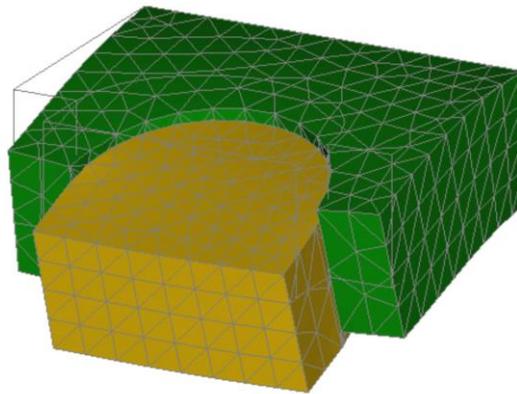
8. Several points are selected:



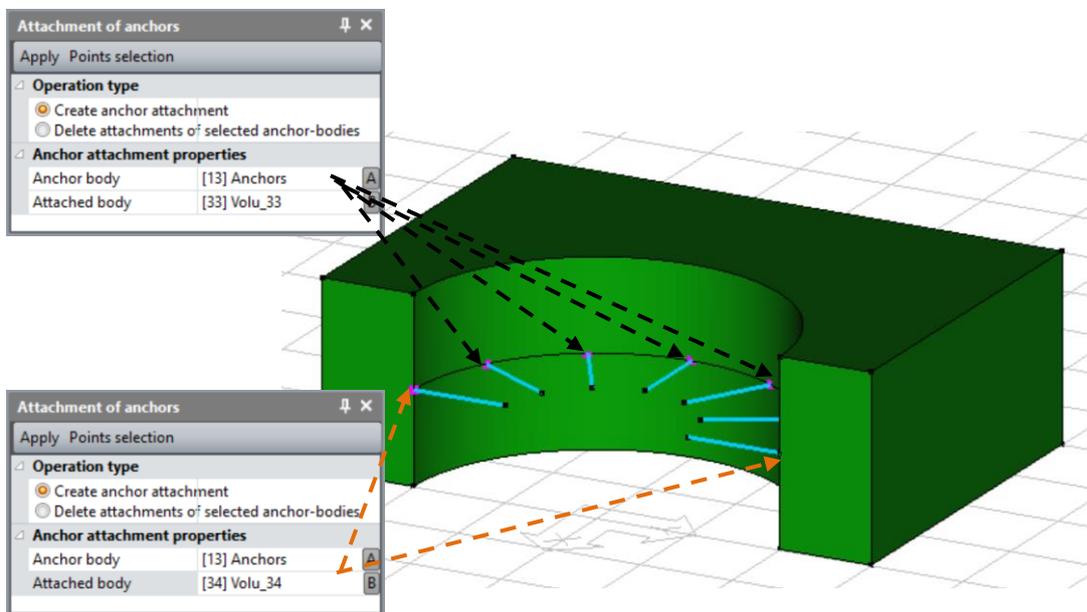
9. Press "Apply" so that the attachments of the anchors are created. They are identified by a red circle.



Results after calculation show that the anchors are correctly connected with the volume.



Attachments of anchors are associated to points. Thus, user can define different attachment for the same group of several anchors, as illustrated below.



Deleting attachment of anchors

1. Define the attached bodies: 1 anchor body and 1 attached body.
2. Click on "Points selection" in order to select the attachment points.
3. Select also the anchor body (use [Maj] for multi-selections)
4. "Apply".



Body/wall attachment

Default behaviour links anchors to walls (1D bodies in 2D or surface bodies in 3D).

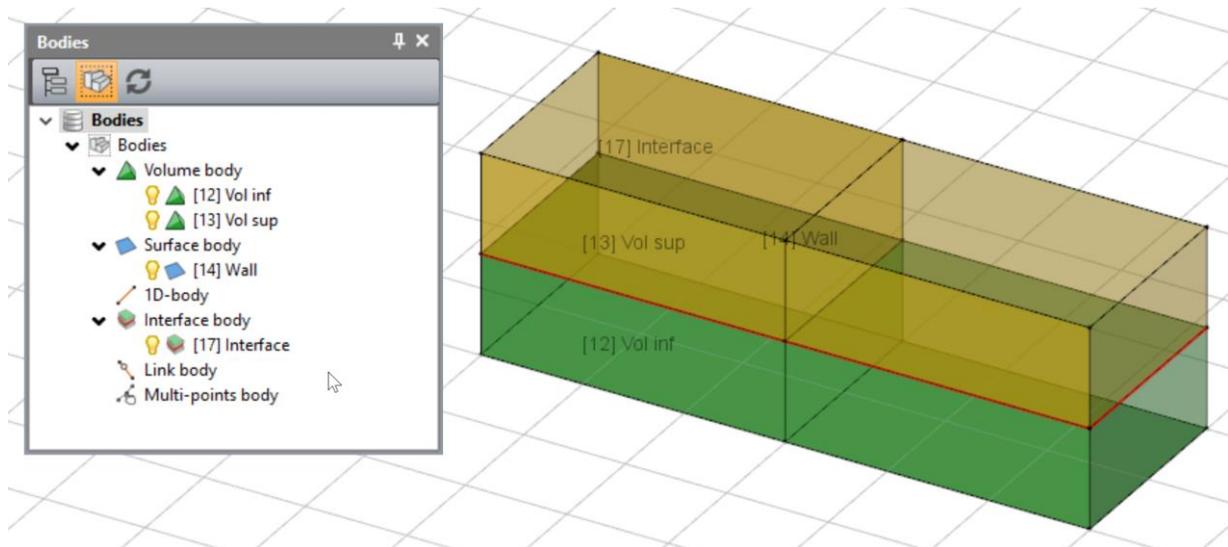
If ambiguous, user can select the body where the anchor will link.

Target state

- The user creates contact between two bodies (material)
- The user creates a "wall" on the same location between the two bodies
- The user must be able to choose the side of the material that is ATTACHED to the wall (no contact elements)
- With this information, when calculating CESAR creates one group of contacts on the side that is NOT ATTACHED with the material
- If the user allows contact elements on both sides of the wall (upper / lower side) by not defining attachments, he/she must be able to apply different material properties to each side.

Definition procedure

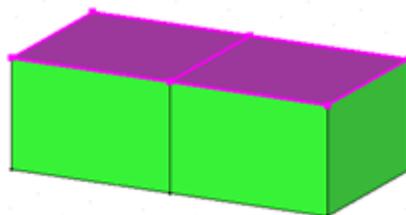
We have this situation:



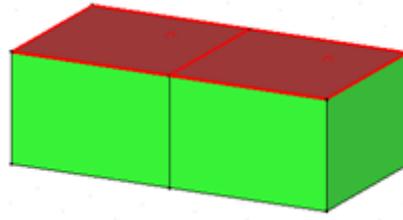
When no attachments defined, contact elements are created on both sides of the wall-bodies.

To attach the lower volume bodies to the wall bodies we proceed as follows:

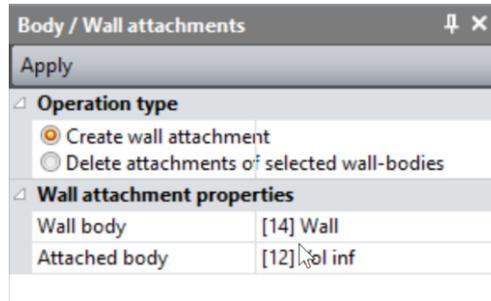
1. Select the tool wall attachment 
2. Hide the upper volume bodies
3. Select the common faces between lower volume bodies and wall-bodies



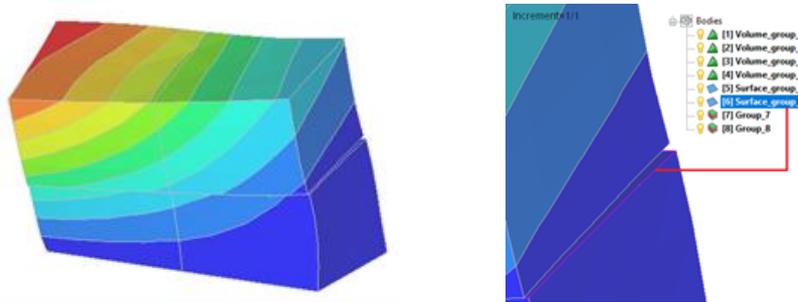
4. Choose "Create wall attachment"
5. Press the toolbar button "Apply"
6. The attachments are created on the wall-bodies and the small red markers indicate their existence:



By right clicking on the wall-face, the wall attachment properties are shown:



After the calculations, it can be seen that the wall-group is merged with the lower volumes:



Results after attaching lower volumes to wall bodies



Scripts

This tool offers the possibility to access scripts from the scripts library or created by the user (refer to the reference manual for edition of scripts using Python and XML).

The default script proposed is TunnelM; it allows to draw common tunnel sections.

Script TunnelM

This script gives access to 3 types of tunnel geometries:

- Circular tunnel,
- Tunnel defined by 2 radius and 2 angles,
- Tunnel defined by 3 radius and 3 angles.

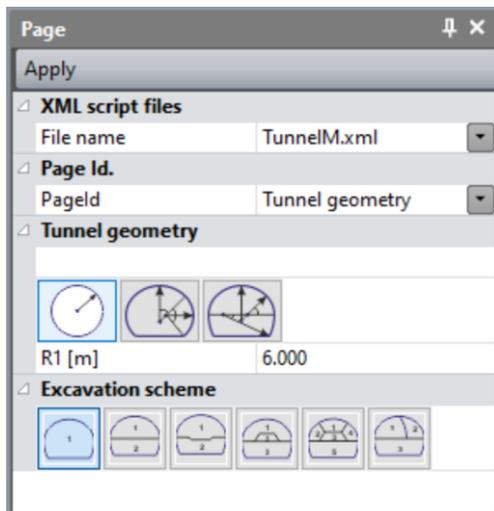
For the 2 last types, it is possible to:

- Declare an asymmetry,
- Declare a non flat invert.

For each tunnel type, 6 schemes of excavation are proposed:

- Full section
- Bench and top heading
- With excavation en section inférieure et section supérieure
- With bench cut
- With top heading cut in 2 parts (method #1)
- With top heading cut in 4 parts
- With top heading cut in 2 parts (method #2)

Use of the script TunnelM



1. Select on type of tunnel. Fill-in the associated parameters.
2. Select an excavation scheme. Fill-in the associated parameters.
3. Apply.



Import of the geometry is done in the working plane previously defined by the user.



Delete

The "Delete" tool allows removing the selected entities. Let's note here that deleting a surface or volume body will, by default, cause the lines on the working drawing identifying the body contours to be deleted provided these lines are not being used by another model body. The dialogue box proposed with this tool offers an option to save these lines if so desired.

Delete the selected bodies

1. Select the bodies to be deleted.
2. Uncheck the "Delete body lines" option if it is preferred to keep these lines.
3. Press "Apply" to complete the deletion operation.



Undo

This option serves to cancel the most recent action performed. It is possible to cancel several successive operations by activating several times the "Edition / Undo" menu option or by clicking several times on the "Undo" button.

"Undo" is also possible with the classic keyboard key combination: [Ctrl] + z.



Redo

This option serves to restore the most recent cancelled action. It is possible to restore several previously cancelled operations by activating the "Edition / Redo" menu option several times or by clicking several times on the "Redo" button.



Colors

This option serves to modify the color attribute of the selected entities containing a color attribute.

Modifying the color of selected entities

1. Select the targeted entities.
2. Choose the color to assign by clicking in the corresponding pad in the dialogue box.



Note that the vertical "palette" toolbar has the same purpose.



Model properties

Running this tool displays a dialogue box presenting the primary characteristics of the geometric model being defined (number of points, lines, bodies, surface, volume, etc.).



Body-related information

This option serves to visualize the geometric properties and attributes of the body selected by a right-click; it also allows modifying the name and color attribute of the given body.

Modifying the color and name of a body

1. Right-click on the targeted body; its properties and attributes are now displayed in the dialogue box.
2. Modify as needed the body name.
3. Modify as needed the body color.
4. Press "Apply" to save these changes.



Correction of the geometry

This option checks the consistency of all geometrical items generated in the model. After activating several Boolean operations, after editing, after undo/redo, etc., the model is composed of several and diverse geometrical items. Among them, some may present defaults more or less crucial for the completion of the study (coincident points or edges for example).

Checking/Fixing the geometry

1. Click on feature « Fix geometry ».
2. Select one, several or all entities to check.
3. Click on « Check ».
4. If the boxes in the « Results » section are not « 0 », then click « Fix ».

Fix geometry	
Check	Fix
Check points	
Coincident points	<input type="checkbox"/>
Check edges	
Coincident edges	<input type="checkbox"/>
Check faces	
Faces with bad orientation	<input type="checkbox"/>
Check volumes	
Solids with bad faces orientation	<input type="checkbox"/>
Check bodies	
Faces not connected to bodies	<input type="checkbox"/>
Check bodies	<input type="checkbox"/>
Results	
Coincident points	0
Coincident edges	0
Faces with bad orientation	0
Faces without parent bodies	0
Empty bodies	0
Bodies with wrong type	0
Bodies with missing or wrong mesh	0

6. Mesh definition

The mesh generation step requires two steps for volume, surface or 1D bodies.

The 1D body mesh requires:

1. Cutout of this element,
2. Creation of the 1D element.

The surface body mesh requires:

1. Cutout of its contours,
2. Creation of the mesh surface.

The volume body mesh requires:

1. Cutout of its contours,
2. Filling using volume elements. During this procedure, regardless of the selected meshing program, the software will begin by meshing the surfaces defining the volume body envelope as a background task.

It is advised to mesh with a high density in zones with strong stress gradients.



Constant density

This tool allows setting the number of equal-length intervals (element side lengths) desired on the selected model boundaries.

Defining cutouts by number of intervals

1. Select the set of boundaries for which a cutout is to be assigned.
2. Activate the "Constant density" tool.
3. Enter the number of intervals desired by clicking on one of the predefined values in the dialog box.
4. Press the "Apply" button in order to assign the cutouts.



Model boundaries may be selected in any order either before or after activating the "Cutouts per number of intervals" tool.

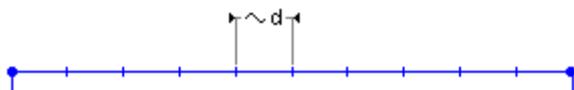
Recovering the number of intervals assigned on a given boundary

1. Activate the "Cutouts per number of intervals" tool.
2. Right click on the given boundary. The number of intervals assigned on this boundary is then displayed in the dialog box.



Fixed length density

This tool serves to define the number of desired intervals on the selected model boundaries by inputting a value for length "d".



For each boundary, the number of assigned intervals is computed such that the length of each interval falls as close as possible to the inputted length "d".

Defining density per distance

1. Select the set of boundaries for which a cutout is to be assigned.
2. Activate the "Fixed length density" tool.
3. Enter the appropriate distance "d" value in the dialog box.
4. Press the "Apply" button to assign the cutouts.

Recovering the interval length assigned on a given boundary

1. Activate the "Fixed length density" tool.
2. Right click on an interval of the given boundary. The length of this interval is then displayed in the dialog box.



Progressive-density

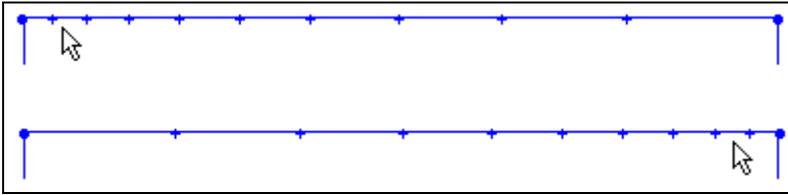
This tool enables defining variable-length cutouts on a given boundary. The characteristics of the cutout to be assigned are determined by entering values for any two of the four data items listed below:

- Length of the first interval
- Length of the last interval
- Total number of intervals
- Geometrical increment

i Note that it is impossible to assign a multiple selection ("Apply" button deactivated) since using the mouse to designate a contour serves to establish the position of the first interval on the given boundary.

In general, the procedure for applying this tool is based on the following sequence.

1. Input any two data values from among the four specified in the dialog box.
2. Click on the given boundary to assign the defined cutout and then indicate the position of the first interval.



Influence of the selected point position

Assigning a progressive density on a boundary

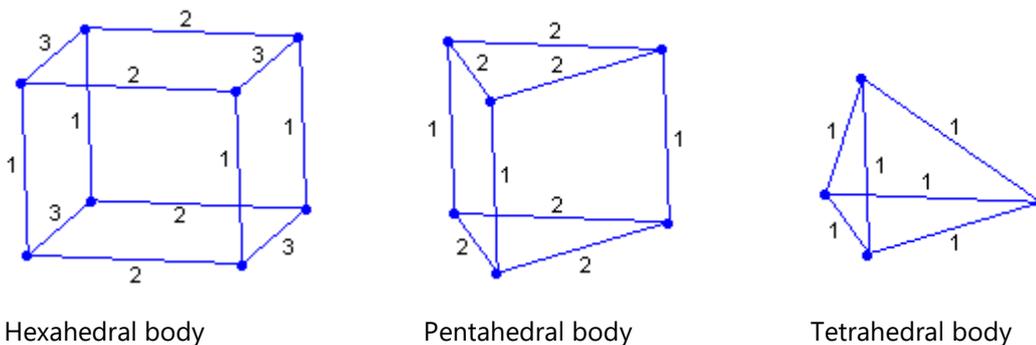
1. Activate the "Progressive density" tool.
2. In the dialog box, tick the two values used for defining the variable-length cutout.
3. Left click on the given boundary. It should be recalled that the selected point position on the boundary also serves to define the position of the first interval.

Recovering the characteristics of a density assigned on a given boundary

1. Activate the "Progressive density" tool.
2. Right click on the given boundary. The characteristics of the cutout assigned on this boundary are then displayed in the dialog box.

i Meshing of "Super elements" bodies

Given the procedures used to generate the mesh for "super element" bodies of the hexahedral, pentahedral or tetrahedral type, certain special conditions are imposed on the cutouts associated with the boundaries composing these bodies. From a schematic standpoint, we can state that cutouts assigned on opposing boundaries must be identical.



The boundaries ascribed the same number on the above figure are thus required to display identical densities in order for the volume element mesh of the given body to be generated.

These conditions have been automatically incorporated herein. Inputting a cutout on a particular boundary will therefore engender propagation of this cutout onto all boundaries deemed to be opposing.



Mesh of 1D-bodies

This tool enables generating 1D elements on the set of selected segments. Activating this tool displays a dialog box that allows the user to choose the type of interpolation (linear or quadratic) desired for these newly-created elements.



It is possible to reverse this default direction by activating the "Line element orientation" tool.

Generating 1D-elements

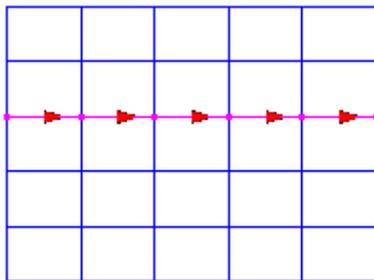
1. Select the set of 1D-bodies.
2. Activate the "Mesh of 1D-bodies" tool.
3. Specify the type of interpolation (linear or quadratic).
4. Use the "Apply" button to create 1D- elements.



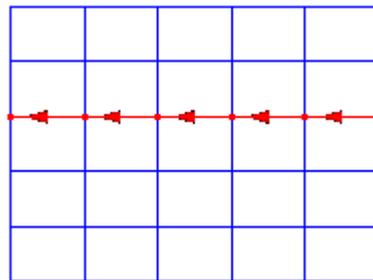
1D element orientation

Activating this tool reverses the direction of all selected 1D elements. The new direction associated with the selected 1D elements can be visualized graphically as follows:

Orientation of 1D elements has an influence on the sign of the results for beams or bars.



Initial direction



After direction reversal

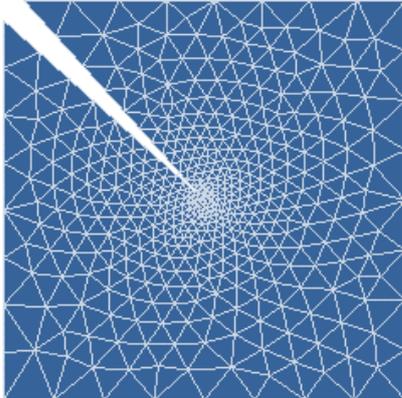
Inverse 1D element direction

1. Select the set of 1D elements whose direction is to be changed.
2. Activate the "1D element orientation" tool.
3. "Apply"

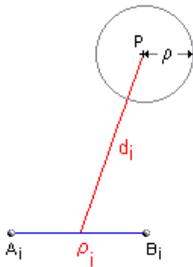


Surface meshing

This tool serves to generate automatically the mesh on selected surface bodies.



The meshing procedures used enable incorporating variable-length cutouts. At a given point P of the surface region to be meshed, the meshing algorithm must determine the "optimal" size ρ of the element to be generated.



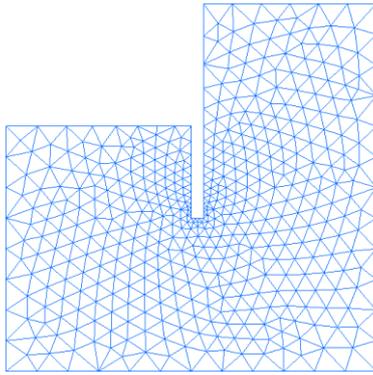
This size ρ depends both on the length of each "segment" ρ_i constituting the inside and outside edges of the surface region to be meshed and on the distance d_i between point P and the particular segment. The relationship can be schematically described as follows:

$$\rho(x, y) = f\left(\rho_i, \frac{1}{d_i^N}\right)$$

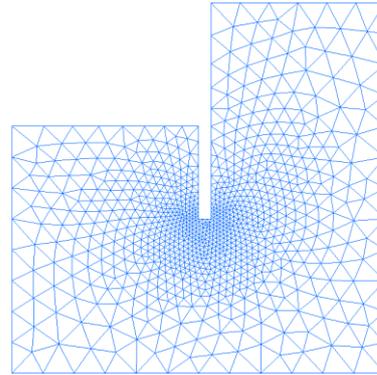
In the above expression, an exponent N has been attributed to the distance connecting point P to a given segment. This exponent may assume one of three following values:

- N = 1: Linear function
- N = 2: Square function
- N = 3: Cubic function

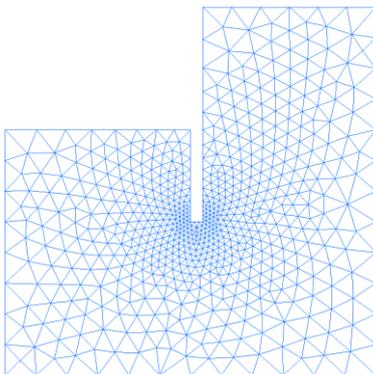
At this juncture, the user may influence the meshing procedure employed by choosing any one of the three functions listed above. In schematic terms, it can be stated that meshes exhibit a finer level of detail in the vicinity of small-sized segments as the degree of the chosen function increases.



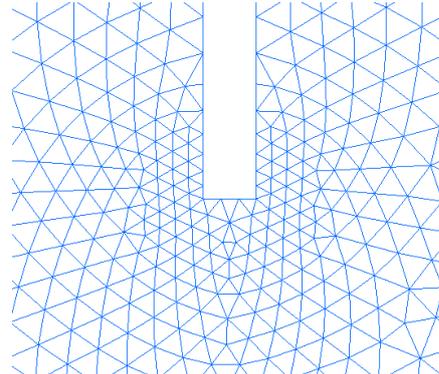
Linear function



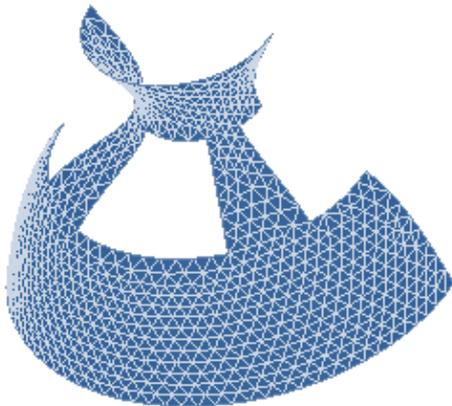
Cubic function



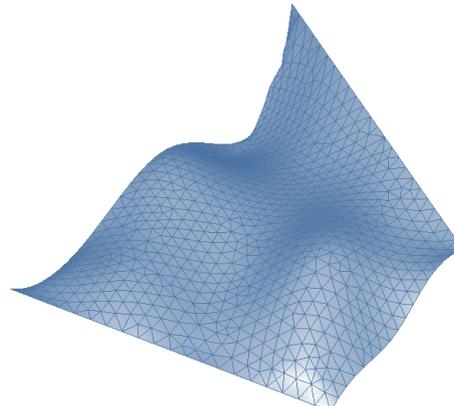
Mesh obtained using the square function



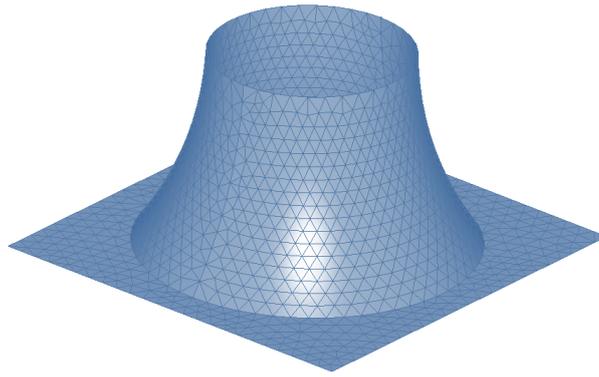
The figures below provide several examples of meshes that can be obtained on various types of surface regions.



Surface region of the "revolution" type



Surface region of the "Coon's patch" type



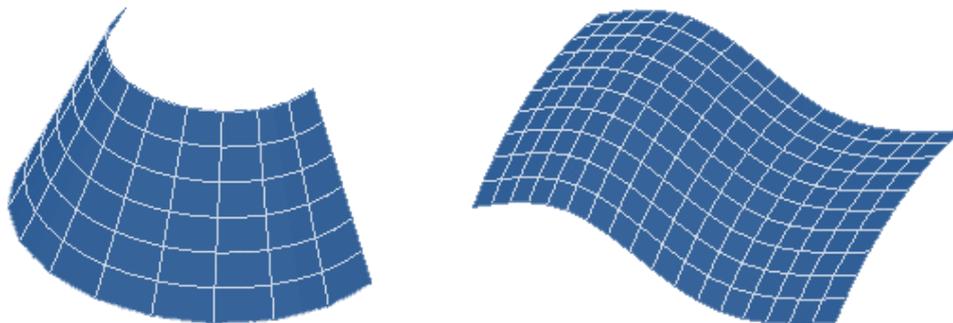
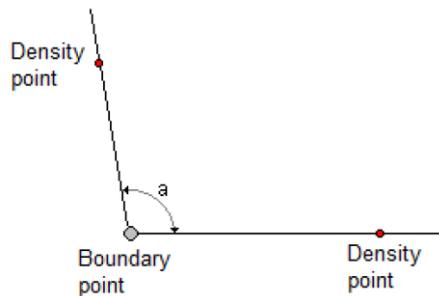
Mesh of both revolution regions and planar regions

Activating this tool displays a dialog box that enables assigning the following attributes:

- Type of interpolation for the generated elements. The user can choose whether the triangles or quadrilaterals generated will feature a linear (three-node triangles and four-node quadrilaterals) or quadratic (six-node triangles and eight-node quadrilaterals) type of interpolation.
- Type of elements generated. The user can also choose the type of element to be generated (triangle, quadrangle, no preference).

It should be pointed out that the choice of a quadrangle will only be accorded for so-called "regular" surface regions. The term "regular" is intended herein to connote a surface region that satisfies the following conditions:

- The region is composed of four boundaries.
- The numbers of intervals (cutouts) on opposite boundaries are identical.
- The angle between two contiguous boundaries is less than a given value a_{lim} , which for present purposes has been set at 120 degrees.



Mesh generated using quadrangle elements of regular regions

Generating meshes of surface bodies

1. Select the set of surface bodies to be meshed.
2. Activate the "Surface body meshing" tool.
3. Indicate the type of interpolation (linear or quadratic) for the elements to be generated.
4. Specify the desired type of elements (triangle, quadrangle, no preference).
5. Use the "Apply" button to generate the mesh of the selected surface bodies.

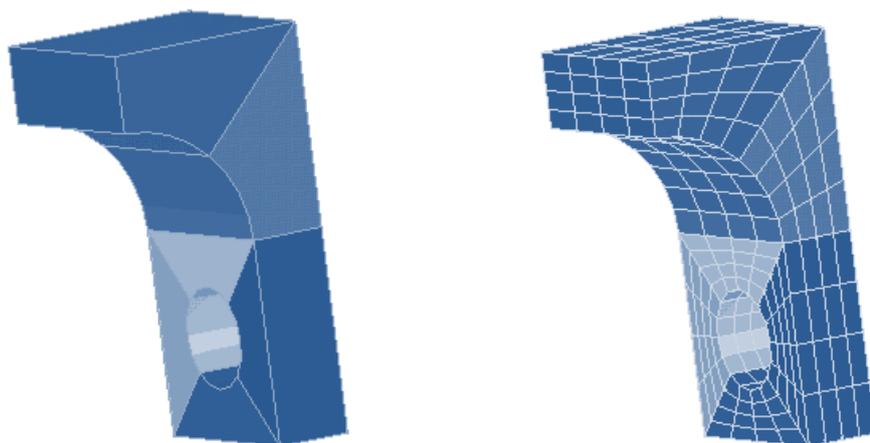


Volume meshing

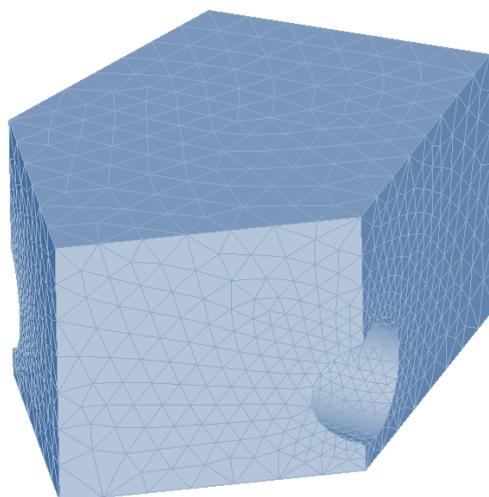
Cet outil permet la réalisation automatique du maillage de blocs volumiques quelconques.

Activating this tool displays a dialog box that enables assigning the following attributes:

- Type of elements generated. The user can also choose the type of mesh: regular for "super element" bodies or tetrahedron for general volume bodies.
- Type of interpolation for the generated elements. The user can choose whether elements generated will feature a linear (4-nodes tetrahedron, 6-nodes pentahedron, 8-nodes hexahedron) or quadratic (10-nodes tetrahedron, 15-nodes pentahedron, 20-nodes hexahedron) type of interpolation.



Maillage « régulier » réalisé à l'aide de six régions volumiques. Chaque région volumique possède ici une couleur différente



Remplissage par tétraèdres d'une région volumique quelconque

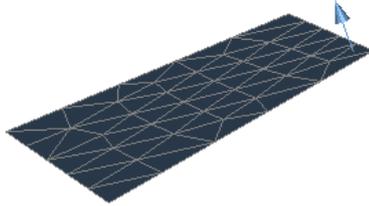
Meshing of a volume body

1. Select the set of volume regions to be meshed.
2. Activate the "Volume meshing" tool.
3. Identify the type of interpolation (linear or quadratic) for the elements to be generated.
4. Use the "Apply" or "Validate" button to create the mesh of the selected volume regions.

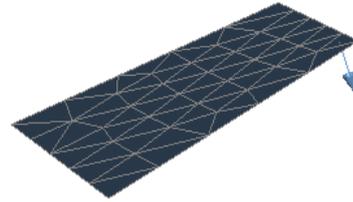


Surface body orientation

This tool serves to reverse the positive normal orientation of the set of selected surface bodies. The revised orientation associated with the selected surface bodies is visualized graphically (activate the "View normal to the surfaces" option under "Preferences / Mesh").



Initial orientation



After inversion

Changing the surface body orientation

1. Select the set of surface bodies whose orientation is intended to be changed.
2. Activate the "Surface body orientation" tool.
3. Press "Apply" to execute the given operation.



Mesh properties

This tool displays main characteristics of the displayed mesh elements (number of nodes, number and type of elements). It allows also to check the quality of the mesh.



Displayed properties are updated in function of the displayed parts of the mesh.

Check the quality of the mesh elements

By clicking " cliquant « Elements quality check », user gets the qualitative status of all the volume elements. The process checks after the shape of the elements and calculate their associated Jacobian.

At this step, it is highly recommended to have no elements with problems, as:

- Run of the calculation might be impossible;
- If the calculation is run, the quality of the results might questionable.

Mesh properties	
Update Elements quality check	
Modelled bodies	
Number of bodies	0
Mesh size	
Number of groups	0
Total number of nodes	0
Total number of elements	0
1D-elements	0 (-)
Surface elements	0 (-)
Volume elements	0 (-)
Mesh properties	
Area [m2]	0.000
Volume [m3]	0.000
Elements quality check	
Number of elements entirely with $J < 0$	0
Number of elements partially with $J < 0$	0
Number of very distorted elements	0
Number of distorted surface elements	0

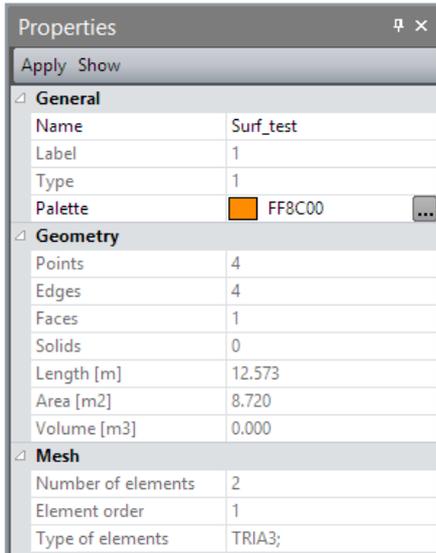


Body-related information

This option serves to visualize the geometric properties and attributes of the body selected by a right-click; it also allows modifying the name and color attribute of the given body.

Modifying the color and name of a body

1. Right-click on the targeted body; its properties and attributes are now displayed in the dialogue box.
2. Modify as needed the body name and/or the body color.
3. Press "Apply" to save these changes.

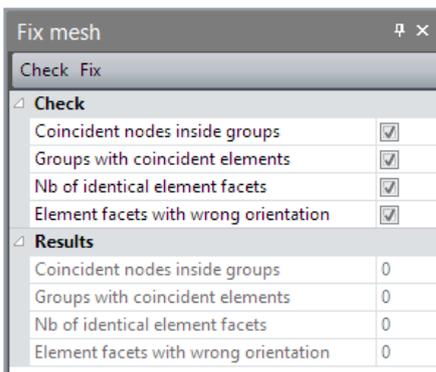


Correction of the mesh

This option checks the consistency of all items in the mesh. After activating several Boolean operations, after editing, after undo/redo, etc., the mesh is composed of numerous and diverse type elements. Among them, some may present defaults more or less crucial for the completion of the study (coincident nodes or elements for example).

Checking/Fixing the mesh

1. Click on feature « Fix geometry ».
2. Click on « Check ».
3. If the boxes in the « Results » section are not « 0 », then click « Fix ».



7. Calculation modules and elementary physical problems

CESAR integrates a list of algorithms dealing with several elementary or coupled physical issues.

Library of elementary physical problems

The above table defines the elementary physical problems associated to each calculation module for a given field of application.

Field of application: Statics		
Module	Function	Physics
LINE	Resolution of a linear problem by means of the direct method	Mechanics
MCNL	Resolution of a mechanics problem exhibiting non-linear behavior	Mechanics
TCNL	Resolution of a contact problem between elastoplastic solids	Mechanics
Field of application: Dynamics		
Module	Function	
DYNI	Determination of response to a dynamic load by means of direct integration for linear elastic structures	Mechanics
MODE	Determination of eigenmodes: Eigenvalues and eigenvectors	Mechanics
SUMO	Determination of the response to a dynamic load by means of modal superposition	Mechanics
FLAM	Search for buckling modes	Mechanics
LINC	Determination of response to a harmonic load with damping (resolution of a linear problem in complex variables)	Mechanics
LINH	Determination of response to a harmonic load without damping	Mechanics
Field of application: Hydrogeology		
Module	Function	
DTNL	Resolution of a non-linear transient hydrogeology problem	Hydrogeology
NSAT	Resolution of a flow problem in unsaturated porous media (transient state)	Hydrogeology
LINE	Resolution of a flow problem in permanent state	Hydrogeology
Field of application: Heat transfer		
Module	Function	
DTNL	Resolution of a non-linear transient thermal problem	Thermics
LINE	Resolution of a thermal problem in permanent state	Thermics
Field of application: Concrete at early age		
MEXO	Evolution of stresses in early-age concrete	Mechanics
TEXO	Computation of the temperature field evolution in a concrete specimen during setting (transient state)	Thermics
Field of application: Consolidation		

Module	Function	
CSNL	Resolution of a consolidation problem involving saturated elasto-plastic materials	Mechanics Hydrogeology
Field of application: Coupled problems		
Module	Function	
MPNL	Resolution of a non-linear evolution problem in a porous medium with thermal coupling (thermo-poro-plasticity)	Mechanics Hydrogeology Thermics

Content of each elementary physical problem

Associated to the choice of the calculation module, the model will be defined by:

- a set of material properties,
- initialisation conditions,
- one or several boundary conditions sets,
- one or several loading sets.

In the particular case where the field of application where the field of application is a coupled one (Consolidation for example), initialization of parameters, boundary conditions and loadings are associated to several elementary physical problems.

For example, solving a problem of consolidation (CSNL calculation module) requires the consideration of two elementary physical problems: "Mechanics" and "Hydrogeology". It is thus necessary to apply boundary conditions on "displacements" and "hydraulic loads" parameters; and it is possible to generate loading sets of "mechanical" or "hydrogeological"-types (see example below).



The graphical user interface will be updated in function of user's choice to propose appropriate features.

Compatibility of "physics"

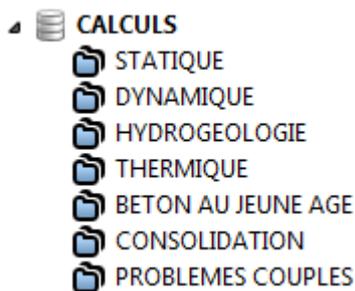
CESAR allows the combination of several elementary physics on the same calculation model. Some adaptations are required to assure the compatibility.

Body	Mechanics	Hydrogeology/Heat transfer
Volume	Active	Active
Shell	Active	Inactive
1D	Active	Inactive
Interface	Active	Permeable / Impervious Infinite conductivity / Thermal insulation
Special	Active	Inactive

8. Model definition

"Model" management

The "Models" tree diagram offered in the right-hand part of the screen presents the various "computation fields" potentially applicable to CESAR. This diagram will be used to create and edit the project "models". It is initially presented as shown below:



(+) Static computations

This "Branch" contains the set of models associated with static computations. It also serves to treat the conventional problem of scheduling works phases.

(+) Dynamics

This "Branch" contains the set of models associated with dynamic computations (eigenmodes, time-dependent dynamic response, etc.).

(+) Hydrogeology

This "Branch" contains the set of models created in order to solve a hydrogeology problem.

(+) Heat exchange

This "Branch" contains the models created in order to solve a heat exchange problem. TEXO is not included in the list of eligible tools under this branch.

(+) Early-age concrete

This "Branch" contains the models (TEXO-MEXO) created in order to solve an early-age concrete problem.

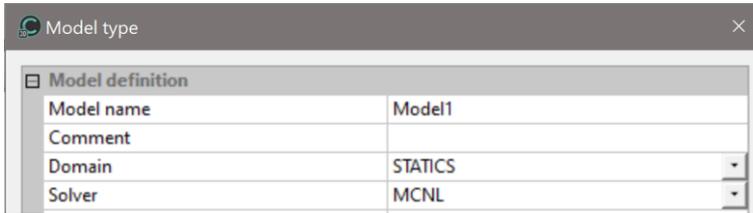
(+) Consolidation

This "Branch" contains the models created in order to solve a consolidation-type coupled problem.

(+) Coupled problems

This "Branch" contains the models created to solve a "thermo-poro-mechanical" type of coupled problem.

The action of right-clicking on one of the initial tree diagram items displays a context menu proposing the creation of a new model. Activating this menu item then displays a dialogue box that allows establishing the main characteristics of the targeted model.

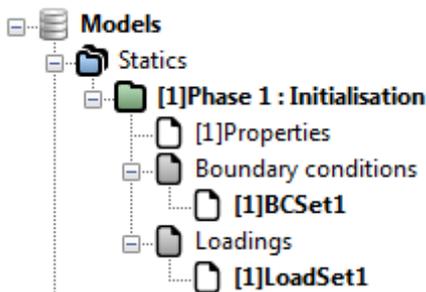


Above all, the following need to be defined:

- The user name assigned to the field
- The field application (e.g. Statics, Dynamics)
- The associated "solver" (e.g. LINE, MCNL).

The following section will provide greater detail of the other characteristics defining a model, as determined by the context.

Once this model definition dialogue box has been validated, the tree diagram is modified, as shown in the following example:



The tree diagram presented above also allows performing the following basic operations:

- 1) Right-clicking on "Model name" opens the following context menu:
 - General properties (update the model's general properties)
 - Activate model (the selected model becomes the current model)
 - Delete model
 - Copy model (create a copied model from the current model)
 - Record the computational settings (settings for the selected solver).

- 2) Right-clicking on the "Boundary conditions" section opens the following context menu:
 - Add a "Set of boundary conditions" (if allowed by the current solver).

- 3) Right-clicking on the "Load case" section opens the following context menu:
 - Add a "Load case" (if allowed by the current solver).

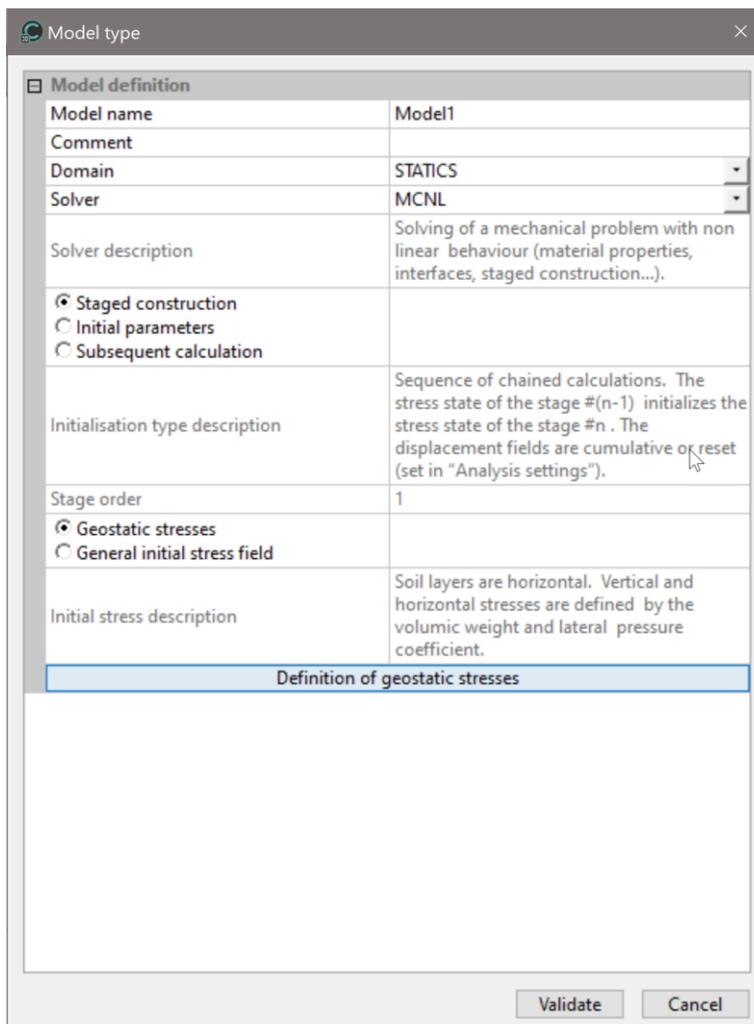
- 4) Right-clicking on a set of "Boundary conditions" or "Load cases" opens the following context menu:
- Activate the set (the selected set becomes the current (active) set in defining the boundary conditions or load cases)
 - Delete the set (if allowed by the current solver)
 - Delete sharing (if the selected set is being shared (see following section)).

i Let's also note the possibility to change the names of sets of properties, boundary conditions and load cases by clicking on the selected set and then pressing the F2 key on the keyboard.

Properties of a model

Previous chapter detailed the process of creating a model. It displays a toolbox for definition of "Properties" of the model. User can at any time access his toolbox by a right click on the model name. More than the name of the model, this toolbox helps to set the physical domain of application and the associated solver (see chapter "Calculation modules and elementary physical problems").

i The list of fields of application and associated calculation modules is limited by the activated CESAR license.



Initialisation type

After definition of the physical domain, it is necessary to choose the type of initialization.

For some of the computation modules, it is necessary to set the initial value on a number of parameters. As an example, in order to conduct a dynamic analysis by means of direct integration (the DYNl computation module), the initial values of the displacement fields, speed fields and stress fields would all have to be defined.

Generally speaking, four initialization types or methods are made available to the user.

a) No initialization required

In this particular case, no additional information has to be set.

b) Initial parameters

Use of this method makes it possible to "directly" define the initial parameters. This definition process is performed within the "Parameter initialization" module whose functional features will be discussed in a subsequent chapter.

c) Subsequent calculation

This initialization method is chosen whenever the given model actually constitutes a "continuation" of computations set forth in a previous model. In returning to the example of a dynamic analysis by means of direct integration, it may be anticipated that a model (denoted "A") serves to analyze the behavior of a structure between time 0 and time T. The behavior between times T and T+T' could then be assessed using another model (denoted "B"), for which a "Simple restart" type of initialization will be requested. As shown in greater depth in the "Parameter initialization" chapter below, opting for this method simply requires entering the name of the file created by model "A" that contains the parameter value at time T.

d) Staged construction

This type of initialization can be considered as an extension to the "Subsequent calculation" method; it merely introduces a predefined initialization process that enables simplifying the user's task.

If the "Staged construction" method were chosen, a number of order N would automatically get assigned to the given model, according to the number (N-1) of models already created using a "Staged construction"-type initialization.

The initialization process associated with each model (stage) is thereby defined in accordance with the following specifications:

Stage	Initialization
1	Initial displacements: zero Initial stresses: either zero or geostatic
2	Initial displacements: - zero, if the initial stresses defined in Phase 1 equal zero; - results of Stage 1, if the initial stresses defined in Phase 1 are geostatic Initial stresses: results of Stage 1
N (N>2)	Initial displacements: results of Stage N-1 Initial stresses: results of Stage N-1

The above table reveals that the user is required to choose the type of initial stresses for just the model corresponding to Stage 1. For all other models, the initializations are automatically set once the "Staged construction" initialization method has been chosen.

Initial stress field

Relevant in geotechnical studies, the initial stress field is required for the good estimation of the stress field in the soil before the works. It can have a large influence on the results (displacements or stresses).

Two methods of stress field generation are offered.

a) Geostatic stresses

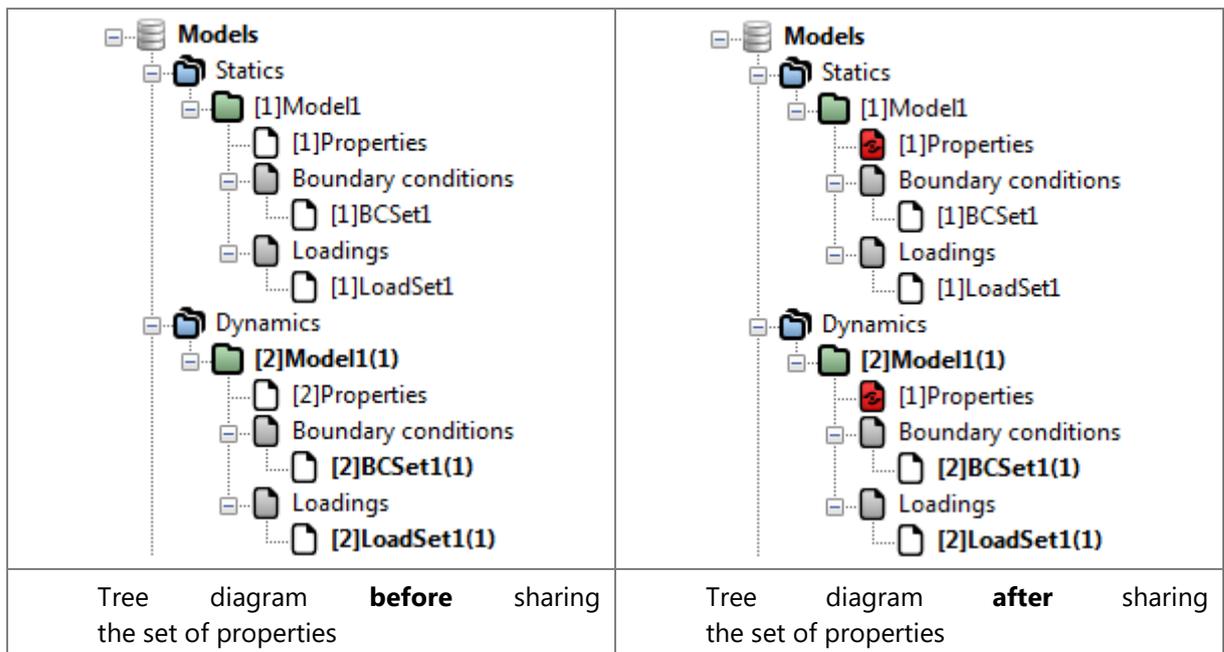
The soil layers are horizontal. Vertical and horizontal stresses are defined by the volumic weight and lateral pressure coefficients.

b) General initial stress field

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, so that user has the choice for the accurate modelling of this important initial state of the model.

"Sharing" of sets (properties, boundary conditions, loads) among several models

The tree diagram presented above also allows for the possible sharing of "sets" across several models. Let's assume for the sake of illustration a preference to use the same set of properties for a standard "static computation" type model as for a "dynamic (vibrational Eigen modes)" type model. The tree diagram then takes the following form:



In order to use the set of properties "[1] Properties" associated with the "Static / Model1 computations" model within the "Dynamics / Model2" model, it will be necessary to "slide" (by dragging and dropping) the item "[1] Properties" onto the "[2] Properties" set.

This action generates a dialogue box proposing the following three choices:

- Copy: In selecting this option, the set of properties "[2] Properties" is automatically initialized with data contained in the set "[1] Properties". When this action is complete, the two datasets remain independent and modifying one does not affect the other.
- Share: In selecting this option, the two models "share" the same set of properties. In this case, modifying the properties in one model will thus be immediately taken into account in the other model.
- Cancel: The copy or share action is cancelled by selection this option.

The above image reveals that models "Model1" and "Model2" are now using the same set of properties "[1] Properties". A symbol ( shown here) indicates that this selected set is shared across several models. This sharing procedure can be similarly used for sets of boundary conditions and load cases.

The "Shared set" attribute may be "removed" at any time. Right-clicking on the given shared set will display a context menu offering, in addition to the menu options indicated in the previous section, a "Delete sharing" item. Activating this functionality will create a new initialized set based on the previously shared set.

Construction stages process in geotechnical analysis

The simulation of the constructions stages is important for a correct representation of the phenomena and a good analysis of the stresses and strains in the structures. Each stage of construction will be defined as a new model in CESAR-LCPC.

In order to link the stages of construction, calculations will be initialized with the stress field calculated at previous stage. Hence, the following process:

Models	Initial stress field	Load sets	Calculated stress field
Stage 1	σ_0	ΔF_1	σ_1
Stage 2	σ_1	ΔF_2	σ_2
...			
Stage n	σ_{n-1}	ΔF_n	σ_n

It is important to note here that each model is unbalanced by an additional load set. Previous loads sets are already taken into account in the resulting calculated stress field.

If user choses the initialisation type "Staged construction", the process will be automatically done:

Models	Initial stress field	Load sets	Calculated stress field
Stage 1 Initialization type = Staged construction	σ_0 build with: - Ko procedure - or self-weight procedure - or WTB procedure	ΔF_1	Automatic storage of the stress field σ_1
Stage 2 Initialization type = Staged construction	Automatic initialisation of the stress field with σ_1	ΔF_2	Automatic storage of the stress field σ_2
...			
Stage n Initialization type = Staged construction	Automatic initialisation of the stress field with σ_{n-1}	ΔF_n	Automatic storage of the stress field σ_n

If user choses the initialization type "Initial parameters", the process will be defined through the following operations:

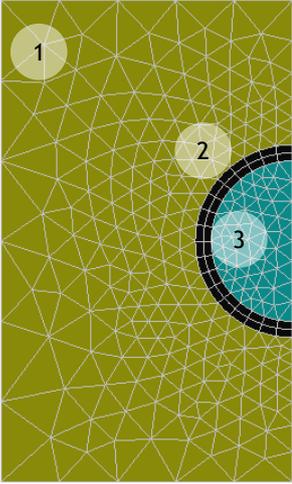
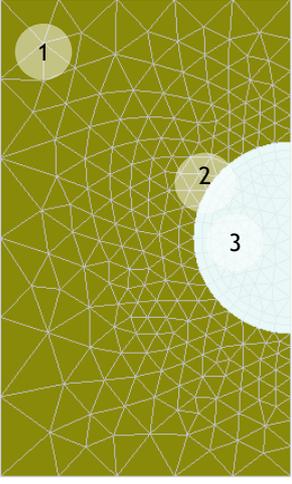
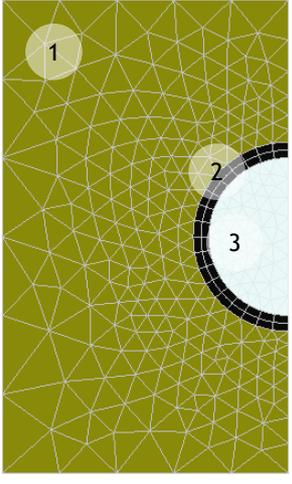
Models	Initial stress field	Load sets	Calculated stress field
Stage 1 Initialization type = Initial parameters	σ_0 build with:  Uniform stress per group  Geostatic stresses  Stresses from file	ΔF_1	Storage for restart = file, <i>Phase1.rst</i>

Stage 2 Initialization type = Subsequent calculation	Name of the file = Phase1.rst	ΔF_2	Storage for restart = file, <i>Phase2.rst</i>
...			
Stage n Initialization type = Subsequent calculation	Name of the file = Phase(n-1).rst	ΔF_n	Storage for restart = file, <i>Phase(n).rst</i>

Activation / inactivation of bodies during staged construction process.

Sets of elements having same properties and status during the calculation process will be a “group of elements”.

Example during tunnelling construction.

Phase 0 : Initial stress field	Phase 1 : Excavation of the tunnel section.	Phase 2 : Setup of the lining elements.
		
Groups 1, 2 & 3 are active.	Deactivation of groups 2 & 3.	Activation of group 2.
Properties of groups 1, 2, 3: soil parameters.	Properties of group 1: soil parameters.	Properties of group 1: soil parameters. Properties of group 2: concrete parameters.
Initial stress field: geostatic stresses	Initial stress field: $\{\sigma_0\}$	Initial stress field: $\{\sigma_1\}$
Load set: none.	Load set 1: Excavation forces on the limits of the tunnel section. → Tool Excavation forces 	Load set 2: Self-weight of lining elements → Tool Gravity forces 
Results: $\{\sigma_0\}$, $\{\varepsilon_0\}$, $\{F_0\}$	Results: $\{\sigma_1\}$, $\{\varepsilon_1\}$, $\{F_1\}$	Results: $\{\sigma_2\}$, $\{\varepsilon_2\}$, $\{\sigma_2\}$

Initial stress field for geotechnical analysis

In previous section, the initial stress field is initialized in the first calculation, set as "Phase 1". In given cases, it is necessary to isolate the initial stress field:

- control of the computed initial stresses,
- generation of initial stresses with the self-weight of the soil layers.

Therefore, 3 procedures are distinguished for initial stress field computation:

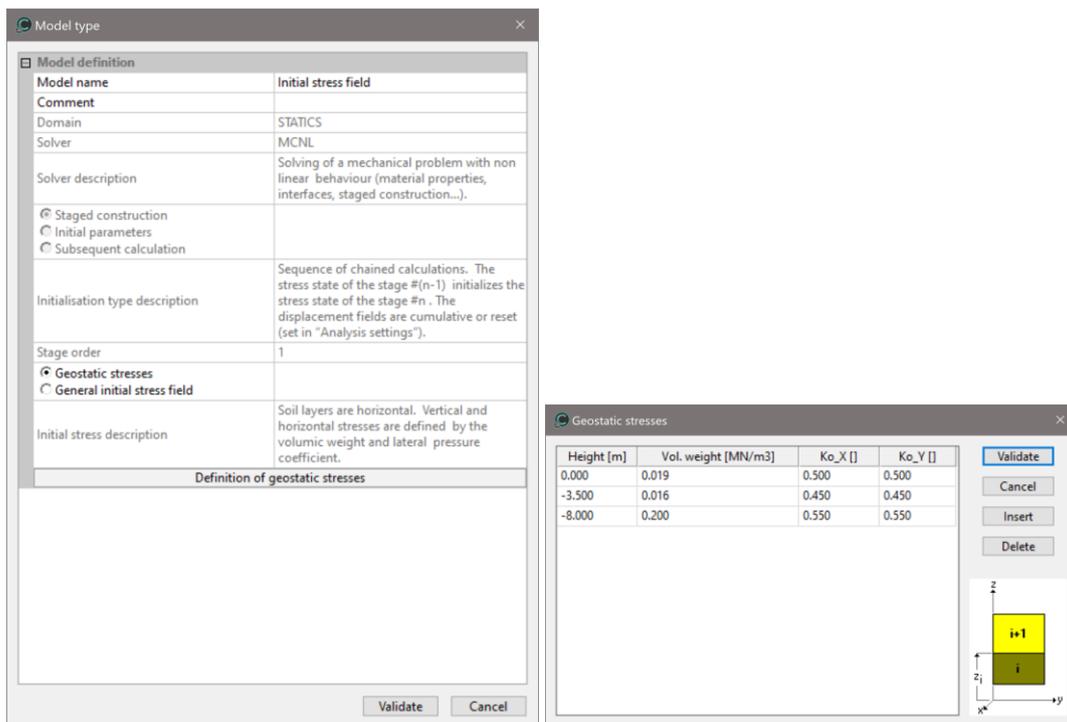
- geostatic stress field in horizontal layers,
- self-weight procedure,
- water table position (WTB) procedure.

Geostatic stress field in horizontal layers: the "K0 procedure"

The self-weight procedure is to be used in following cases:

- the soil layers are horizontal,
- the soil layers materials are homogeneous,
- interface elements are not activated.

The parameters used for the self-weight procedure are input in the **Geostatic stresses** definition:



where Ko_x and Ko_y are the earth pressure coefficients in the x and y directions respectively.

Hence in a uniform homogeneous layer, we get:

- Vertical stress $\sigma_v = \gamma \times h$
- Horizontal stress $\sigma_h = K_0 \times \sigma_v$

The process is described in the following table.

Models	Initial stress field	Load sets	Calculated stress field
<i>Stage 0</i> Initialization type = Staged construction	 Geostatic stresses	No load	Automatic storage of the stress field
<i>Stage 1</i> Initialization type = Staged construction	Automatic initialization of the stress field from <i>Stage 0</i>	Load set 1	Automatic storage of the stress field
...			

The “Self-weight procedure”

The self-weight procedure is to be used in following cases:

- the soil layers are not horizontal,
- the soil layers materials are not homogeneous,
- interface elements are activated.

The parameters used for the self-weight procedure are the elastic ones input in the **PROPERTIES** tab: density, Poisson’s ratio...

Hence in a uniform homogeneous layer, we get:

- Vertical stress $\sigma_v = \rho \times g \times z$
- Horizontal stress $\sigma_h = \frac{\nu}{1-\nu} \times \sigma_v$

The process is described in the following table.

Models	Initial stress field	Load sets	Calculated stress field
<i>Stage 0</i> Initialization type = Staged construction	General initial stress field	Self-weight of the soil layers with tool Gravity forces 	Automatic storage of the stress field
<i>Stage 1</i> Initialization type = Staged construction	Automatic initialization of the stress field from <i>Stage 0</i>	Load set 1	Automatic storage of the stress field
...			

The “WTB procedure”

The WTB procedure is to be used in following cases:

- the soil layers are not horizontal,
- the soil layers materials are not homogeneous,
- interface elements are activated,
- a water table exists.

The parameters used for the WTB procedure are the dry and saturated unit weights of the layers elastic ones input in the **PROPERTIES** tab: density, Poisson’s ratio...

Hence in a uniform homogeneous layer, we get:

- Vertical stress $\sigma_v = (\gamma_s - \gamma_w) \times z$ below the WTB level
 $\sigma_v = \gamma_d \times z$ above the WTB level
- Horizontal stress $\sigma_h = \frac{\nu}{1-\nu} \times \sigma_v$

The process is described in the following table.

Models	Initial stress field	Load sets	Calculated stress field
<i>Stage 0</i> Initialization type = Staged construction	General initial stress field	WTB stresses  and 	Automatic storage of the stress field
<i>Stage 1</i> Initialization type = Staged construction	Automatic initialization of the stress field from <i>Stage 0</i>	Load set 1	Automatic storage of the stress field
...			

9. Definition of model properties

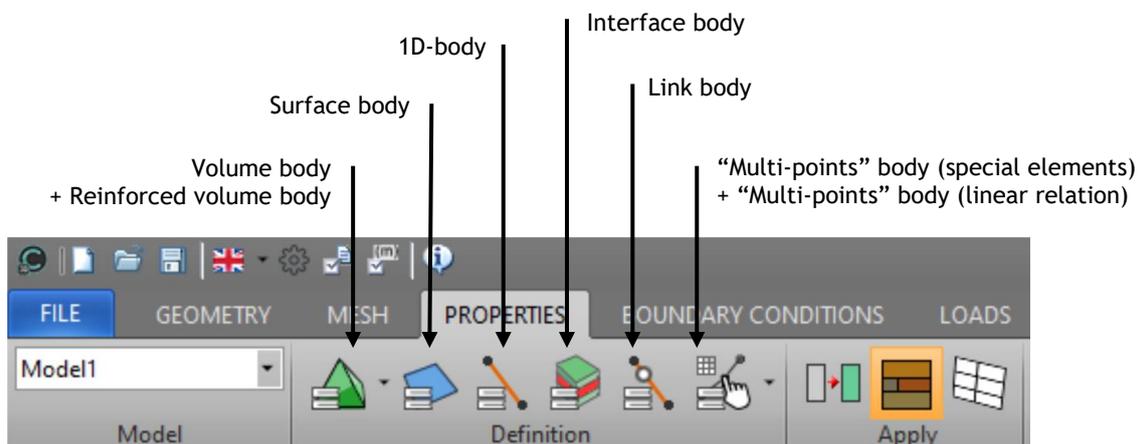
Definition of the property tables

For each of the model bodies in a given model, a "set of properties" needs to be defined (mechanical, hydraulic, geometric characteristics, etc.). The expression "property table" will be employed herein to indicate this combination of settings.

A property table may include the range of settings required to conduct all types of analyses made possible by the software (mechanical, heat exchange, hydraulic, etc.). To incorporate and organize these various categories of model settings, let's consider that a property table is in fact a set of "sub-tables", each of a specific "type", as exemplified by the following distinct sub-tables:

- Elasticity
- Plasticity
- Hydrogeology
- Heat exchange, etc.

In CESAR, the property tables associated with a project are defined independently of their assignment to the bodies of a given model. It is advised to begin by constituting a "library" containing the property tables to be implemented. This operation is performed using a series of tools proposed under the "Properties" tab.



Property table entries depend for the most part on the type of associated bodies. As such, the property table contents associated with "volume bodies" will differ from those associated with "surface bodies". For this reason, building a library of properties relies on a list of tools, with each tool being specialized for a given type of body. Let's distinguish the following:

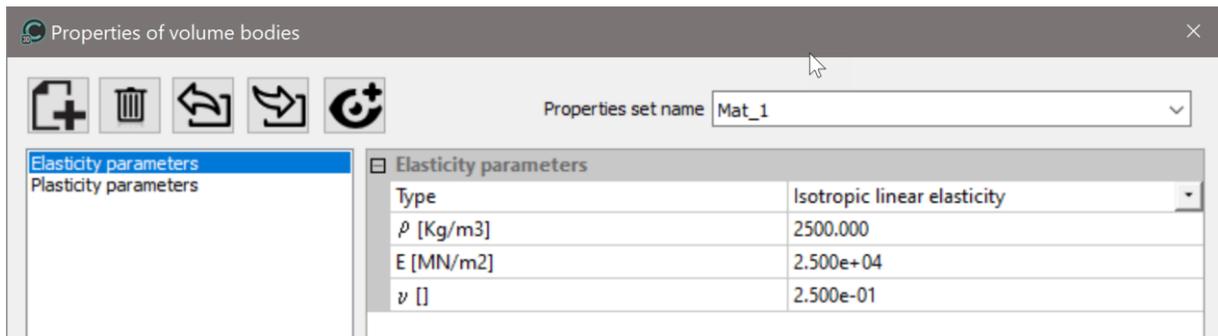
- Definition of properties for "volume bodies": the distinction is drawn here between "homogeneous" volume bodies and a reinforced material type of volume body
- Definition of properties for "surface bodies"
- Definition of properties for "1D bodies"

- Definition of properties for "interface bodies"
- Definition of properties for "connection bodies"
- Definition of properties for "multi-point bodies": this distinction is drawn here between "Linear relation" type multi-point bodies and "Special element" type multi-point bodies (with the user defining the matrices manually).

Running each of these tools displays a dialogue box offering the following actions for the particular type of body under consideration:

- Create a new "property table"
- Edit the characteristics of an existing "property table". Let's note here that the modifications introduced in the library will be automatically included in all models using this particular property table.
- Delete a "property table"
- Export a property table into an "external" library that's independent of the current project. This general library may thus contain the various property tables used repeatedly in several studies.
- Import a "property table" from the general library to the current project library.

These actions are all carried out via the toolbar available in the relevant dialogue box.



Creating the property tables

1. When opening this dialogue box, a default material has already been created. It is thus advised to simply update these properties, as indicated below.
2. Assign a name to the property table by editing it within the relevant drop-down list.
3. Choose the type of sub-property table to be completed. In the above example, the choice will be made between "Elasticity" and "Plasticity". The visibility of sub-tables i through N depends on the data entered in sub-tables 1 through i-1. As an illustration, if an "orthotropic elasticity" type of material is selected in the "Elasticity" sub-table, then the "Plasticity" sub-table will become invisible due to its incompatibility with this choice.
4. Define the properties associated with the selected sub-property table.
5. Repeat steps 3 and 4 for each sub-table presented in the dialogue box.
6. Click on the "New" button in the toolbar appearing in the dialogue box if it is intended to create a new property table. A default name shown in the list of library tables is assigned; simply repeat steps 2 through 4 as needed.
7. Click on the "Validate" button to save the newly created property tables in the "project library".

Modifying the property table settings

1. Choose the property table to edit using the drop-down list containing the set of project tables.
2. Choose the type of sub-property table containing the settings to be modified.
3. Update the settings associated with the selected sub-property table.
4. Repeat steps 3 and 4 for each sub-table to be modified appearing in the dialogue box.
5. Click on the "Validate" button to save newly updated property tables in the "project library".



It should be noted that the changes introduced through this sequence will be taken into account without requiring any further action in the models using the property tables modified herein.



Exporting property tables to an external library

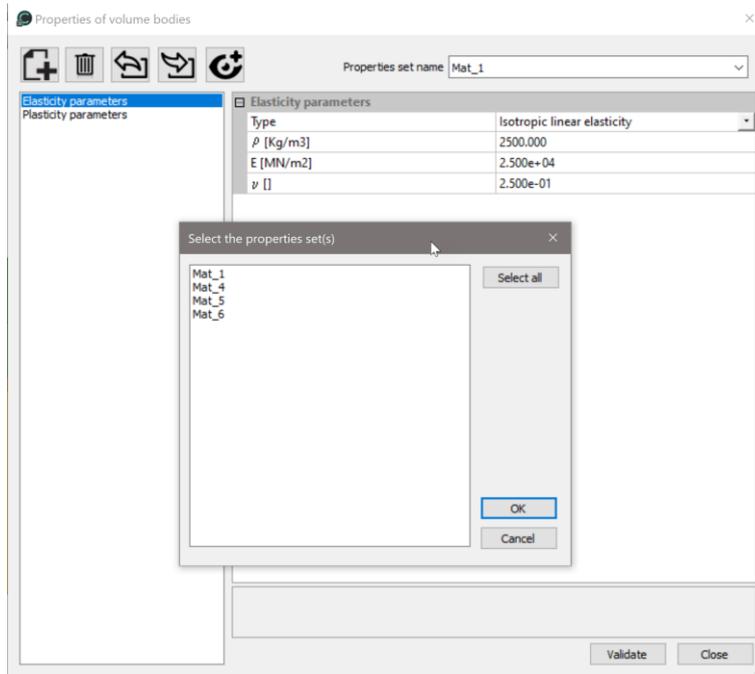
In many cases, it may be worthwhile to back up in an external library (independent of the current project) the property tables used repeatedly in several studies. To perform such a backup, simply follow the procedure outlined below.

1. Press the "Export" button in the dialogue box. This action displays a dialogue box that allows defining both the location and name of this "general library".
2. Once this information has been entered, click on the dialogue box "Save" button.
3. A new dialogue box then opens for the purpose of selecting the property tables to export into the library.



Importing property tables from an external library

1. Press the "Import" button in the dialogue box. This step displays a dialogue box that allows defining both the location and name of the "general library" containing the property table to import.
2. Entering this name triggers the opening of a dialogue box presenting the list of tables contained in the general library. Choose the table to import from this list.
3. Click on the "Validate" button to import the selected property table into the current project library.



Example of a dataset imported from an external library



Viewing and editing all the sub-tables within a property table

The tools serving to set up the properties library may be accessed in the toolbar under the "Properties" tab. This toolbar is primarily intended to define the properties of the current model chosen from the relevant drop-down list.



Active model

The current model is associated with both a specific field (Statics, Dynamics, Hydrogeology, etc.) and a solver applicable to the particular field (LINE, MCNL, MODE, etc.).

For the current model, only some of the "sub-property tables" will thus be used. In order to simplify the user's task, just those sub-tables introduced into the current model are proposed for the definition of settings. Nonetheless, the set of sub-tables for a given table may still be viewed and edited at any time according to the following procedure.

1. Choose the table to be edited from the drop-down list of available tables.
2. Click on the "View all sub-tables" button  in the dialogue box. The set of sub-tables is now accessible in the list presented on the left-hand side of the dialogue box.
3. Edit the property table settings, as indicated in the previous sections.



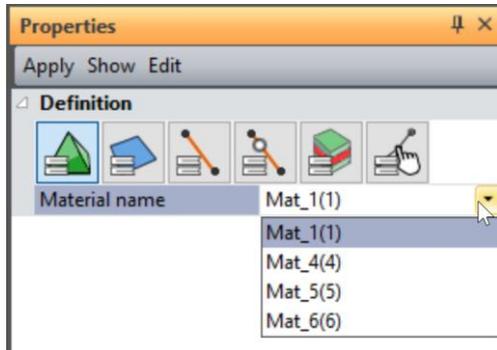
Deleting a property table

Pressing this button serves to delete the current property table. This action will only be effective once the "Validate" button in the general dialogue box has been clicked.



"Graphical" application of properties

This tool performs the standard application of a property table to the set of selected bodies. Running this tool displays a dialogue box that allows executing this operation according to the following sequence.



Applying the property tables graphically

1. Select the bodies whose property tables have been selected for application.
2. In the dialogue box grid, identify the targeted body type.
3. From the drop-down list, choose the property table to be assigned. This list only contains the property tables associated with the "body type" indicated above.
4. Click on the "Apply" button to assign the property table to the selected bodies of the defined type.

Verification of the property table assignment

Two methods are proposed herein:

1. Choose a set of properties from the list of recorded sets, then click on "Display".
2. Right-click on one of the model bodies. If a set of properties is associated, the dialogue box grid is updated by showing the type of body and the associated set of properties. Should no set of properties be associated, then the dialogue box grid is updated by continuing to display no property set name.



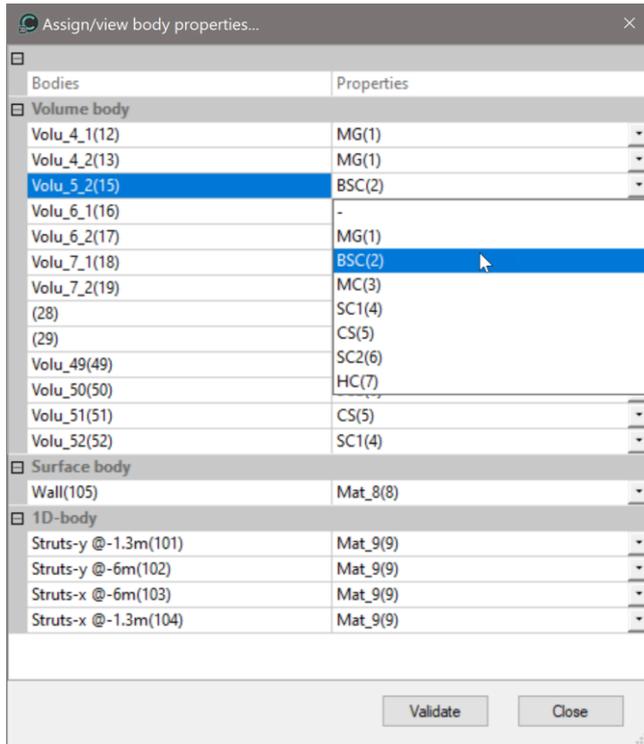
Assign/view of properties in a table

This tool allows to affect properties to a support body. All the bodies being listed per type, user will be proposed for each body the list of available properties sets.

The advantage for the user is to have an immediate overview of the bodies and associated properties, giving him the possibility for immediate correction.

Assign properties in a table

1. For each body, develop the list of available properties sets. Select the appropriate set of properties.
2. Repeat the previous operation for all the bodies for each type (volume, surface, 1D, interface...).
3. "Validate"



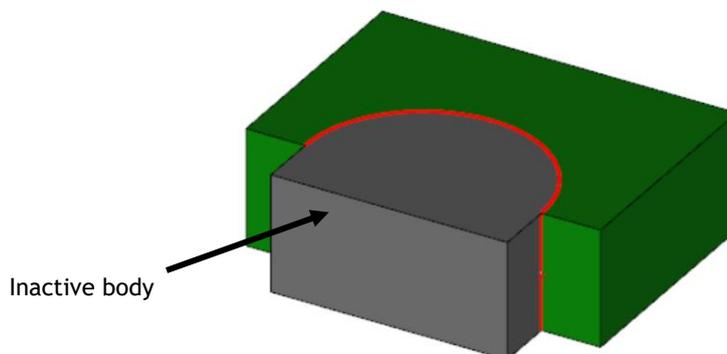
 Note here the utility of naming the bodies using the tool  **Bodies properties** in GEOMETRY.



Definition of the Active / Inactive attribute

This tool serves to assign the "Active / Inactive" attribute on the selected model bodies. In any given model, all bodies are considered by default as active. The inactive groups are visualized on the screen by a specific colour (light gray) to distinguish them more easily.

 The colour of the inactive bodies can be modified in "Application settings" > Display > Neutral colour.



Assigning the Active / Inactive attribute

1. Select the bodies whose Active / Inactive attribute is intended to be modified.
2. In the grid, choose the attribute value (i.e. Active or Inactive) to be assigned.
3. Click on the "Apply" button to complete this body attribute assignment.

10. Initialisations of parameters

Introduction

The processing of certain computation modules necessitates defining a set of "initial values". In the case of a computation module enabling, for example, the resolution of a dynamic problem, it becomes essential to specify at least the initial displacements and speeds associated with each node of the model.

The following table indicates the parameters to be incorporated for each of the computation modules requiring such initialization steps.

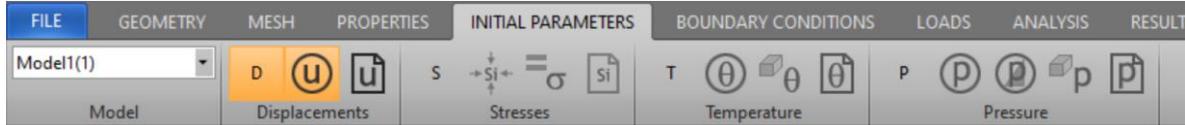
Module	Displacement	Speed	Stress	Load	Temperature	Pressure
MCNL			yes			
TCNL			yes			
DYNI	yes	yes	yes			
SUMO	yes	yes	yes			
DTLI				yes	yes	
DTNL				yes	yes	
NSAT				yes		
MEXO			yes			
TEXO					yes	
CSNL	yes		yes	yes		
MPNL	yes		yes		yes	yes

 This module is optional in that the parameters associated with the given computation module have been ascribed the default initialization listed in the table below:

Parameter	Default initialization
Displacement	0.
Speed	0.
Stresses	0.
Load	0.
Temperature	0.
Pressure	0.

Activating the "Parameter initialization" option generates:

- display of a toolbox to allow initializing the parameters included in the current model;
- display of a view representing the given model in the screen's graphics workspace. Note herein that the initial parameters are not being explicitly visualized from a graphical perspective.



The tools proposed in the toolbox are updated on the basis of the parameter selected from the multiple-choice list.

We have presented below the set of tools associated with each of the six parameters capable of being initialized.

Parameter	Tool
Displacements	Displacement field of a rigid solid Input of the displacement field from a file
Speeds	Speed field of a rigid solid Input of the speed field from a file
Stresses	Uniform stress tensor by group Geostatic stresses Input of initial stresses from a file
Load	Uniform load at any point Position-based load Input of the load from an external file Uniform load by group
Temperature	Uniform temperature at any point Input of the temperature from an external file Uniform temperature by group
Pressure	Uniform pressure at any point Position-based pressure Input of the pressure from an external file Uniform pressure by group

**Initialization of displacements by rigid body motion**

This tool serves to characterize the initial displacement of each node in the given model by means of entering a rigid body motion.

Running this tool generates a dialog box that allows inputting the following data:

- X, Y, Z: "Reference point" coordinates expressed within the global reference,
- U, V, W: Displacement vector components beginning at the reference point along the X, Y and Z axes of the global reference,
- Tx, Ty, Tz: Components within the global reference of the rotation vector at the reference point.



In the special case where the initial displacement field has already been defined by a rigid body displacement, the characteristics of this displacement are displayed in the dialog box. Otherwise, all of the fields appear blank in the dialog box.

Initializing displacements by means of rigid body motion

1. Complete the dialog box fields that enable defining a rigid body motion.
2. Click the "Apply" button.

**Initialization of displacements by file input**

This tool serves to characterize the initial displacement of each node in the given model by means of file input.

Initializing displacements by file input

1. Specify the name of the file from which the initial displacements are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.
2. Click the "Apply" button.



In the special case where the initial displacement field has already been defined by entering a file name, this name gets displayed in the dialog box. Otherwise, the "Name" field appears blank in the dialog box.

**Initialization of speeds by rigid body motion**

This tool serves to characterize the initial speed of each node in the given model by means of entering a rigid body motion.

Running this tool generates a dialog box that allows inputting the following data:

- X, Y, Z: "Reference point" coordinates expressed within the global reference,
- Vx, Vy, Vz: Speed vector components from the reference point along the X, Y and Z axes of the global reference.

Initializing speeds by means of rigid body motion

1. Complete the dialog box fields that enable defining a rigid body motion.
2. Click the "Apply" button.

-  In the special case where the initial speed field has already been defined by rigid body motion, the characteristics of such motion are displayed in the dialog box. Otherwise, all of the fields appear blank in the dialog box.



Initialization of speeds by file input

This tool serves to characterize the initial speed of each node in the given model by means of file input.

Initializing speeds by file input

1. Specify the name of the file from which the initial speeds are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.
2. Click the "Validate" button.

-  In the special case where the initial speed field has already been defined by entering a file name, this name gets displayed in the dialog box. Otherwise, the "Name" field appears blank in the dialog box.



Uniform initial stresses by group

This tool makes it possible to assign uniform initial stresses to a selected set of volume body.

Running this tool generates a dialog box that allows inputting the following data:

- S_{xx} , S_{yy} , S_{zz} , S_{xy} , S_{yz} , S_{xz} : Components of the stress tensor

Initializing stresses on the selected volume body

1. Complete the dialog box fields that enable defining the initial stress tensor.
2. Click the "Apply" button to assign this initial stress tensor to the entire set of selected volume body.

Identifying the initial stress assigned to a given volume body

1. Right click on the given volume body. In cases where the initial stress tensor of this body would have been defined by the present tool, the characteristics of this assigned tensor are displayed in the dialog box. Otherwise, the fields appear blank.

Selecting the set of bodies with a given initial stress

1. Complete the dialog box fields that enable defining the components of the given initial stress.
2. Click the "Show" button to select the set of volume body exhibiting this initial stress. Note herein that only those bodies whose initial stress has been defined using the present tool will be selected.

-  Recall herein that using this option acts to nullify all definitions entered previously, if applicable, through both the "Initial geostatic stresses" and "Initial stresses by file input" tools.



Initial geostatic stresses

The initial stresses present in the ground are, in most instances, due solely to the weight of the soil. In the special case where the ground is composed of horizontal strata, it is possible to evaluate very

straightforwardly the state of initial stresses on the basis of both soil volume weight and lateral thrust coefficients. Under such circumstances, the following relation is applicable:

$$S_{zz} = -\int_z^{zs} p_v \cdot z \cdot dz$$

$$S_{xx} = CP_x \cdot S_{zz}$$

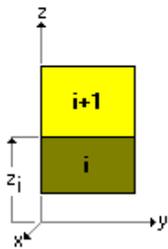
$$S_{yy} = CP_y \cdot S_{zz}$$

with:

- S_{zz} : Vertical stress at a point located in position z
- S_{xx} : Horizontal stress in the x direction at a point located in position z
- S_{yy} : Horizontal stress in the y direction at a point located in position z
- P_v : Volumic weight
- CP_x, CP_y : Lateral thrust coefficients in both the x and y directions
- Z_s : Position of the model's upper boundary

The "Initial geostatic stresses" tool thereby allows assigning initial stresses, computed as indicated above, to all of the model's volume element groups. Activating this tool opens a dialog box that primarily contains a grid for the purpose of specifying the properties of each soil stratum, which in effect requires identifying the following:

- *Altitude*: Position Z of the layer's upper boundary
- P_v : Volumic weight
- CP : Lateral thrust coefficients
- CP_x, CP_y : Lateral thrust coefficients in both the x and y directions



i Reminder: For the uppermost layer, it is possible to enter a higher position reading than that of the actual upper boundary, in order to allow margin for subsequent covering.

i In the special case where initial stresses in the volume element groups have already been defined by the present tool, the pertinent soil strata characteristics are presented in the dialog box. Otherwise, the fields appear blank in the dialog box.

Assigning initial geostatic stresses to the volume element groups

1. Complete the grid cells that enable defining the characteristics of each soil layer, which herein is characterized by a separate row of the grid.
2. The "Insert" button appearing in the dialog box allows inserting a new row in the grid just after the row indicated by the current cursor position. Conversely, the "Delete" button serves to delete the row indicated by the cursor position.
3. Click the "Validate" button to assign these initial stresses to the set of volume bodies of the model.



Initialization of stresses by file input

This tool serves to initialize stresses for the set of model elements by means of file input.

Running this tool generates a dialog box that allows inputting the following data:

- Name: Name of file from which the initial stresses for each model element are to be read.

 In the special case where the initial stresses have already been defined by the present tool, the corresponding file name gets displayed in the dialog box. Otherwise, the "Name" field appears blank in the dialog box.

Initializing stresses by file input

1. Indicate the name of the file from which the initial stresses are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.
2. Click the "Validate" button.



Uniform initialization of loads at any point

This tool serves to define a uniform initial load at any point.

Activating this tool opens a dialog box that enables inputting the desired initial load.

 In the special case where the initial loads have already been defined by the present tool, the assigned uniform initial load gets displayed in the dialog box. Otherwise, the "Load" field appears blank in the dialog box.

Initializing loads uniformly at any point

1. Complete the dialog box field that allows specifying the initial load.
2. Click the "Validate" button.



Position-based load initialization

This tool serves to define an initial load that depends on the position z of each model node.

Running this tool generates a dialog box that enables entering the following data:

- Position z_1 : Position of the points featuring initial load h_1
- Load h_1 : Initial load of the points located in position z_1
- Position z_2 : Position of the points featuring initial load h_2 .
- Load h_2 : Initial load of the points located in position z_2 .

In light of these data, the initial load at a point located in position z is simply obtained by means of linear interpolation, as follows:

$$h = \frac{h_1 - h_2}{z_1 - z_2} (z - z_1) + h_1$$

 In the special case where the initial loads have already been defined using this tool, the values inputted for parameters h_1 , z_1 , h_2 and z_2 are displayed in the corresponding dialog box fields. Otherwise, the pertinent dialog box fields appear blank.

Initializing loads on the basis of position

1. Complete dialog box fields h_1 , z_1 , h_2 and z_2 that enable specifying position-dependent initial loads.
2. Click the "Validate" button.



Uniform initial load by body

This tool makes it possible to assign uniform initial loads on all nodes contained within the selected volume bodies.

Running this tool generates a dialog box that allows entering the initial load.

i Recall herein that selecting this option acts to nullify all definitions entered previously, if applicable, through any one of the three other tools used for deriving a global definition of the initial load.

Initializing load on nodes belonging to the selected volume body

1. Complete the dialog box field that enables defining the initial load.
2. Click the "Apply" button to assign this initial load to all nodes belonging to the selected volume bodies.

Identifying the initial load assigned to the nodes of a given volume body

1. Right click on the given volume body. In cases where the initial load of this group's nodes would have been defined by the present tool, the value of this assigned initial load is displayed in the dialog box. Otherwise, the field appears blank.

Selecting the set of bodies whose nodes display a given initial load

Complete the dialog box field that enables defining the given initial load.

1. Click the " Show " button to select the set of volume body whose nodes display this initial load. Note herein that only those bodies whose initial load has been defined using the present tool will be selected.



Initialization of loads by file input

This tool serves to initialize the loads of all model nodes by means of file input.

Running this tool opens a dialog box that enables inputting the following data:

- Name: Name of file from which the initial loads for each model node are to be read.

i In the special case where the initial loads have already been defined by the present tool, the corresponding file name is displayed in the dialog box. Otherwise, the "Name" field appears blank in the dialog box.

Initializing loads by file input

1. Indicate the name of the file from which the initial loads are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.
2. Click the "Validate" button.



Uniform initialization of temperatures at any point

This tool serves to define a uniform initial temperature at any point.

Activating this tool opens a dialog box that enables inputting the desired initial temperature.

-  In the special case where the initial temperatures have already been defined by the present tool, the assigned uniform initial temperature gets displayed in the dialog box. Otherwise, the "Temperature" field appears blank in the dialog box.

Initializing temperatures uniformly at any point

1. Complete the dialog box field that allows specifying the initial temperature.
2. Click the "Validate" button.



Uniform initial temperature by body

This tool makes it possible to assign uniform initial temperatures on all nodes contained within the selected volume bodies.

Activating this tool opens a dialog box that allows inputting the initial temperature.

-  Recall herein that selecting this option acts to nullify all definitions entered previously, if applicable, through any one of the three other tools used for deriving a global definition of the initial temperature.

Initializing temperature on nodes belonging to the selected volume bodies

1. Complete the dialog box field that enables defining the initial temperature.
2. Click the "Apply" button to assign this initial temperature to all nodes belonging to the selected volume bodies.

Identifying the initial temperature assigned to the nodes of a given volume body

1. Right click on the given volume body. In cases where the initial temperature of this group's nodes would have been defined by the present tool, the value of this assigned initial temperature is displayed in the dialog box. Otherwise, the field appears blank.

Selecting the set of bodies whose nodes display a given initial temperature

1. Complete the dialog box field that enables defining the given initial temperature.
2. Click the "Show" button to select the set of volume bodies whose nodes display this initial temperature. Note herein that only those bodies whose initial temperature has been defined using the present tool will be selected.



Initialization of temperatures by file input

This tool serves to initialize the temperatures of all model nodes by means of file input.

Running this tool opens a dialog box that enables inputting the following data:

- Name: Name of file from which the initial temperatures for each model node are to be read.

-  In the special case where the initial temperatures have already been defined by the present tool, the corresponding file name gets displayed in the dialog box. Otherwise, the "Name" field appears blank in the dialog box.

Initializing temperatures by file input

1. Indicate the name of the file from which the initial temperatures are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.

2. Click the "Validate" button.



Uniform initialization of pressures at any point

This tool serves to define a uniform initial pressure at any point.

Running this tool opens a dialog box that enables inputting the desired initial pressure.



In the special case where the initial pressures have already been defined by the present tool, the assigned uniform initial pressure gets displayed in the dialog box. Otherwise, the "Pressure" field appears blank in the dialog box.

Initializing pressures uniformly at any point

1. Complete the dialog box field that allows specifying the initial pressure.
2. Click the "Validate" button.



Position-based pressure initialization

This tool serves to define an initial pressure that depends on the position z of each model node.

Running this tool generates a dialog box that enables entering the following data:

- Position z_1 : Position of the points featuring initial pressure p_1
- Pressure p_1 : Initial pressure of the points located in position z_1
- Position z_2 : Position of the points featuring initial pressure p_2
- Pressure p_2 : Initial pressure of the points located in position z_2 .

In light of these data, the initial pressure at a point located in position z is simply obtained by means of linear interpolation, as follows:

$$p = \frac{p_1 - p_2}{z_1 - z_2} (z - z_1) + p_1$$



In the special case where the initial pressures have already been defined using this tool, the values inputted for parameters z_1 , p_1 , z_2 and p_2 are displayed in the corresponding dialog box fields. Otherwise, the pertinent dialog box fields appear blank.

Initializing pressures on the basis of position

1. Complete dialog box fields z_1 , p_1 , z_2 and p_2 that enable specifying position-dependent initial pressures.
2. Click the "Validate" button.



Uniform initial pressure by body

This tool makes it possible to assign uniform initial pressures on all nodes contained within the selected volume bodies.

Activating this tool opens a dialog box that allows inputting the initial pressure.

i Recall herein that selecting this option acts to nullify all definitions entered previously, if applicable, through any one of the three other tools used for deriving a global definition of the initial pressure.

Initializing pressure on nodes belonging to the selected volume bodies

1. Complete the dialog box field that enables defining the initial pressure.
2. Click the "Apply" button to assign this initial pressure to all nodes belonging to the selected volume bodies.

Identifying the initial pressure assigned to the nodes of a given volume body

1. Right click on the given volume body. In cases where the initial pressure of this body's nodes would have been defined by the present tool, the value of this assigned initial pressure is displayed in the dialog box. Otherwise, the field appears blank.

Selecting the set of bodies whose nodes display a given initial pressure

1. Complete the dialog box field that enables defining the given initial pressure.
2. Click the "Show" button to select the set of volume bodies whose nodes display this initial pressure. Note herein that only those bodies whose initial pressure has been defined using the present tool will be selected.



Initialization of pressures by file input

This tool serves to initialize the pressures of all model nodes by means of file input.

Running this tool opens a dialog box that enables entering the following data:

- Name: Name of file from which the initial pressures for each model node are to be read.

i In the special case where the initial pressures have already been defined by the present tool, the corresponding file name gets displayed in the dialog box. Otherwise, the "Name" field appears blank.

Initializing pressures by file input

1. Indicate the name of the file from which the initial pressures are to be read in the "Name" field. This file name may also be obtained in the habitual manner by using the "Open" button.
2. Click the "Validate" button.

1 1. Definition of a set of boundary conditions

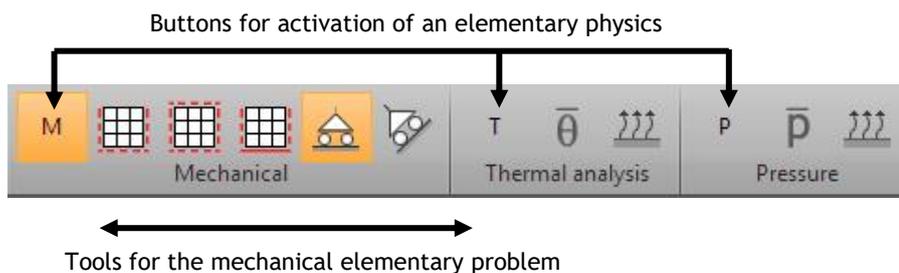
Introduction

The "Boundary conditions" tab serves to define the characteristics of the current set of boundary conditions.

Activating this option displays:

- a toolbox for entering the boundary conditions associated with the current set;
- a view, within the graphics zone of the screen, representing the model with the previously-defined boundary conditions.

In order to account for the eventual multiplicity of elementary physical problems, the toolbox serving to define boundary conditions will be updated to appear as follows:



As shown in the above figure, the toolbox has been decomposed into as many entities as there are elementary physical problems for the particular computation module (cf. Chapter "Calculation modules and elementary physical problems").

At any given point in time, just one elementary physical problem is "active". Clicking on the activation button of an elementary physical problem serves to:

- activate the tools associated with the chosen elementary problem;
- "deactivate" the tools associated with the other elementary physical problems;
- display, within the graphics zone, the boundary conditions associated with the active elementary physical problem.

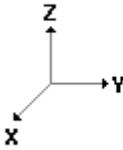
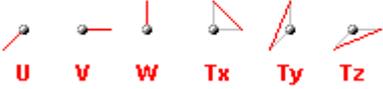
The set of tools associated with each of the potentially-applicable elementary physical problems have been listed below.

Elementary physical problem	Tools
Mechanics (M)	 Sides and bottom ($w=0$) supports  Sides and bottom ($u=v=w=0$) supports  Sides, top and bottom ($w=0$) supports

Definition of a set of boundary conditions

	 General definition of imposed displacements  Coordinate system changes for supports
Hydrogeology (H)	 Imposed load  Imposed load varying linearly with depth  Seepage conditions  Exchange conditions
Heat transfer (T)	 Imposed temperature  Exchange conditions
Pressure (P)	 Imposed pressure  Exchange conditions

The inclusion of nodal boundary conditions is visualized on the screen by means of a symbol. The following table provides definitions of the various symbols used.

Imposed displacements	  <p>Given the use of this convention, the special cases of "Simple support" and "Embedment" will be visualized at each node by means of the following symbols:</p>  Simple support  Embedment
Imposed load	
Imposed temperature	
Imposed pressure	

For any given model, it is in certain cases possible to define several sets of boundary conditions. The following table presents the authorized number of sets of boundary conditions for each of the computation modules available in the current version.

LIN E	MCN L	TCN L	DYN I	LIN C	LIN H	MOD E	SUM O	DTN L	NSA T	TEX O	MEX O	CSL I	MPN L
1	≥1	≥1	≥1	≤2	1	1	1	≥1	1	≥1	≥1	≥1	≥1

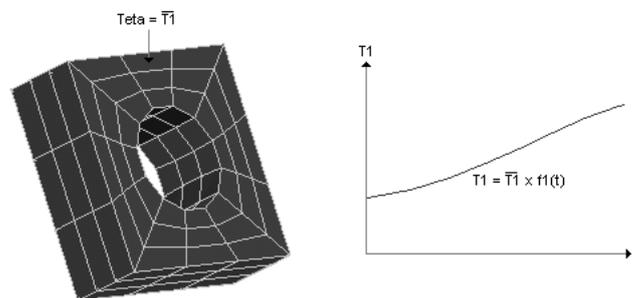
As remarked in the above table, the possibility of establishing several sets of boundary conditions basically applies to those modules that allow performing either non-linear computations (i.e. involving several increments) or "time function" computations (i.e. involving several time steps).

In order to illustrate this functional feature, let's first consider a transient thermal problem (the DTLI computation module).

Let's assume that the problem is characterized by the following boundary conditions:

- A first set of nodes at an imposed temperature whose variation vs. time is defined by the function $T_1(t) = \bar{T}_1 \cdot f_1(t)$,
- A second set of nodes at an imposed temperature whose variation vs. time is defined by the function $T_2(t) = \bar{T}_2 \cdot f_2(t)$.

In order to model the fact that two distinct time functions are involved, two sets of boundary conditions will be required.



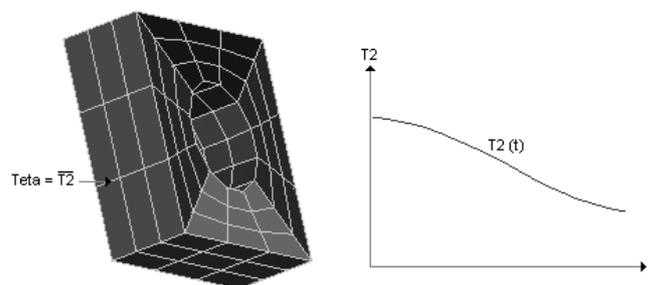
Set of boundary conditions 1

In general, all conditions displaying the same time-based evolution will be structured within each set in a way that allows the imposed temperatures to be expressed in the following form:

$$\{T_1(t)\} = \{\bar{T}_1\} \cdot f_1(t)$$

with:

3. $\{\bar{T}_1\}$: Vector of imposed temperatures in the given set of boundary conditions;
4. $f_1(t)$: Time function - This function is defined in the data characterizing the computation module (see chapter on "Computation Data").



Set of boundary conditions 2

The incorporated boundary conditions will be obtained by "superimposing" the conditions contributed by each set. The conditions imposed at time t will be such that:

$$\{T(t)\} = \{\bar{T}_1\} \cdot f_1(t) + \{\bar{T}_2\} \cdot f_2(t)$$

Creation of a new set of boundary conditions

Let's note here that this function will obviously only be accessible provided the computation module associated with the given model enables defining several sets of boundary conditions.

Creating a new set of boundary conditions

1. Activate the menu item "Creation of a set of boundary conditions" by right-clicking on the "Boundary conditions" option of the given model within the "Models" tree diagram. This activation step displays a dialogue box that allows inputting the "name" assigned to the new set of boundary conditions.
2. In the "Name" box, enter the label of the set of boundary conditions to be initialized.
3. If applicable, choose a reference set from the "Source" drop-down list. Use of this option will lead to creating a set of boundary conditions initialized with the set of all reference conditions. To characterize this new set, simply modify as needed the data dedicated to this set.
4. Press "Validate". The set of boundary conditions initialized in this manner now becomes the current set.

Deleting a set of boundary conditions

To delete a set of previously defined boundary conditions, simply follow the procedure outlined below.

1. Activate the menu item entitled "Delete a set of boundary conditions" by right-clicking on the name of the given set of boundary conditions in the "Models" tree diagram.



Mechanics: Sides and bottom ($w=0$) supports

Activating this tool automatically imposes the following conditions:

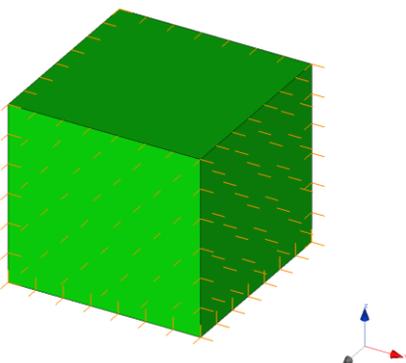
- $U = 0$ on the set of visible faces (in G-Mode) or nodes (in M-Mode) with coordinates $x=x_{min}$ or $x=x_{max}$,
- $V = 0$ on the set of visible faces (in G-Mode) or nodes (in M-Mode) with coordinates $y=y_{min}$ or $y=y_{max}$,
- $W = 0$ on the set of visible faces (in G-Mode) or nodes (in M-Mode) with coordinates $z=z_{min}$,

where:

- U: displacement along the X axis,
- V: displacement along the Y axis.
- W: displacement along the Z axis.



If the model includes plates and/or beams that extend to the boundaries previously defined, rotations will be fixed.





Mechanics: Sides and bottom ($u=v=w=0$) supports

L'activation de cet outil entraîne l'imposition automatique des conditions :

- $U = 0$ sur l'ensemble des nœuds de coordonnées $x=x_{\min}$ ou $x=x_{\max}$
- $V = 0$ sur l'ensemble des nœuds de coordonnées $y=y_{\min}$ ou $y=y_{\max}$
- $U = 0$ sur l'ensemble des nœuds de coordonnées $z=z_{\min}$

$V = 0$ sur l'ensemble des nœuds de coordonnées $z=z_{\min}$

$W = 0$ sur l'ensemble des nœuds de coordonnées $z=z_{\min}$

where:

- U: displacement along the X axis,
- V: displacement along the Y axis.
- W: displacement along the Z axis.



If the model includes plates and/or beams that extend to the boundaries previously defined, rotations will be fixed.



Mechanics: Sides, top and bottom ($w=0$) supports

Activating this tool automatically imposes the following conditions:

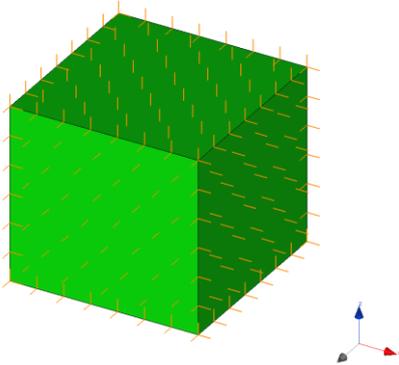
- $U = 0$ on the set of visible nodes with coordinates $x=x_{\min}$ or $x=x_{\max}$,
- $V = 0$ on the set of visible nodes with coordinates $y=y_{\min}$ or $y=y_{\max}$.
- $W = 0$ on the set of visible nodes with coordinates $z=z_{\min}$ or $z=z_{\max}$.

where:

- U: displacement along the X axis,
- V: displacement along the Y axis.
- W: displacement along the Z axis.



If the model includes plates and/or beams that extend to the boundaries previously defined, rotations will be fixed.



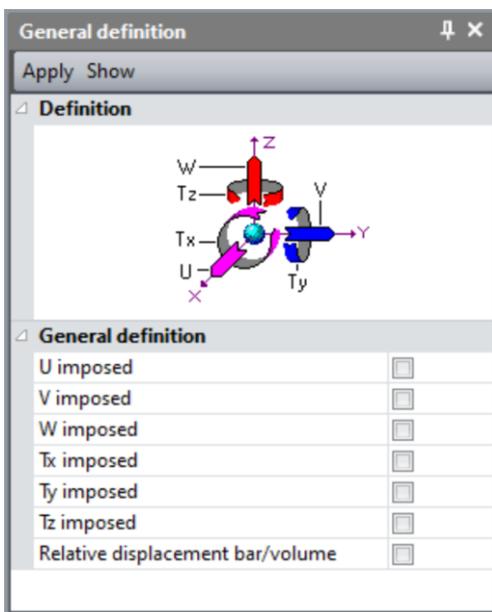


Mechanics: General definition of imposed displacements

This tool enables assigning the support conditions existing at certain nodes of the structure. Imposed displacement- or rotation-type conditions (either zero or nonzero) may be specified for each degree of freedom at the nodes of the structure.

The degrees of freedom associated with each node consist of the three displacements denoted U, V and W along the X, Y and Z axes of the global coordinate system and, eventually, the three rotations Tx, Ty and Tz around these same axes.

If the model contains 1D bodies with the type of "Friction bar", then an additional 7th degree of freedom is proposed: relative displacement between the 1D body and the surrounding surface body.



i Rotation parameters are only activated if the model contains beam- or shell-type "structural elements".

i It should be pointed out that conditions may be imposed in any coordinate system provided a system specific to the studied nodes has been defined using the "Coordinate system changes" tool.

Assigning the support conditions

1. Activate the "General definition of imposed displacements" tool.
2. Select the targets (any type of body part in G-Mode, any node in M-Mode) that will have the same type of boundary conditions.
3. Define the imposed condition for each of the three (or six) degrees of freedom.
4. Press the "Apply" button to assign these support conditions to the selected set.

The figure below shows an example of the definitions entered for the various types of conditions applicable to displacement parameter U.

- | | |
|-----------|--------------------------|
| U imposed | <input type="checkbox"/> |
|-----------|--------------------------|

 No condition
- | | |
|-----------|-------------------------------------|
| U imposed | <input checked="" type="checkbox"/> |
| U [mm] | 0.000 |

 Displacement U imposed at a zero value
- | | |
|-----------|-------------------------------------|
| U imposed | <input checked="" type="checkbox"/> |
| U [mm] | 0.015 |

 Displacement U imposed at a nonzero value

 The conditions imposed on the rotation parameters will only be effective on those nodes containing rotation degree of freedom.

Identifying the support conditions assigned

1. Activate the "General definition of imposed displacements" tool.
2. Right click on the target (any type of body element in G-Mode, any node in M-Mode). The support conditions assigned to the clicked target are then displayed in the dialog box.

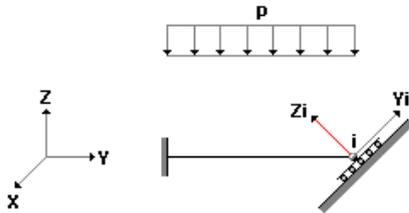
Visualizing all nodes featuring the same support conditions

1. Activate the "General definition of imposed displacements" tool.
2. Enter in the dialog box the particular support conditions.
3. Press the "Show" button. All targets (any type of body part in G-Mode, any node in M-Mode) featuring the entered boundary conditions are then selected.



Mechanics: Coordinate system changes

Support conditions are imposed by default for the degrees of freedom expressed in the global coordinate system of the structure. In order to be able to impose conditions that are not easily expressed in the global system (e.g. inclined plane support), it must be possible to define a system associated with the nodes supporting such conditions. The tool analysed herein is intended for this purpose.



Assigning coordinate system changes

1. Select the nodes for which the same coordinate system is to be applied.
2. Activate the "Coordinate system change" tool.
3. Enter the components, within the global coordinate system, of the two vectors V_x and V_y that serve to specify directions X and Y of the system to be defined. On the basis of these two vectors, an orthonormal trihedral is constituted as follows:

$$\vec{e}_x = \frac{\vec{V}_x}{\|\vec{V}_x\|} \quad \vec{e}_z = \frac{\vec{V}_x \wedge \vec{V}_y}{\|\vec{V}_x \wedge \vec{V}_y\|} \quad \vec{e}_y = \vec{e}_z \wedge \vec{e}_x$$

4. Press the "Apply" button to assign the defined coordinate system to the set of selected nodes. Inclusion of these coordinate systems is visualized on the screen by the display of a trihedral that respects the indicated directions.

Identifying the coordinate system assigned at a node

Activate the "Coordinate system change" tool.

1. Right click on the target node. The characteristics of the coordinate system assigned to the clicked node are then displayed in the dialog box.

Visualizing all nodes featuring the same change in coordinate system

Activate the "Coordinate system change" tool.

1. Enter in the dialog box the characteristics of the given coordinate system change.
2. Press the " Show" button. All nodes exhibiting this coordinate system change are then selected.



Hydrogeology: Imposed loads

This tool serves to assign a constant imposed load on the set of selected model nodes.

Assigning an imposed load

1. Select the targets (any type of body part in G-Mode, any node in M-Mode).displaying the same imposed load.
2. Activate the "Imposed load" tool.
3. Enter the value of the imposed load.
4. Press the "Apply" button to assign this imposed load to the selected set.

Identifying the imposed load at a node

Activate the "Imposed load" tool.

Right click on the target node. The imposed load at the clicked target (any type of body part in G-Mode, any node in M-Mode) is then displayed in the dialog box.

Visualizing all nodes featuring the same imposed load

1. Activate the "Imposed load" tool.
2. Enter the given imposed load in the dialog box.
3. Press the " Show " button. All targets (any type of body part in G-Mode, any node in M-Mode) featuring this imposed load are then selected.



Hydrogeology: Imposed loads varying linearly with depth

This tool allows assigning an imposed load that varies linearly with depth z on the set of selected model nodes. The variation in load as a function of z is established by data consisting of two side lengths z and the corresponding loads.

Assigning an imposed load varying linearly as a function of depth

1. Select the targets (any type of body part in G-Mode, any node in M-Mode) at which a depth-varying load is to be imposed.
2. Activate the "Imposed linearly-varying load" tool.
3. Enter two (different) side lengths z and the corresponding loads.
4. Press the "Apply" button to assign this imposed load to the selected set.



Please note that the imposed load may be identified at each target (any type of body part in G-Mode, any node in M-Mode) by use of the general "Imposed load" tool.



Hydrogeology: Seepage conditions

Activating this tool leads to the inclusion of a seepage condition applicable to the set of selected facets.



Hydrogeology: Exchange conditions

This tool enables assigning linear or non-linear exchange conditions on the sets of selected facets.

Assigning linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the load at infinity "Cinf".
4. Choose the "Linear" menu option.
5. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.

Assigning non-linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the load at infinity "Cinf".
4. Choose the "Non-linear" menu option.
5. In the proposed table, specify values for the associated "Load" and "**Relative** exchange coefficient" couples characterizing the non-linearity of the exchange.
6. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.



Note herein that this table requires the input of a relative exchange coefficient. For a given load, the incorporated exchange coefficient will thereby equal the product of the reference value "he" multiplied by the relative exchange coefficient obtained from interpolation using the values provided in this table.

Identifying the exchange conditions assigned on a facet

1. Activate the "Exchange conditions" tool.
2. Right click on the target facet. The characteristics of the exchange assigned on the clicked facet are then displayed in the dialog box.

Visualizing all facets featuring the same exchange conditions

Activate the "Exchange conditions" tool.

Enter in the dialog box the characteristics of the given exchange.

1. Press the "Show" button. All facets displaying these exchange conditions are then selected.



Thermics: Imposed temperatures

This tool serves to assign a constant imposed temperature on the set of selected model nodes.

Assigning an imposed temperature

1. Select the targets (any type of body part in G-Mode, any node in M-Mode) on which the same imposed temperature will be imposed.
2. Activate the "Imposed temperature" tool.
3. Enter the value of the imposed temperature.
4. Press the "Apply" button to assign this imposed temperature to the selected set.

Identifying the imposed temperature at a node

Activate the "Imposed temperature" tool.

Right click with the mouse on the target. The imposed temperature at the clicked node is then displayed in the dialog box.

Visualizing all nodes featuring the same imposed temperature

1. Activate the "Imposed temperature" tool.
2. Enter in the dialog box the given imposed temperature.
3. Press the "Show" button. All targets (any type of body part in G-Mode, any node in M-Mode) featuring this imposed temperature are then selected.



Heat transfer: Exchange conditions

This tool enables assigning linear or non-linear exchange conditions on the sets of selected facets.

Assigning linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the temperature at infinity "Tinf".
4. Choose the "Linear" menu option.
5. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.

Assigning non-linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the temperature at infinity "Tinf".
4. Choose the "Non-linear" menu option.
5. In the proposed table, specify values for the associated "Temperature" and "**Relative** exchange coefficient" couples characterizing the non-linearity of the exchange.
6. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.



Note herein that this table requires the input of a relative exchange coefficient. For a given temperature, the incorporated exchange coefficient will thereby equal the product of the reference value "he" multiplied by the relative exchange coefficient obtained from interpolation using the values provided in this table.

Identifying the exchange conditions assigned on a facet

1. Activate the "Exchange conditions" tool.
2. Right click on the target facet. The characteristics of the exchange assigned on the clicked facet are then displayed in the dialog box.

Visualizing all facets featuring the same exchange conditions

1. Activate the "Exchange conditions" tool.
2. Enter in the dialog box the characteristics of the given exchange.
3. Press the " Show " button. All facets featuring these exchange conditions are then selected.



Pressure: Imposed pressures

This tool serves to assign a constant imposed pressure on the set of selected model nodes.

Assigning an imposed pressure

1. Select the targets (any type of body part in G-Mode, any node in M-Mode) on which the same imposed pressure will be imposed.
2. Activate the "Imposed pressure" tool.
3. Enter the value of the imposed pressure.
4. Press the "Apply" button to assign this imposed pressure to the selected set.

Identifying the imposed pressure at a node

Activate the "Imposed pressure" tool.

Right click on the target node. The imposed pressure at the clicked node is then displayed in the dialog box.

Visualizing all nodes featuring the same imposed pressure

1. Activate the "Imposed pressure" tool.
2. Enter in the dialog box the given imposed pressure.
3. Press the " Show " button. All targets (any type of body part in G-Mode, any node in M-Mode) featuring this imposed pressure are then selected.



Pressure: Exchange conditions

This tool enables assigning linear or non-linear exchange conditions on the sets of selected facets.

Assigning linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the pressure at infinity "Pinf".
4. Choose the "Linear" menu option.
5. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.

Assigning non-linear exchange conditions

1. Select the facets characterized by the same exchange condition.
2. Activate the "Exchange conditions" tool.
3. Enter the value of both the exchange coefficient "he" and the pressure at infinity "Pinf".
4. Choose the "Non-linear" menu option.
5. In the proposed table, specify values for the associated "Pressure" and "**Relative** exchange coefficient" couples characterizing the non-linearity of the exchange.
6. Press the "Apply" button to assign the defined exchange condition to the set of selected facets.



Note herein that this table requires the input of a relative exchange coefficient. For a given pressure, the incorporated exchange coefficient will thereby equal the product of the reference value "he"

multiplied by the relative exchange coefficient obtained from interpolation using the values provided in this table.

Identifying the exchange conditions assigned on a facet

1. Activate the "Exchange conditions" tool.
2. Right click on the target facet. The characteristics of the exchange assigned on the clicked facet are then displayed in the dialog box.

Visualizing all facets featuring the same exchange conditions

Activate the "Exchange conditions" tool.

Enter in the dialog box the characteristics of the given exchange.

1. Press the "Show" button. All facets featuring these exchange conditions are then selected.

12. Definition of a load sets

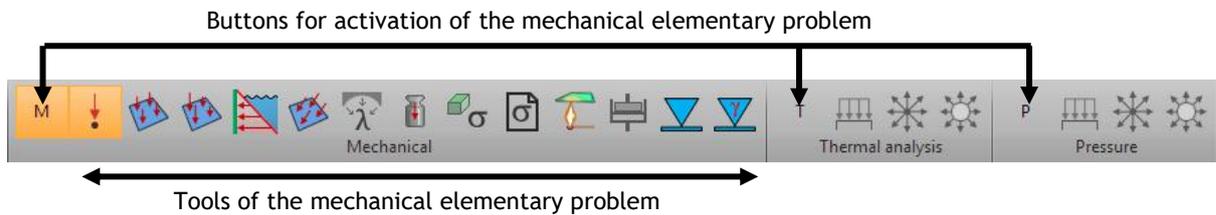
Introduction

The "Loads" tab serves to define the characteristics of the current load set.

Activating this option displays:

- a toolbox to allow entering load conditions for the current case;
- a view, within the graphics zone of the screen, representing the model with all previously-defined load conditions.

In order to account for the eventual multiplicity of elementary physical problems, the toolbox used for defining a load case will be designed to appear as follows:



As shown in the above figure, the toolbox has been decomposed into as many entities as there are elementary physical problems for the particular computation module (cf. Chapter "Calculation modules and elementary physical problems").

At any given point in time, just one elementary physical problem is "active". Clicking on the activation button of an elementary physical problem serves to:

- activate the tools associated with the chosen elementary problem;
- "deactivate" the tools associated with the other elementary physical problems;
- display, within the graphics zone, the load conditions associated with the active elementary problem.

The set of tools associated with each of the potentially-applicable elementary physical problems have been listed below.

Elementary physical problem	Tools
Mechanics (M)	<ul style="list-style-type: none"> Nodal forces Uniformly-distributed pressure Non uniformly distributed pressure Surface forces Hydrostatic pressure Excavation forces Gravitational forces

	 Constant stresses  Stress field input from a file  Thermal stresses  Long-term effects  Water table movements
Hydrogeology (H)	 Uniformly-distributed flow  Point flow rate  Volumic flow rate
Thermics (T)	 Uniformly-distributed flow  Point flow rate  Volumic flow rate
Pressure (P)	 Uniformly-distributed flow  Point flow rate  Volumic flow rate

For any given model, it is in certain cases possible to define several "load cases". The following table presents the authorized number of load cases for each of the computation modules available in the current version.

LINE	MCNL	TCNL	DYNI	LINC	LINH	MODE	SUMO	DTNL	NSAT	TEXO	MEXO	CSNL	MPNL
≥1	≥1	≥1	≥1	≤2	≥1	0	1	≥1	1	≥1	≥1	≥1	≥1

i In the preceding table, it is necessary to distinguish between the two following cases:

a) Linear problems (Modules LIGC, LINE and LINH)

In the special case where the computation module associated with the given model enables solving a linear problem (modules LIGC, LINE and LINH), each load case corresponds to an "independent" problem.

For each load case i , a problem capable of being placed in the following schematic form will actually be solved:

$$[K]\{U_i\} = \{F_i\}$$

b) Non-linear problems or time functions

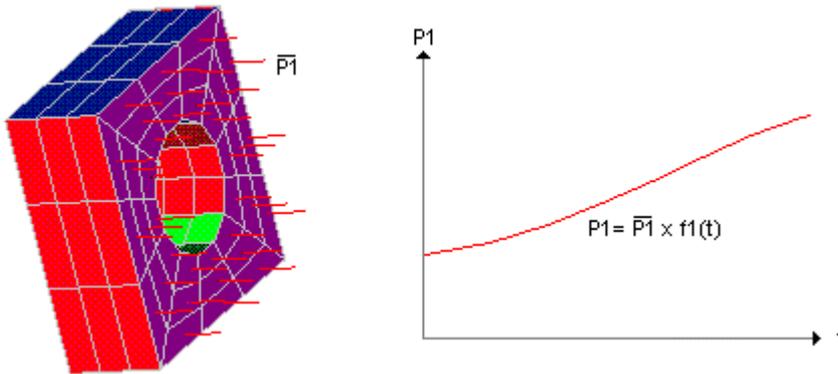
The possibility of defining several load cases is also used both for modules that allow conducting non-linear computations (containing several increments) and for modules enabling "time function" computations (containing several time steps). In this instance, the load cases serve to establish the appropriate "loading" at each increment (or time step).

To illustrate this functional feature, let's consider a transient dynamic problem (e.g. the DYNI computation module).

Let's assume for the purpose of illustration that the time-based loading applied to the structure is characterized as follows:

- A first set of facets is submitted to an imposed pressure whose variation vs. time is defined by the function $P_1(t) = \bar{P}_1 \cdot f_1(t)$;
- A second set of facets is submitted to an imposed pressure whose variation vs. time is defined by the function $P_2(t) = \bar{P}_2 \cdot f_2(t)$.

In order to model the fact that two distinct time functions are involved, two sets of load cases will be required.



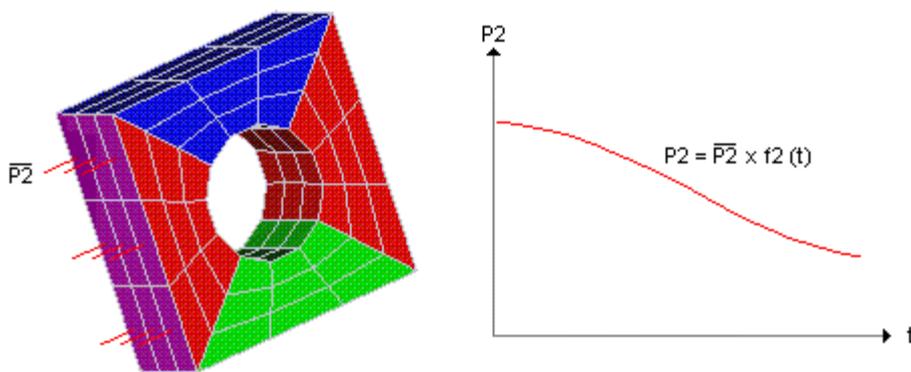
Load case 1

In general, all conditions displaying the same time-based evolution will be defined within each load case such that the applied "force vector" corresponding to this particular case can be expressed in the following form:

$$\{F_1(t)\} = \{\bar{F}_1\} \cdot f_1(t)$$

with:

- $\{\bar{F}_1\}$: Vector of imposed forces in the given load case;
- $f_1(t)$: Time function - This function is defined in the data characterizing the computation module (see chapter on "Computation Data").



Load case 2

The incorporated load conditions will be obtained by "superimposing" the conditions contributed by each case. Load conditions imposed at time t will be such that:

$$\{F(t)\} = \{\bar{F}_1\} \cdot f_1(t) + \{\bar{F}_2\} \cdot f_2(t)$$

Creation of a new load case

Let's note here that this function will obviously only be accessible provided the computation module associated with the given model enables defining several sets of load cases.

Creating a new load case

1. Activate the menu item "Creation of a load case" by right-clicking on the "Load case" option of the given model within the "Models" tree diagram. This activation step displays a dialogue box that allows inputting the "name" assigned to the new load case.
- 2.
-  As a default, the software assigns a predefined label to each load case created. The user has the opportunity to modify this name and personalize it using the [F2] key.
3. In the "Name" box, enter the label of the load case to be initialized.
4. If applicable, choose a reference set from the "Source" drop-down list. Use of this option will lead to creating a load case initialized with the set of all reference loads. To characterize this new set, simply modify as needed the data dedicated to this set.
5. Press "Validate". The load case initialized in this manner now becomes the current case.

Deleting a load case

To delete a previously defined load case, simply follow the procedure outlined below.

1. Activate the "Delete a load case" menu item by right-clicking on the name of the given load case in the "Models" tree diagram.
-  Please note here the impossibility of deleting the active load case.

Listing the loadings defined by each load case

Several loadings may be defined per each load case. The user has the possibility to visualize these loadings by using the "data tree diagram".

Activating this display enables visualizing the set of defined "loading" entities.

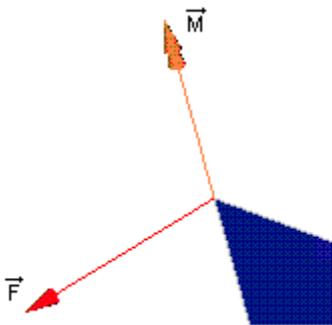
Clicking on one of the loadings activates the selection of elements assigned by this loading.



• **Mechanics: Nodal forces**

This tool serves to assign "nodal forces" to the set of selected points. A nodal force is characterized by the following components:

- F_x, F_y, F_z : Components of the force vector to be applied;
- M_x, M_y, M_z : Components of the moment vector to be applied. This vector only exerts an effect on those points having rotational degrees of freedom.



Both the force vector (F) and moment vector (M) are represented on the screen by a series of arrows with lengths proportional to the vector intensity.

Assigning nodal forces

1. Select the points possessing identical nodal forces.
2. Activate the "Nodal forces" tool.
3. Enter the components of both the force and moment vectors to be applied.
4. Press the "Apply" button to assign these nodal forces to the selected set.



Moments will only exert an effect on nodes possessing rotational parameters.

Identifying components of the nodal force assigned at a node

1. Activate the "Nodal forces" tool.
2. Right click on the target point. The components of the nodal force assigned at the clicked node are then displayed in the dialog box.



If no load is associated to the target node, then a warning message (#1515) is displayed.

Visualizing all nodes featuring the given nodal forces

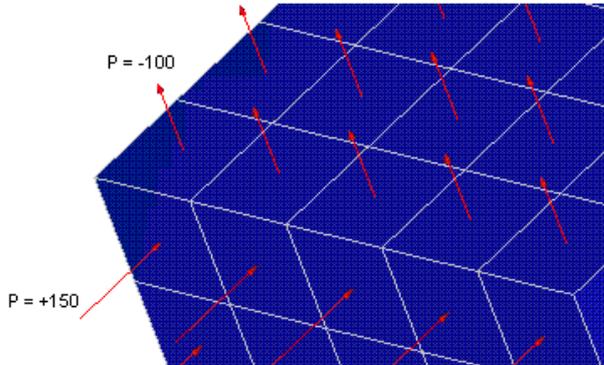
1. Activate the "Nodal forces" tool.
2. Enter the components of the indicated nodal force in the dialog box.
3. Press the "Show" button. All points featuring the given nodal force are then selected.



Mechanics: Uniformly-distributed pressure

This tool allows assigning uniform pressures on either the facets of selected surface bodies or the external facets of volume bodies.

Pressures are represented on the screen by a series of arrows positioned at the center of each facet with lengths proportional to the pressure intensity.



i For the external facets of volume bodies, the pressure is considered to be positive if directed towards the interior of the volume. The arrow depicted on the screen indicates the direction of the incorporated pressure.

Assigning a uniformly-distributed pressure

1. Select the facets possessing the same uniform pressure.
2. Activate the "Uniform pressure distribution" tool.
3. Enter the value of the imposed pressure.
4. Press the "Apply" button to assign this pressure to the set of selected facets.

Identifying the pressure imposed on a facet

1. Activate the "Uniform pressure distribution" tool.
2. Right click on the target facet. The pressure imposed on the clicked facet is then displayed in the dialog box.

i If no load is associated to the target surface, then a warning message (#1515) is displayed.

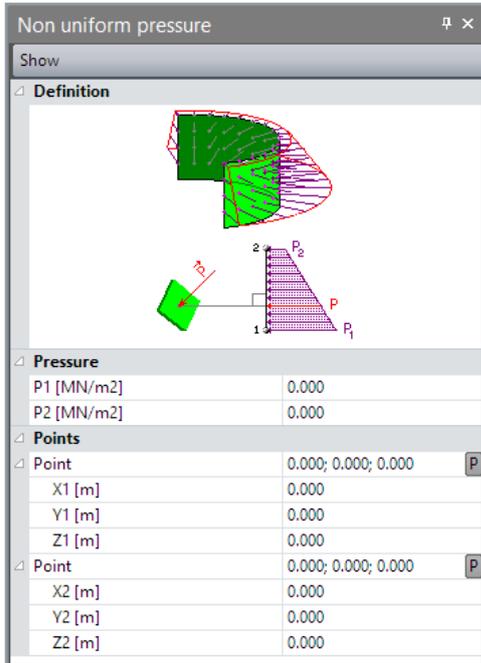
Visualizing all facets featuring the same imposed pressure

1. Activate the "Uniform pressure distribution" tool.
2. Enter the given imposed pressure in the dialog box.
3. Press the "Show" button. All facets featuring this imposed pressure are then selected.



Mechanics: Non-uniform distributed pressure

This tool serves to assign non-uniform pressures on the model "surfaces". As indicated in the figure below, the pressures considered herein exhibit a linear variation between two points defined by the user.



Pressures are represented on the screen by arrows whose length is proportional to the pressure intensity. Pressure is directly applied by clicking on the targeted surface. As a convention, it is considered that a positive pressure is oriented towards the clicked surface. The arrow shown on the screen indicates the direction of the pressure being analyzed.

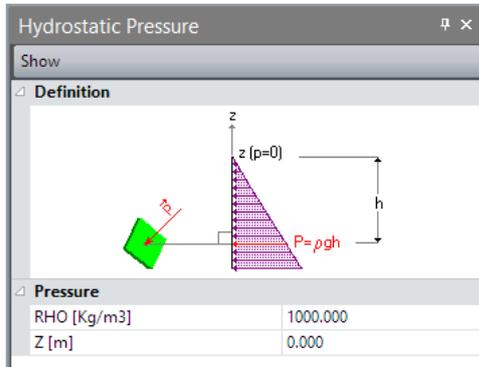
Assigning a non-uniform distributed pressure

1. Activate the "Non-uniform distributed pressure" tool.
2. Set the imposed pressure values P1 and P2 at the level of points 1 and 2, respectively.
3. Specify the coordinates of points 1 and 2 used to assign the pressure variation.
4. Click on the surface where the variable pressure is to be applied.



Mechanics: Hydrostatic pressure

This tool serves to assign hydrostatic-type pressures on either the facets of selected surface regions or the external facets of volume bodies.



The assignment of hydrostatic pressures presumes that the two following data values are known:

- Z (p=0): Height z at which the pressure is zero;
- Rho: Volumic mass of the pressure-exerting fluid; the hydrostatic pressure on a point of height z is computed as follows:

$$p = 0 \text{ if } z \geq Z (p=0),$$

$$p = \text{Rho} \cdot g \cdot (Z (p=0) - z) \text{ if } z < Z (p=0)$$

Pressures are represented on the screen by a series of arrows positioned at the center of each facet, with lengths proportional to the hydrostatic pressure intensity.

Assigning a hydrostatic pressure

1. Activate the "Hydrostatic pressure" tool.
2. Enter the values of the height Z where p=0 and the fluid density, Rho.
3. Click on the surfaces where will be assigned this hydrostatic pressure.

Identifying the hydrostatic pressure imposed on a facet

1. Activate the "Hydrostatic pressure" tool.
2. Right click on the target facet. The values of both Z(p=0) and Rho serving to compute the hydrostatic pressure imposed on the clicked facet are then displayed in the dialog box.



If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all facets submitted to a given hydrostatic pressure

1. Activate the "Hydrostatic pressure" tool.
2. Enter the values of Z(p=0) and Rho defining the given hydrostatic pressure in the dialog box.
3. Press the "Show" button. All facets submitted to this hydrostatic pressure are then selected.

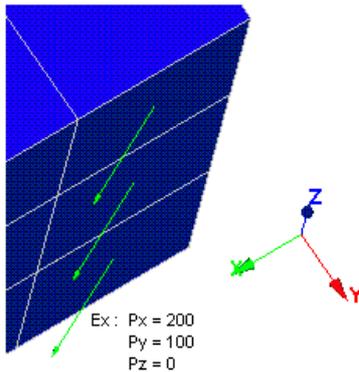


Mechanics: Surface forces

This tool serves to assign surface force on either the facets of selected surface regions or the external facets of volume bodies.

A surface force is defined by means of its three components P_x , P_y , P_z within the global coordinate system.

Surface forces are represented on the screen by a series of arrows positioned at the center of each facet, with lengths proportional to the intensity of the surface force and direction defined by the vector components.



Assigning a surface force

1. Select those facets exhibiting the same surface force.
2. Activate the "Surface force" tool.
3. Enter the components of the surface force vector.
4. Press the "Apply" button to assign this surface force to the set of selected facets.

Identifying the surface force imposed on a facet

1. Activate the "Surface force" tool.
2. Right click on the target facet. The components of the surface force vector being applied on the clicked facet are then displayed in the dialog box.



If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all facets featuring the same surface force

1. Activate the "Surface force" tool.
2. Enter the given surface force vector components in the dialog box.
3. Press the " Show" button. All facets featuring this imposed surface force are then selected.

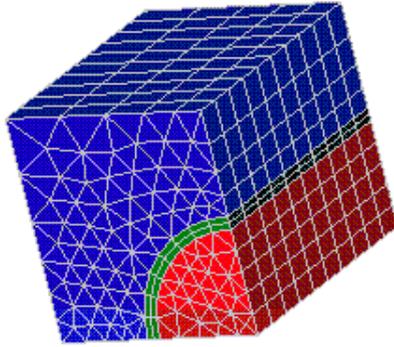


Mechanics: Excavation forces

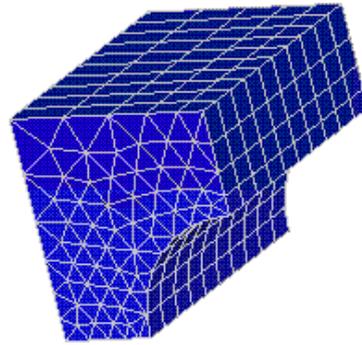
This tool enables assigning "confinement stress relaxation"-type surface forces on the external facets of volume regions.

Excavation forces are used to model the influence of an excavation on the adjacent soil, mainly the confinement relaxation.

i The tool is only activated for load cases associated with a "Phasing"-type model.



Phase preceding excavation (stress origin)



Excavation phase

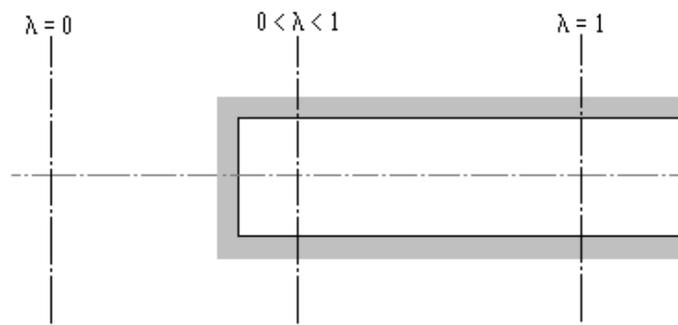
In order to model an excavation, one conventional method consists of applying surface forces on the set of facets that become "external" during the excavation-simulation stage (i.e. the Excavation Stage).

These applicable surface forces can be expressed as follows:

$$\begin{Bmatrix} T_x \\ T_y \\ T_z \end{Bmatrix} = -\lambda \begin{Bmatrix} \sigma_{xx}^0 \cdot n_x + \sigma_{xy}^0 \cdot n_y + \sigma_{xz}^0 \cdot n_z \\ \sigma_{yx}^0 \cdot n_x + \sigma_{yy}^0 \cdot n_y + \sigma_{yz}^0 \cdot n_z \\ \sigma_{zx}^0 \cdot n_x + \sigma_{zy}^0 \cdot n_y + \sigma_{zz}^0 \cdot n_z \end{Bmatrix}$$

with:

- T_x, T_y, T_z : Surface force vector components (confinement stress relaxation) applicable at a point of a facet that has become external during the excavation-simulation phase;
- σ_{ij}^0 : "Initial" stress tensor components expressed at this same point and existing in the soil prior to excavation.
- n_x, n_y, n_z : Components of the vector normal to the given facet at this point;
- λ : Coefficient with a value of between 0 and 1, inclusive. This coefficient λ serves to apply a portion of the confinement relaxation forces in order to simulate eventual tunnel face proximity (2D problems).

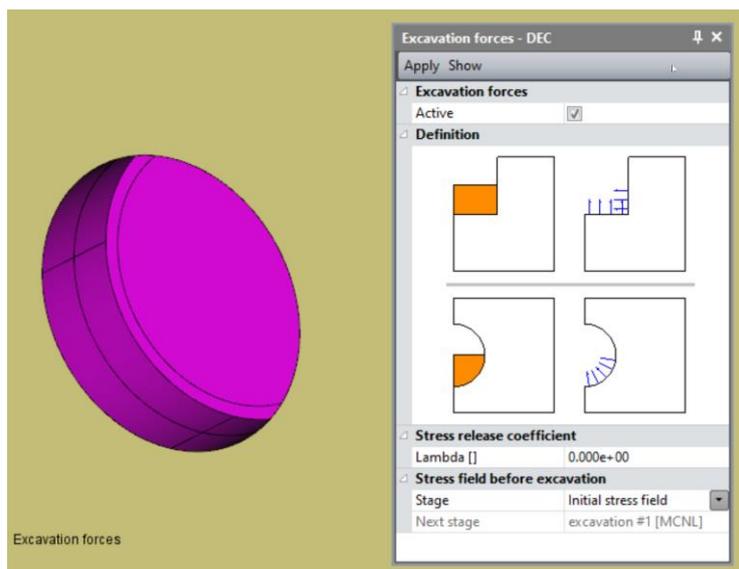


Simulation of tunnel face proximity using the coefficient λ

- i** In the event the given coefficient is not set equal to 1 during the excavation phase, it would then become necessary to include the "complement" of these surface forces within a subsequent phase (tunnel face placed at infinity).
- i** Since version 2020.0, a new definition of the excavation forces is introduced as default tool (named DEC). It replaces the former excavation tool (LAM), still proposed for the compatibility with older versions.
- i** It should be noted herein that this tool remains inactive in the special case where the model corresponds with the first stage and where initial stresses during this phase are zero.

Assigning excavation forces (DEC)

1. Activate the "Excavation forces" tool.
2. Tick "Active"
3. Specify the value of the stress release coefficient ("Lambda", between 0 and 1).
4. Enter the type of "Stress origin", the stress field before excavation.
5. Press the "Apply" button to assign these confinement relaxation forces to the set of selected faces. "Excavation forces" is written in the left-bottom corner of the working space.



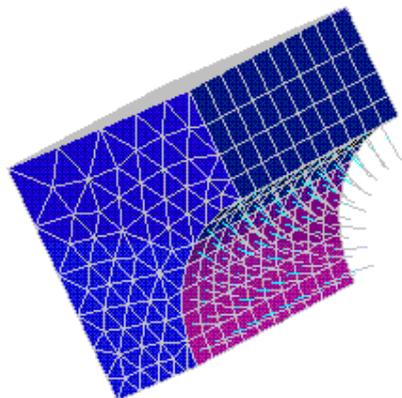
Display of excavation forces (DEC)

Assigning excavation forces (LAM)

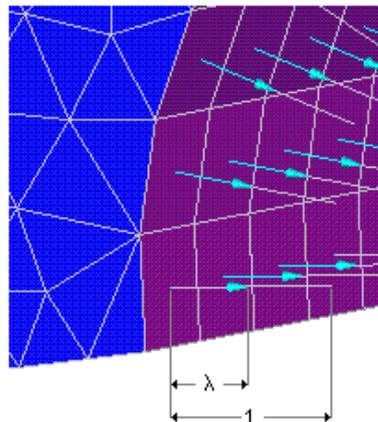
1. Activate the "Excavation forces" tool.
2. Specify the value of the stress release coefficient ("Lambda", between 0 and 1).
3. Enter the type of "Stress origin", the stress field before excavation.
4. Select those faces on which the confinement relaxation forces are acting. Use "Automatic selection" to select the set of faces to become "external" during the **subsequent** "Stress origin" phase.
5. Press the "Apply" button to assign these confinement relaxation forces to the set of selected faces. "Excavation forces" is written in the left-bottom corner of the working space.

- i** In the special case where the stress origin is a stage, the dialog box contains a button labeled "Automatic selection", which when activated selects the set of facets to become "external" during the **subsequent** "Stress origin" phase.

Excavation forces are displayed on the screen by arrows having a length function of the value of lambda coefficient.



Display of excavation forces (LAM)



Detail of the display



Mechanics: Gravitational forces

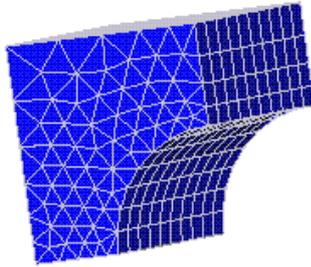
This tool allows assigning gravitational forces on the set of selected bodies.

Assigning gravitational forces

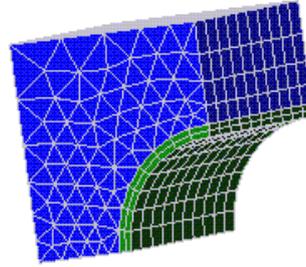
1. Activate the "Gravitational forces" tool.
2. Enter the value of the global coordinate system components for the given acceleration g .
3. Select those bodies on which the gravitational forces are acting.
4. Press the "Apply" button to assign these gravitational forces to the selected set. "Gravity forces" is written in the left-bottom corner of the working space.



Activating the "Automatic selection" button serves to select the set of inactive bodies from the previous phase to become active during the desired phase. This button can only be activated if the given model is of the "Phase" type and if it corresponds to a phase other than Phase 1.



Preceding stage



Current stage (the body represented in green has become active)

Identifying the gravitational force assigned to a body

1. Activate the "Gravitational forces" tool.
2. Right click on the target body. The characteristics of the acceleration imposed on the clicked group are then displayed in the dialog box.



If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all bodies submitted to a given gravitational force

1. Activate the "Gravitational forces" tool.
2. Enter the components of the given acceleration into the dialog box.
3. Press the "Show" button. All bodies submitted to this gravitational force are then selected.



Mechanics: Constant stresses

This tool enables assigning uniform initial stresses on volume bodies.

Assigning constant initial stresses

1. Select the volume bodies for which specific constant initial stresses are to be defined.
2. Activate the "Constant initial stresses" tool.
3. Enter the value of the six components in the global coordinate system of the given initial stress tensor.
4. Press the "Apply" button to assign these initial stresses to the set of selected volume element groups.

Identifying the stresses assigned to a volume body

1. Activate the "Constant stresses" tool.
2. Right click on the target body. The components of the stress tensor imposed on the clicked body are then displayed in the dialog box.



If no load is associated to the target body, then a warning message (#1515) is displayed.

Visualizing all bodies submitted to constant stresses

1. Activate the "Constant stresses" tool.
2. Enter the components of the given stress tensor in the dialog box.
3. Press the "Show" button. All volume bodies submitted to this initial stress are then selected.



Mechanics: stresses input from a file

This tool allows assigning initial stresses input from a file.

Assigning stresses input from a file

1. Activate the "Initial stresses input from a file" tool.
2. Enter the name of the file containing the initial stresses. In the case where a loading of this file-input type has already been assigned, the particular file name is displayed in the dialog box. Otherwise, the cell labeled "Name" is left empty.
3. Press the "Apply" button to assign these initial stresses to the set of selected volume bodies.

**Mechanics: Initial thermal stresses**

This tool serves to assign initial stresses of a thermal origin on the set of selected volume bodies. The temperatures causing these initial stresses are either given as constant per body or input from a file.

Assigning constant initial thermal stresses per group

1. Activate the "Initial thermal stresses" tool.
2. Enter the coefficient of thermal dilatation for the considered bodies.
3. Specify the reference temperature T_{ref} .
4. Specify the temperature T of the selected bodies. The initial stresses will then be computed from the temperature rise $T - T_{ref}$.
5. Press the "Apply" button to assign these initial stresses to the set of selected bodies.

Assigning initial thermal stresses with temperatures input from a file

1. Activate the "Initial thermal stresses" tool.
2. Enter the coefficient of thermal dilatation for the considered bodies.
3. Specify the reference temperature T_{ref} .
4. Enter the name of the file containing the given temperatures. This file name may also be obtained in the conventional manner by using the "Open" button.
5. Press the "Apply" button to assign these initial stresses to the set of selected bodies.



Mechanics: long-term effects

i The formulation of this tool is only compatible with material having a constitutive law with linear isotropic elasticity. Thus, it is not compatible with Hardening Soil Model for example.

For certain soil types, modifying the stress state in the soil mass, due for example to an excavation, may cause deferred deformations that evolve very slowly over a long period of time; such deformations may have several origins: movements of pore water, variations of volume in the presence of water (swelling clayey materials), viscoelastic or viscoplastic soil behavior. The focus herein lies in this third effect.

In the case of plastic materials, such a computation theoretically requires monitoring step-by-step the evolution taking place in this phenomenon and hence solving a "viscoplasticity" type of problem. Since the computations involved are rather sizable, a simplified method that proves accurate in the case of materials displaying linear behavior is typically adopted, provided a certain number of additional hypotheses have been included.

It can be shown that this method encompasses an "initial stress"-type of loading, whereby the particular stresses can be computed as follows:

$$\sigma_d = \frac{\Delta\mu}{\mu_1}(\sigma_1 - \sigma_0) + \frac{(\mu_1 \cdot \Delta\lambda - \Delta\mu \cdot \lambda_1)}{\mu_1 \cdot (3\lambda_1 + 2\mu_1)} \text{tr}(\sigma_1 - \sigma_0)I$$

with:

- σ_1 : Stress state at the end of the construction phases,
- σ_0 : Initial stress state,
- (λ_1, μ_1) : Instantaneous Lamé coefficients,
- (λ_2, μ_2) : Long-term Lamé coefficients,
- I : Unit matrix.

The "Long-term effects" tool enables applying an initial stress loading, defined as indicated above, on the set of selected volume bodies.

i The tool is only activated for load cases associated with a "Staged construction"-type model and remains inactive both for Stage #1, in the case where this phase features initial stresses of the geostatic type, and for Stages #1 and #2 in the case where Stage #1 initial stresses are zero.

Assigning initial stresses of the "deferred-effect" type

1. Select the volume bodies for which initial stresses of the deferred-effect type are to be defined.
2. Activate the "Long-term effects" tool.
3. Specify the "initial" stress state. At this point, the user is given the choice between "Geostatic origin" (only in the case where Stage #1 initial stresses are of the geostatic type) and "Stage".
4. If the choice is "Stage", the user must then specify the stage serving to define initial stresses from the drop-down list containing the titles of each phase previous to the current stage.
5. Define the "current" stress state. The user is to indicate here the phase serving to define current stresses from the drop-down list containing the titles of each phase previous to the current phase.
6. Enter the "instantaneous" values of both Young's modulus and Poisson's ratio for the selected bodies.
7. Press the "Apply" button to assign to the set of selected volume element groups.

-  Note that the "long-term" values are defined under the properties associated with each volume body (see Model / Properties).

Identifying initial stresses of the deferred-effect type assigned to a volume body

1. Activate the "Deferred effects" tool.
2. Right click on the target body. The characteristics serving to compute the initial stresses imposed on the clicked body are then displayed in the dialog box.

-  If no load is associated to the target volume, then a warning message (#1515) is displayed.

Visualizing all bodies submitted to initial stresses of the long term-effect type

1. Activate the "Long-term effects" tool.
2. Enter the characteristics serving to compute these initial stresses into the dialog box.
3. Press the " Show" button. All volume bodies submitted to this initial stress are then selected.



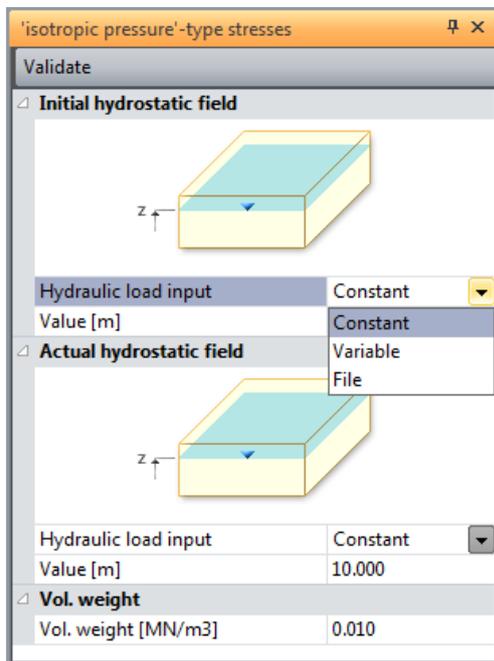
Change in water table position

This tool serves to assign a loading that enables:

- initializing the effective stresses inside a body subjected to a known hydraulic load field (e.g. one determined by means of a preliminary "Hydrogeology" type computation);
- or computing the strains stemming from a variation in water table, or more precisely from the difference between a final hydraulic load field and an initial state.

Three types of hydraulic loads defining the water table may be assigned, namely:

- constant,
- variable,
- read from a file, provided a "Hydraulics" type computation has been performed first.



Assigning an initial water table level

- constant

1. Select the "Constant" type hydraulic load data.
2. Set the initial load value.

- variable

1. Select the "Variable" type hydraulic load data.
2. Indicate the flow variation plane (XZ or YZ).
3. Enter the number of points (X_i, Z_i) or (Y_i, Z_i) used to define the water table variation.
4. Assign each (X_i, Z_i) or (Y_i, Z_i) couple.

- read from the file

1. Select the "File" type hydraulic load data.
2. Browse in order to specify the storage file for the hydraulic load field resulting from a "Hydrogeology" type computation. The format (name and extension) of this file is not restricted.

Assigning a final water table level

- constant

1. Select the "Constant" type hydraulic load data.
2. Set the initial load value.

- variable

1. Select the "Variable" type hydraulic load data.
2. Indicate the flow variation plane (XZ or YZ).
3. Enter the number of points (Xi, Zi) or (Yi, Zi) used to define the water table variation.
4. Assign each (Xi, Zi) or (Yi, Zi) couple.

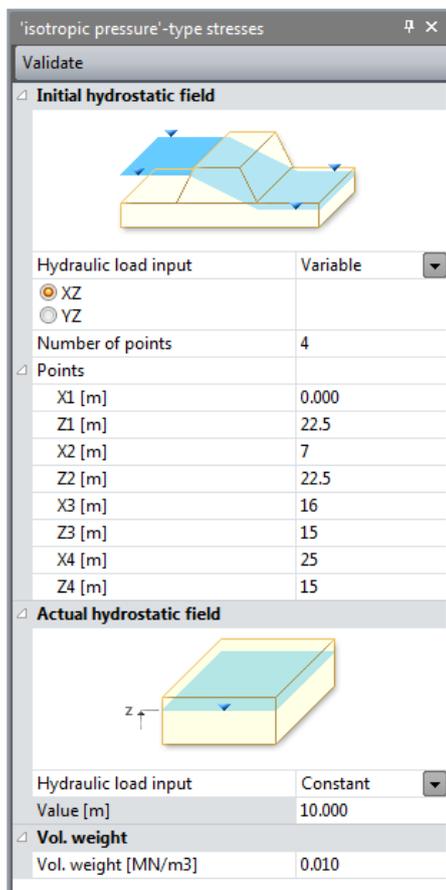
- read from the file

1. Select the "File" type hydraulic load data.
2. Browse in order to specify the storage file for the hydraulic load field resulting from a "Hydrogeology" type computation. The format (name and extension) of this file is not restricted.

Fluid unit weight data

1. Enter the unit weight value of the given fluid. As a default, water is assigned the value 10 kN/m³.

The operation of entering loadings into the model ends by pressing the "Validate" button.



Input of the water table variation properties



Result applied on a model



Definition of the bodies assigned due to the change in water table

Since the loading associated with the water table variation has been previously defined, the bodies assigned by this loading and their associated dry and saturated unit weights must now be specified. Impermeable bodies (e.g. inner walls) do not lie within the scope of this tool.

Assigning a unit weight

2. Select the bodies displaying the same properties.
3. Run the "Unit weight definition" tool.
4. Enter the unit weight values of both the dry soil and the saturated soil.
5. Press "Apply" to assign these data to the entire set of selected bodies.



Uniformly-distributed flow

This tool enables assigning uniformly-distributed flows on the selected external facets of the volume bodies.

Assigning a uniformly-distributed flow

1. Select those facets having the same uniformly-distributed flow.
2. Activate the "Uniform flow distribution" tool.
3. Enter the value of the imposed flow.
4. Press the "Apply" button to assign this flow to the set of selected facets.

Identifying the uniformly-distributed flow imposed on a facet

1. Activate the "Uniform flow distribution" tool.
2. Right click on the target facet. The flow imposed on the clicked facet is then displayed in the dialog box.



If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all facets featuring the same imposed uniformly-distributed flow

1. Activate the "Uniform flow distribution" tool.
2. Enter the given imposed flow in the dialog box.
3. Press the " Show" button. All facets featuring this imposed flow are then selected.



Point flow rates

This tool serves to assign point flow rates on the set of selected points.

Assigning a point flow rate

1. Select the points having the same point flow rate.
2. Activate the "Point flow rate" tool.
3. Enter the value of the imposed point flow rate.
4. Press the "Apply" button to assign this flow rate to the set of selected points.

Identifying the point flow rate imposed on a node

1. Activate the "Point flow rate" tool.
2. Right click on the target node. The point flow rate assigned at the clicked node is then displayed in the dialog box.

 If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all nodes featuring the same imposed point flow rate

1. Activate the "Point flow rate" tool.
2. Enter the given imposed flow rate in the dialog box.
3. Press the " Show" button. All nodes featuring this imposed flow rate are then selected.



Volumic flow rates

This tool enables assigning volumic flow rates on the set of selected volume bodies.

Assigning a volumic flow rate

1. Select the bodies having the same volumic flow rate.
2. Activate the "Volumic flow rate" tool.
3. Enter the value of the imposed volumic flow rate.
4. Press the "Apply" button to assign this flow rate to the set of selected bodies.

Identifying the volumic flow rate imposed on a body

1. Activate the "Volumic flow rate" tool.
2. Right click on the target body. The volumic flow rate assigned to the clicked body is then displayed in the dialog box.

 If no load is associated to the target surface, then a warning message (#1515) is displayed.

Visualizing all bodies featuring the same imposed volumic flow rate

1. Activate the "Volumic flow rate" tool.
2. Enter the given imposed flow rate in the dialog box.
3. Press the " Show" button. All bodies featuring this imposed flow rate are then selected.

1 3. Definition of "computation module" data

Introduction

In addition to the intrinsic characteristics of the model being developed, solving a specific problem requires inputting a number of data elements associated with the selected "Computation module" (e.g. solution method, convergence criteria, temporal discretizations).

This chapter describes the data required for each of the computation modules implemented in the present version of the software. Each computation module is characterized by a four-letter keyword (see list provided in the section entitled "Model definition").

As a general rule, the computation module data used for a given model are easily defined by activating the "Computation module settings" item available when right-clicking on the particular model in the "Models" tree diagram.

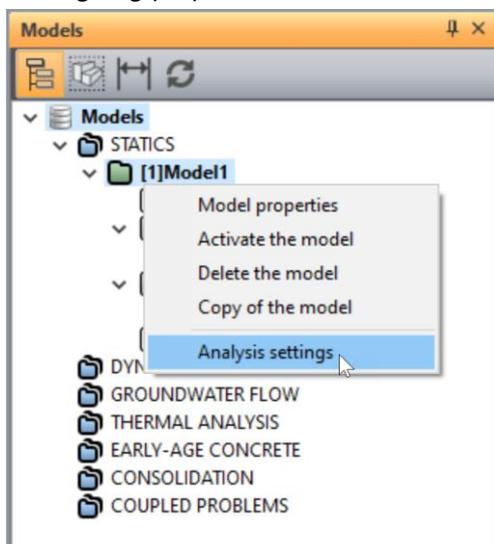
The computation modules may use one of the 3 following algorithms:

- "Pardiso" is the most powerful one, but requires a computer with large RAM size;
- "Multifrontal" method is less powerful than Pardiso, but compatible with almost all computers;
- "Skyline" method is the less powerful one, but it is the most stable.

Pardiso is recommended.

Accessing the computation settings

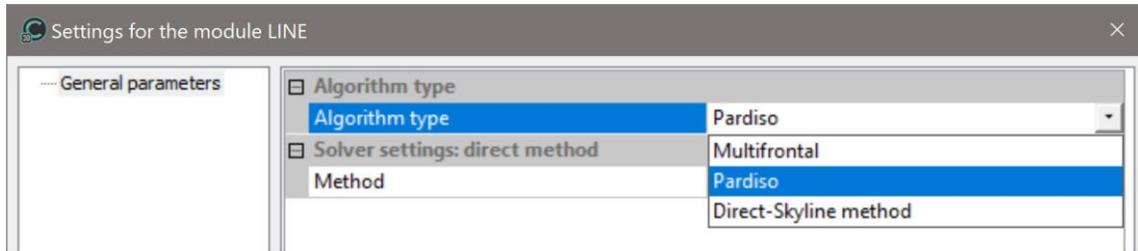
The computation settings definition menu is accessed by right-clicking on the name of the model undergoing preparation.



The LINE module

The LINE module is primarily used for solving linear problems.

The data required to run this computation module are defined via a dialog box containing the following tab.



Tab: General parameters

The computation of a linear problem may use one of the 3 following algorithms:

- "Pardiso" Method,
- "Multifrontal" method,
- "Skyline" method.

Parameters for "Multifrontal" algorithm

1. Choose the "Multifrontal" algorithm type.
2. Tick "Analysis with secondary storage on file" if required to store coefficients of factor matrix on files (on hard disk).



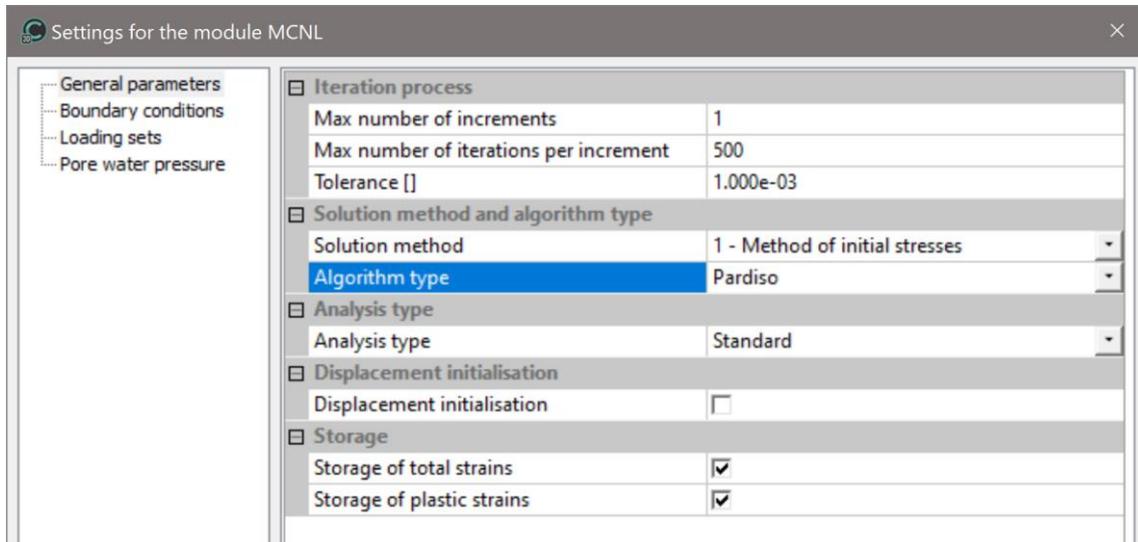
The multifrontal solver does not require any storage file, in contrary of the classic « Skyline » method. It results of the use of multifrontal algorithm that a larger random memory size is required. If this memory size is too large for computation on computers with weak memory capacity, it is advised to opt for the writing of the coefficients of factor matrix on hard disk.

Parameters for "Skyline" algorithm

1. Choose the "Direct Skyline method" algorithm type.
2. Choose between one of the three cases considered herein:
3. New computation without storage of the factorized matrix.
4. New computation with storage of the factorized matrix. This case may be chosen should response of the same structure when exposed to the effect of new load cases constitute the objective. Keep in mind that this option, in spite of all other considerations, is only of real interest in the case of "large" models possessing several thousand degrees of freedom. If this option were chosen, the user would have to identify the names of files for storing both the upper part of the stiffness matrix and the diagonal.
5. Computations including use of a factorized matrix. It is considered that matrix factorization has already been performed. When choosing this option, the user must list the names of the files created during a previous step and then used for storing both the upper part of the stiffness matrix and the diagonal.

The MCNL module

The MCNL module allows solving non-linear problems featuring mechanical behavior (elastoplasticity with strain hardening, non-linear elasticity) for the various materials (soils, concretes, rocks, pavement layer structures).



The data required to run this computation module are defined via a dialog box containing the following tabs.

Tab 1: General parameters

This tab enables defining parameters of the iterative method used for the computation.

Inputting the iterative process parameters

1. Define the iterative process:
2. Input the corresponding number of increments.
3. Input the maximum number of iterations per increment.
4. Set the relative tolerance on the convergence.
- i** A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.
5. Select the appropriate solution method from among the list proposed below:
 - Initial stresses method (default),
 - Tangential stiffness method,
 - Initial stresses method coupled with the modified D-F-P method,
6. Select the analysis algorithm:
 - "Pardiso" method (recommended),
 - "Multifrontal" method,
 - "Skyline" method.
7. Select the type of analysis between the 3 proposed.
 - Standard iterative analysis (default),
 - Calculation of safety factors on strengths properties, so called « c-phi reduction ». This option computes the highest R coefficient that divides the strengths parameters (c and $\tan \varphi$) of soil

bodies while achieving convergence of the elastoplastic calculation, i.e. stability (for the given number of iterations and tolerance).

- Data input:
 - Limits (Min values and Max value of 1/R) of the interval of research;
 - Accuracy on the R coefficient
 - Detection of non-convergence: when activated, it helps to speed up the analysis at the cost of lower stability of the process.

i Optimisation is led on all volume bodies of the model with the limitation that only following constitutive laws may be used: Mohr Coulomb, Drucker Prager without hardening, von Mises without hardening.

i Iterative process is automatically adjusted; therefore parameters of the iteration process are not taken into account (except for the tolerance).

- Calculation of safety factors on loads. This option computes the highest R coefficient that multiplies on of the loading set while achieving convergence of the elastoplastic calculation, i.e. stability (for the given number of iterations and tolerance). This analysis is an approach of the limit load.

- Data input:
 - Selection of the loading set.
 - Limits (Min values and Max value of R) of the interval of research;
 - Accuracy on the R coefficient
 - Detection of non-convergence: when activated, it helps to speed up the analysis at the cost of lower stability of the process.

i Iterative process is automatically adjusted; therefore parameters of the iteration process are not taken into account (except for the tolerance).

8. Storage of results: tick « total strains » and/or « plastic strains » will store associated fields of results for a graphical analysis in RESULTS step.

Tab 2: Monitoring of boundary conditions

This tab serves to define the evolution of any eventual nonzero displacement conditions imposed during the incremental process. This evolution is characterized by inputting a value $f(i)$ for each of the given increments.

The displacements imposed in a set of boundary conditions will be ascribed an increment i , such that:

$$\{U(i)\} = \{\bar{U}\} f(i)$$

where:

- $\{\bar{U}\}$: Vector of the displacements imposed in the given set of boundary conditions,
- $f(i)$: Value of the function for increment i .

i If no nonzero imposed displacements are contained within the given model, the data associated with this tab are not to be used.

i If the imposed displacements do not vary during the incremental process, it would simply be necessary to retain the values proposed by default, i.e.: $f(i) = 1 \forall i$

i In the special case where several evolutions of displacement-related boundary conditions must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter

entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different function f for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the set of conditions for which the function is to be specified from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(i)$ for each increment.
3. Repeat steps 2 and 3 for all sets of boundary conditions defined in the model.

 The Nincr number of values characterizing function f may also be input from a file by activating the "Initialization from file" button.

Tab 3: Monitoring of the loadings

This tab enables defining the evolution in the loadings imposed during the incremental process. This evolution is characterized by inputting a value $f(i)$ for each of the considered increments.

The loads imposed within a "load case" at increment i will be such that:

$$\{F(i)\} = \{\bar{F}\} \cdot f(i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loads imposed for the given load case,
- $f(i)$: Value of the function at increment i .

 In the special case where several evolutions of load conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different function f for each of these cases.

Entering the load monitoring coefficients

1. Select the load case for which function f is to be specified from among the load cases defined in the model.
2. Enter the values of the function $f(i)$ for each increment.
3. Repeat steps 2 and 3 for all load cases defined in the model.

 The Nincr number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

Tab 4: Pore water pressure

In the case of undrained analysis (specified in PROPERTIES of the material), this option allows to store the pressure field developed under loading on the undrained material.

Defining storage of water pressure for subsequent calculation

4. Tick the cell labeled "Storage for subsequent calculation".
5. Enter the name of the storage file.

Tab 5: Storage for restart

 In the special case where the current model is of the "Staged construction" type, this tab is deactivated, meaning that all storage and restart operations are conducted automatically.

This tab allows storing onto a file the set of elements (results obtained from the last increment) necessary for proceeding with a subsequent "restart".

Defining storage for subsequent calculation

1. Tick the cell labeled "Storage for subsequent calculation".
2. Enter the name of the storage file.

The TCNL module

The TCNL module serves to solve contact problems between two solids exhibiting non-linear mechanical behavior (elastoplasticity with strain hardening, non-linear elasticity).

The data required to run both of these computation modules are defined via a dialog box containing the following tabs.

Tab 1: General parameters

This tab enables defining parameters of the iterative method used for the computation along with the contact criteria to be verified.

Inputting the iterative process parameters

1. Define the iterative process:
2. Input the corresponding number of increments.
3. Input the maximum number of iterations per increment.
4. Set the relative tolerance on the convergence.



A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.

5. Define the criterion of the contact algorithm:

- Tick the cell labeled "Non-interpenetration" if non-interpenetration criteria are to be verified.
- Tick the cell labeled "Debonding" if debonding criteria are to be verified.
- Tick the cell labeled "Friction" if friction criteria are to be verified.

6. Select the analysis algorithm:

- "Multifrontal" method,
- "Skyline" method.

7. Storage of results: tick « plastic strains » will store associated fields of results for a graphical analysis in RESULTS step.

Tab 2: Monitoring of the loads

This tab enables defining the evolution in the loads imposed during the incremental process. This evolution is characterized by inputting a value $f(i)$ for each of the considered increments.

The loads imposed within a "load case" at increment i will be such that:

$$\{F(i)\} = \{\bar{F}\} \cdot f(i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loads imposed for the given load case,
- $f(i)$: Value of the function at increment i .



In the special case where several evolutions of load conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different function f for each of these cases.

Entering the load monitoring coefficients

1. Select the load case for which function f is to be specified from among the load cases defined in the model.
2. Enter the values of the function $f(i)$ for each increment.

3. The Nincr number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.
4. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 3: Storage for restart



In the special case where the current model is of the "phasing" type (only possible for the TCNL module), this tab is deactivated, meaning that all storage and restart operations are conducted automatically.

This tab allows storing onto a file the set of elements (results obtained from the last increment) necessary for proceeding with a subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

The DYNI module

The DYNI module computes, by means of direct integration, the response of a structure submitted to a dynamic load; it enables proceeding with a step-by-step resolution of the dynamic equilibrium equation:

$$[M]\{\ddot{X}(t)\} + [C]\{\dot{X}(t)\} + [K]\{X(t)\} = \{F(t)\}$$

where:

- $X(t)$: Vector of nodal displacements for the given structure,
- $\dot{X}(t), \ddot{X}(t)$: Speed and acceleration vectors,
- $F(t)$: Vector of imposed loads as a function of time,
- $[M], [C], [K]$: Mass, damping and stiffness matrices of the structure, respectively.

This module only enables solving linear problems for which matrices K, C and M are constant.

The integration over time has been based on Newmark's algorithm, which is implicit and unconditionally stable.

It is possible to define a Rayleigh-type damping matrix that assumes the following form:

$$[C] = a.[K] + b.[M]$$

where a and b are two user-defined constants.

If the percentages of critical damping ξ_1, ξ_2 associated with 2 eigenmodes of the structure (circular frequencies ω_1, ω_2) were known, a and b may be estimated from the following expressions:

$$a = \frac{2}{\omega_2^2 - \omega_1^2} (\xi_2 \omega_2 - \xi_1 \omega_1)$$

$$b = \frac{2\omega_1 \omega_2}{\omega_2^2 - \omega_1^2} (\xi_1 \omega_2 - \xi_2 \omega_1)$$

The data required to run this computation module are defined via a dialog box containing the following tabs.

Tab 1: General parameters

This tab serves to define the parameters characterizing both the time discretization and eventual Rayleigh damping.

Inputting the general parameters

1. Define the iterative process:
2. Input the number of pertinent time values $N_{pas} + 1$.
3. Input the initial time t_0 value.
4. Input the corresponding time step value.
5. Define the damping
6. Tick if damping is taken into account.
 - Input the value of the stiffness matrix coefficient.
 - Input the value of the mass matrix coefficient.
7. Select the analysis algorithm:
 - "Pardiso" Method,
 - "Multifrontal" method,
 - "Skyline" method.

- i** Note that these damping parameters are only taken into account for elements not affected by damping parameters set in « PROPERTIES ».

Tab 2: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the primary unknowns. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a "set of boundary conditions" at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\} f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

- i** If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

- i** In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").

- i** The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

3. For each time value, input the corresponding coefficient of "relaxation". This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

Tab 3: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

- i** If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

i If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.

i The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 4: Storage for restart

This tab allows storing onto a file the set of elements (results obtained from the last time step) required for a subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 5: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.

i Selected time steps may be "deselected" using the right mouse button.

3. Choose the storage type from either "Primary results" or "Primary and secondary results".

Tab 6: Listing file contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Mass of the structure,
- Characteristics of the "monitoring" functions (time functions for displacement boundary conditions and imposed loads),
- Displacements,

- Speeds,
- Accelerations.

Defining listing file contents

1. Tick those cells corresponding to the specific entities sought on the listing file.

The FLAM module

The FLAM module enables computing buckling loads and modes of elastic structures within the scope of so-called Euler linear buckling theory. The validity of this type of approach requires both for the pre-buckling geometry changes to be negligible and for the structure to remain entirely within its elastic domain at the time of buckling. The first condition is often verified in practice, except on particularly flexible structures. The second condition may be verified *a posteriori* by comparing the theoretical elastic stress field obtained for the buckling load with the plasticity criterion of the component materials for the given structure.

The FLAM module serves to incorporate loads of the following type:

$$\{F\} = \{F_1\} + \lambda\{F_2\}$$

where:

- $\{F_1\}$: Permanent load applied to the structure,
- $\{F_2\}$: Loading mode used when seeking to predict structural buckling,
- λ : Measurement of the intensity of load $\{F_2\}$.

This module's objective is, in specific terms, to compute the smallest value λ^* of λ capable of causing failure of the structure due to buckling.

The FLAM module has been derived from research on the first eigenmode of the following algebraic system:

$$[K - Kg(F_1)]\{\Delta U^*\} = \lambda^* [Kg(F_2)]\{\Delta U^*\}$$

with:

- K : Typical stiffness matrix of the structure in small deformation,
- $Kg(F_1)$: Initial geometrical stiffness matrix associated with load $\{F_1\}$,
- $Kg(F_2)$: Initial geometrical stiffness matrix associated with load $\{F_2\}$,
- $K_S(F^1)$: Matrix associated with the tracking pressure forces in the case where load F^1 corresponds to a fluid pressure ($K_S = 0$, otherwise),
- $K_S(F^2)$: Matrix associated with the tracking pressure forces in the case where load F^2 corresponds to a fluid pressure ($K_S = 0$, otherwise),
- λ^* : Eigenvalue providing the intensity of the buckling load,
- ΔU^* : Buckling eigenmode.

ΔU^* yields to, within an arbitrary multiplier (whether positive or negative), the "speed" field (increase in the displacement field) that the structure is capable of displaying at the time of buckling.

The search for the first eigenmode of the previous system is based on the inverse iteration algorithm. However, since special applications may necessitate searching for modes of an order greater than 1, this algorithm has been coupled with the subspace method implemented in the MODE module. The "Verification of the number of modes" option allows ensuring that the modes obtained upon completion of the iterations are in actuality the first ones, which might not always be so when the first eigenvalues of the above system are in the "vicinity" of one another.

If such is not the case, this module has been designed to include a shift value VD in the data set that enables, through use of an appropriate choice of option, forcing algorithm convergence onto the first modes corresponding to the eigenvalues closest to VD. In order for this step to be carried out, the effort will be made, by trial-and-error using a known order of magnitude, to assign VD a value close to the

target λ^{*1} value, while ensuring that VD remains strictly less than λ^{*1} . The opposite case ($\lambda^{*1} > VD$) would lead to a negative pivot in the solution process and hence to an interruption in program execution.

The VD shift may also be introduced so as to accelerate algorithm convergence. For a given level of precision, the number of iterations required proves to be inversely related to the proximity of the chosen VD to the buckling level, λ^{*1} .

Loads $\{F_1\}$ and $\{F_2\}$ are defined by the first two load cases associated with the given model, respectively. If just a single load case were to be defined, it would correspond to $\{F_2\}$; under such conditions, $\{F_1\}$ would be set to zero.

For each of the computed buckling modes, the FLAM module successively provides:

- the final load factor λ^* obtained at the end of the iteration,
- the buckling eigenmode ΔU^* , and
- the elastic stress field at the time of buckling submitted to the load $\{F\} = \{F_1\} + \lambda^* \{F_2\}$.



The FLAM module can be used with either the families of solid elements or "beam"-type mechanical structural elements (PB2 and PT2 elements).

Tab 1: General parameters

This tab serves to define the applicable computation mode and all eventual eigenmode computation parameters.

Inputting general parameters for identifying p eigenmodes

1. Enter the given VD eigenvalue. The eigenmodes targeted will be those whose associated eigenvalues lie closest to VD.
2. Indicate the number of buckling modes sought (Nvalp).
3. Define the dimension of the subspace used for searching buckling modes (NSE). If this value is input as zero, the software has been programmed to use $\text{Min}(Nvalp+8, 2*Nvalp)$ as a default.
4. Set the maximum number of iterations authorized.
5. Specify the level of relative precision desired for the eigenvalues.
6. Tick the cell labeled "Verification of the number of modes" if seeking to verify that the number of eigenvalues identified is equal to the number of eigenvalues existing over the given interval.
7. If $VD = 0$, the number of eigenvalues less than or equal to the maximum value of all NVALP identified eigenvalues is sought. If $VD \neq 0$, the program searches for the number of eigenvalues contained over the interval $[VD-A, VD+A]$, with: $A = \text{Max}_I(|VALP(I)-VD|)$ for $I = 1$ to NVALP.

Tab 2: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Eigenvalues,
- Eigenvectors.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The LINC module

The LINC module enables computing the steady-state response of a structure exhibiting linear behavior with damping, submitted to a harmonic load $F(t)$ defined by:

$$\{F(t)\} = \{F_1\} \cdot \text{Cos}(\omega t) + \{F_2\} \cdot \text{Sin}(\omega t)$$

where:

- $\{F_1\}, \{F_2\}$: Real vectors serving to define the imposed load,
- ω : Circular frequency of the imposed load.



From a practical standpoint, the vector F_1 is defined by the first load case associated with the model, whereas F_2 is defined by the second load case. In the event just a single load case was defined, vector F_2 would be set to zero.

The response of the structure is then obtained by solving the linear system with complex variables:

$$([K] + i[C]\omega - [M]\omega^2)\{V\} = \{F\}$$

where:

- V : Vector of complex nodal displacements for the given structure,
- $[M], [C], [K]$: Mass, damping and stiffness matrices of the structure, respectively.

The response in the time of the structure is thereby equal to:

$$\{V(t)\} = \text{Re}\{V e^{i(\omega t + \phi)}\}$$

where:

$$\{V(t)\} = \{V_1\} \cdot \text{Cos}(\omega t + \phi) - \{V_2\} \cdot \text{Sin}(\omega t + \phi)$$

with V_1 and V_2 being real vectors.



In a similar manner to that described for the imposed forces, it is possible to impose both the real and imaginary parts of displacements by means of two sets of boundary conditions.

It is also possible to define a Rayleigh-type damping matrix that assumes the following form:

$$[C] = a \cdot [K] + b \cdot [M]$$

where a and b are two user-defined constants.

If the percentages of critical damping ξ_1, ξ_2 associated with 2 eigenmodes of the structure (circular frequencies ω_1, ω_2) were known, a and b may be estimated from the following expressions:

$$a = \frac{2}{\omega_2^2 - \omega_1^2} (\xi_2 \omega_2 - \xi_1 \omega_1)$$

$$b = \frac{2\omega_1 \omega_2}{\omega_2^2 - \omega_1^2} (\xi_1 \omega_2 - \xi_2 \omega_1)$$

The data required to run the LINC computation module are defined via a dialog box with the following tabs.

Tab 1: General parameters

This tab serves to define the parameters characterizing both the time discretization and eventual Rayleigh damping.

Inputting the general parameters

1. Activate the "Damping" to take into account Rayleigh damping.
 - Input the Stiffness matrix coefficient
 - Input the Mass matrix coefficient
2. Input the angular frequency.

Tab 2: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Computation data,
- Primary results,
- Secondary results.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The LINH module

The LINH module allows computing the response of a structure exhibiting linear behavior without damping, submitted to a harmonic load $F(t)$ of the following type:

$$\{F(t)\} = \{F\} \cdot \text{Cos}(\omega t + \rho)$$

where:

- $\{F\}$: Vector serving to define the imposed load,
- ω : Pulsation of the imposed load,
- ρ : Dephasing of the imposed load.

The response of the structure is then obtained by solving the linear system set forth below:

$$([K] - [M]\omega^2)\{V\} = \{F\}$$

where:

- V : Vector of the nodal displacements for the given structure,
- $[M], [K]$: Mass and stiffness matrices of the structure, respectively.

The data required to run the LINH computation module are defined via a dialog box containing the following tab.

Tab: General parameters

This tab enables defining the frequency of the imposed load.

Inputting the general parameters

1. Input the angular frequency.

The MODE module

The MODE module has been designed to seek the p eigenvalues λ_{\pm} closest to a given value λ_0 , along with the p associated eigenvectors $\{\Phi_{\pm}\}$ satisfying the following relation:

$$[K]\{\Phi_i\} = \lambda_i[M]\{\Phi_i\} \quad (i = 1, p)$$

where:

- $[M], [K]$: Mass and stiffness matrices of the structure, respectively.

The subspace method is employed herein to identify these eigenmodes.

This module may also be used to merely identify the number of eigenvalues less than a given value λ_0 .

In this case, the number of negative pivots encountered during matrix factorization is counted as:

$$[K] - \lambda_0[M]$$

The data required to run the MODE computation module are defined via a dialog box containing following tabs.

Tab 1: General parameters

This tab serves to define the applicable computation mode and all eventual eigenmode computation parameters.

Inputting general parameters to identify the number of eigenvalues less than a given value

1. Tick the cell labeled "Identify the number of frequencies less than a given value".
2. Enter the given frequency VD.

Inputting general parameters for identifying p eigenmodes

1. Tick the cell labeled "Identify vibration eigenmodes".
2. Enter the given frequency VD. The eigenmodes identified will be those with frequencies closest to VD.
3. Indicate the number of eigenmodes being sought (Nvalp).
4. Define the dimension of the subspace used for searching the eigenmodes (NSE). If this value is input as zero, the software has been programmed to use $\text{Min}(Nvalp+8, 2*Nvalp)$ as a default.
5. Set the maximum number of iterations authorized.
6. Specify the level of relative precision desired for the eigenvalues.
7. Tick the cell labeled "Verification of the number of modes" if seeking to verify that the number of eigenvalues identified is equal to the number of eigenvalues existing over the given interval.
8. If $VD = 0$, the number of eigenvalues less than or equal to the maximum value of all NVALP identified eigenvalues is sought. If $VD \neq 0$, the program searches for the number of eigenvalues contained over the interval $[VD-A, VD+A]$, with: $A = \text{Max}_I(|\text{VALP}(I)-VD|)$ for $I = 1$ to NVALP.

Tab 2: Mode storage

This tab enables storing onto a file all of the vibration mode characteristics for subsequent use in conjunction with the SUMO module (Modal superposition).

Defining storage for restart

1. Tick the cell labeled "Storage of vibration modes".
2. Enter the name of the storage file.

Tab 3: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Mass of the structure,
- Eigenvalues,
- Eigenvectors.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The SUMO module

The SUMO module computes, by means of modal superposition, the response of a structure submitted to a dynamic load.

Two types of problems have been considered herein:

- estimation of the maximum response by means of spectral computation (an SRSS- or CQC-type superposition),
- computation of the full response.

The data required to run the SUMO computation module are defined via a dialog box containing following tabs.

Tab 1: General parameters

This tab serves to define the applicable computation mode and the parameters associated with the basis of the eigenmodes used.

Inputting the general parameters

1. Select the type of computation by ticking the corresponding cell.
2. Enter the name of the file containing the basis of the applicable Eigen modes (saved during a previous application of the MODE module).
3. Enter the number of modes in this basis.
4. Indicate, in the grid reserved for this purpose, the percentage of critical damping associated with each selected mode.
5. Choose from the proposed list the selected modes using the left mouse button.



The selected modes may be "deselected" using the right mouse button.

Tab 2: Time step

This tab enables defining the given time discretization.



This tab may only be activated if the "Time function computation" mode of computation has been chosen.

Defining the time steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Input the initial time t_0 value.
3. Input the corresponding time step value.

Tab 3: Spectral computation parameters

This tab serves to define the spectral computation parameters.



This tab may only be activated if the "Spectral computation" mode of computation has been chosen.

Defining the spectral computation parameters

1. Choose the appropriate type of spectrum (displacement or acceleration).
2. Specify the spectrum by inputting the corresponding (T (period), Spectrum (m or m/sec^2)) couples.

Tab 4: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

 This tab may only be activated if the "Time function computation" mode of computation has been chosen.

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

 If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

 If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

 In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.

 The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 5: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

 This tab may only be activated if the "Time function computation" mode of computation has been chosen.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.

 Selected time steps may be "deselected" using the right mouse button.

3. Choose the storage type from either "Primary results" or "Primary and secondary results".

Tab 6: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Computation data,
- Primary results,
- Secondary results.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The DTNL module

i Although the following explanations use the heat equation for purposes of illustration, both the DTLI and DTNL modules actually enable processing, in a general manner, the class of transient diffusion problems. The DTLI module serves to solve linear problems, while DTNL has been designed to solve problems in which conductivity characteristics and exchange coefficients may be non-linear (temperature-related functions).

Modules DTNL allow computing evolution in the temperature field of a structure submitted to a given volumic thermal loading along with a set of boundary conditions. Both modules have been built around providing a valid solution to the heat equation at all points on the structure:

$$c \frac{\partial \theta}{\partial t} - \text{div} \left(\underline{k} \overrightarrow{\text{grad}} \theta \right) = f(x, t)$$

with $f(x, t)$ being a volumic-based term; moreover, the three most common types of boundary conditions are to be incorporated:

- $\Theta = \Theta_{imp}(t)$: Imposed temperature
- $q = q_{imp}(t)$: Imposed flow
- $q = \lambda(\Theta_{imp}(t) - \Theta)$: Linear exchange

where:

- q : Inflow entering the structure
- λ : Exchange coefficient
- $\Theta_{imp}(t)$: Imposed external temperature
- Θ : Skin temperature of the structure

Following discretization using the finite element method, such a problem is rewritten in the form of the classical system of differential equations:

$$[C]\{\dot{\Theta}(t)\} + [K]\{\Theta(t)\} = \{F(t)\}$$

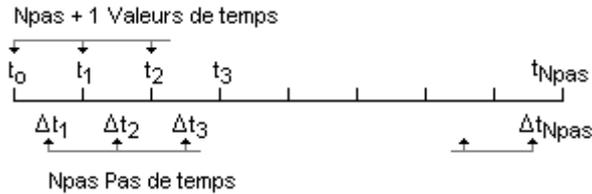
with:

- $\{\Theta(t)\}$: Temperature vector
- $[K]$: Conductivity matrix
- $[C]$: Accumulation matrix
- $\{F(t)\}$: Vector right-hand side

Integration of this system with respect to time is based on the Galerkin step-by-step algorithm.

Tab 1: Time step

This tab enables defining the given time discretization.



Three methods may be employed in order to define the given time values.

2. Constant time step:

According to this method, the user identifies the initial value t_0 of time along with a time step value Δt . The time values considered herein are:

$$t_1 = t_0 + \Delta T,$$

$$t_2 = t_1 + \Delta T, \dots$$

3. Time defined by absolute value:

According to this method, the user directly indicates the $N_{pas}+1$ time values to be taken into consideration.

4. "Group"-defined time steps:

According to this method, the user indicates the initial time value t_0 along with a set of value couples $n_i, \Delta t_i$. It is assumed herein that time step Δt_i remains constant for n_i number of time steps. The time values taken into consideration will be:

$$t_1 = t_0 + \Delta T_1$$

$$t_2 = t_1 + \Delta T_1$$

...

$$t_{n1} = t_{n1-1} + \Delta T_1$$

$$t_{n1+1} = t_{n1} + \Delta T_2$$

...

Defining time discretization by means of constant steps

1. Chose the "Transient" option.
2. Indicate the number of pertinent time values $N_{pas} + 1$.
3. Enter the initial time t_0 value.
4. Select the "Constant time step" option.
5. Input the corresponding time step value.

Defining discretization by means of inputting absolute time values

1. Chose the "Transient" option.
2. Indicate the number of pertinent time values $N_{pas} + 1$.
3. Enter the initial time t_0 value.
4. Select the "Time given by absolute value" option.
5. Input the N_{pas} values of time that complete the initial value in the table reserved for this purpose.

Defining discretization by means of "grouping" time steps

1. Chose the "Transient" option.
2. Indicate the number of pertinent time values $N_{pas} + 1$.
3. Enter the initial time t_0 value.
4. Select the "Time step given by group" option.
5. Press the "Add" button as many times as there are time step groups to be included.
6. Identify the couples n_i, t_i in the table reserved for this purpose.

i It should be noted that the sum of the n_i values must equal N_{pas} .

i For a steady-state computation, input the number of pertinent time values $N_{pas} + 1 = 1$, and then simply indicate the initial time t_0 value.

Tab 2: Solution method (solely for the DTNL module)

This tab enables choosing and defining parameters of the iterative method used for the computation.

Inputting parameters of the iterative process

1. Select the appropriate solution method (either the fixed-point method or Newton's method).
2. Enter the maximum number of iterations authorized per time step.
3. Set the relative tolerance on the nodal temperature values obtained.
4. Select the analysis algorithm:
5. "Multifrontal" method,
6. "Skyline" method.

i A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.

Tab 3: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the primary unknowns. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a set of boundary conditions at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\} f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

i If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").

i The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

3. For each time value, input the corresponding coefficient of relaxation. This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

Tab 4: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

i If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

i If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.
3. Repeat steps 2 and 3 for all load cases defined in the model.

i The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

Tab 5: Storage for restart

This tab allows storing onto a file the computation results from the last time step either for performing a subsequent restart of the diffusion computation or for a mechanical computation that incorporates a thermal or equivalent based loading.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".

2. Enter the name of the storage file.

Tab 6: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.
3. Choose the storage type from either "Primary results" or "Primary and secondary results".



Selected time steps may be "deselected" using the right mouse button.

Tab 7: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Characteristics of the "monitoring" functions (time functions for temperature boundary conditions and imposed loads),
- Primary results,
- Secondary results.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The NSAT module

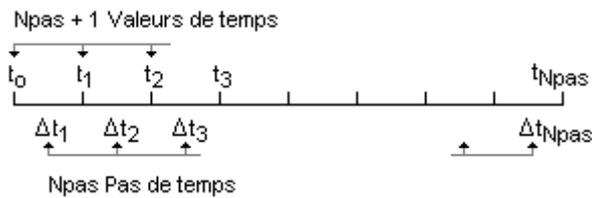
The NSAT module enables solving flow problems in unsaturated porous media. The hydraulic loads at each node constitute the primary result of this module, while the secondary results obtained are:

- Water content,
- Gradients,
- Flows.

The data required to run the NSAT computation module are defined via a dialog box containing the following tabs.

Tab 1: Time step

This tab enables defining the given time discretization.



Three methods may be employed in order to define the given time values.

1. Constant time step:

According to this method, the user identifies the initial value t_0 of time along with a time step value Δt . The time values considered herein are:

$$t_1 = t_0 + \Delta T,$$

$$t_2 = t_1 + \Delta T, \dots$$

2. Time defined by absolute value:

According to this method, the user directly indicates the $N_{pas}+1$ time values to be taken into consideration.

3. "Group"-defined time steps:

According to this method, the user indicates the initial time value t_0 along with a set of value couples $n_i, \Delta t_i$. It is assumed herein that time step Δt_i remains constant for n_i number of time steps. The time values taken into consideration will be:

$$t_1 = t_0 + \Delta T1$$

$$t_2 = t_1 + \Delta T1$$

...

$$t_{n1} = t_{n1-1} + \Delta T1$$

$$t_{n1+1} = t_{n1} + \Delta T2$$

...

Defining time discretization by means of constant steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Constant time step" option.
4. Input the corresponding time step value.

Defining discretization by means of inputting absolute time values

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time given by absolute value" option.
4. Input the N_{pas} values of time that complete the initial value in the table reserved for this purpose.

Defining discretization by means of "grouping" time steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time step given by group" option.
4. Press the "Add" button as many times as there are time step groups to be included.
5. Identify the couples $(n_i, \Delta t_i)$ in the table reserved for this purpose.



It should be noted that the sum of the n_i values must equal N_{pas} .

Tab 2: Solution method

This tab enables choosing and defining parameters of the iterative method used for the computation.

Inputting parameters of the iterative process

1. Select the appropriate solution method (either the fixed-point method or Newton's method).
2. Enter the maximum number of iterations authorized per time step.
3. Set the relative tolerance on the hydraulic load values.



A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.

4. Select the analysis algorithm:
 - "Multifrontal" method,
 - "Skyline" method.

Tab 3: Vertical

This tab allows defining the vertical.

Defining the vertical

1. Enter the components of the rising vertical unit vector within the coordinate system that was used to define the mesh.

Tab 4: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the "primary unknowns". This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a set of boundary conditions at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\} \cdot f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

i If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").

i The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. For each time value, input the corresponding coefficient of relaxation. This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

Tab 5: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

i If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

i If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.

 The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 6: Storage for restart

This tab allows storing onto a file the computation results obtained from the last time step for the purpose of a subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 7: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results" (gradients and flows to be included separately).

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.

 Selected time steps may be "deselected" using the right mouse button.

3. Choose the storage type from either "Primary results" or "Primary and secondary results".

Tab 8: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Characteristics of the "monitoring" functions (time functions for load boundary conditions),
- Primary unknown (load),
- Gradients,
- Speeds.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The TEXO module

The TEXO module is mainly a heat equation solution module, in which the volumic source term dQ/dt has been introduced. This module thereby enables simulating the temperature field developing within a recently-cast concrete specimen. This phenomenon stems from the heat release that accompanies the cement hydration reaction during concrete setting.

TEXO makes it possible, provided the availability of an adequate data set, to incorporate the effects of heat treatment, e.g. stoving, installation of insulation tarpaulins or heating resistors. The module also allows simulating the effects of casting concrete on a new portion of a structural element abutting a portion with previously-cast concrete. The set of computations to be performed relies upon the results of the so-called "QAB" (Quasi-Adiabatic) test, which must be included as part of the input data; this operation serves to characterize the reactivity of the cement used.

After application of the TEXO module, it then becomes possible to deduce the thermally-induced stresses in the specimen; this step requires conducting a computation within the mechanical domain (see MEXO module).

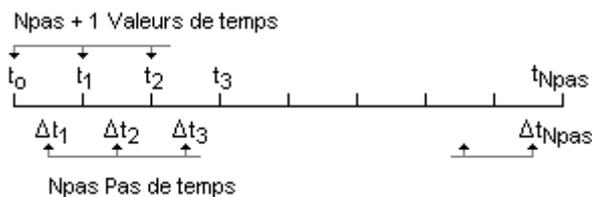
TEXO yields results at each time step or at those time steps specifically selected by the user:

- Primary results: temperature field and state of reaction progress (degree of hydration),
- Secondary results: gradients and flows.

The data required to run the TEXO computation module are defined via a dialog box containing the following tabs.

Tab 1: Time step

This tab enables defining the given time discretization.



Three methods may be employed in order to define the given time values.

1. Constant time step:

According to this method, the user identifies the initial value t_0 of time along with a time step value Δt . The time values considered herein are:

$$t_1 = t_0 + \Delta T,$$

$$t_2 = t_1 + \Delta T, \dots$$

2. Time defined by absolute value:

According to this method, the user directly indicates the $N_{pas}+1$ time values to be taken into consideration.

3. "Group"-defined time steps:

According to this method, the user indicates the initial time value t_0 along with a set of value couples $n_i, \Delta t_i$. It is assumed herein that time step Δt_i remains constant for n_i number of time steps. The time values taken into consideration will be:

$$t_1 = t_0 + \Delta T1$$

$$t_2 = t_1 + \Delta T1$$

...

$$t_{n1} = t_{n1-1} + \Delta T1$$

$$t_{n1+1} = t_{n1} + \Delta T2$$

...

Defining time discretization by means of constant steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Constant time step" option.
4. Input the corresponding time step value.

Defining discretization by means of inputting absolute time values

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time given by absolute value" option.
4. Input the N_{pas} values of time that complete the initial value in the table reserved for this purpose.

Defining discretization by means of "grouping" time steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time step given by group" option.
4. Press the "Add" button as many times as there are time step groups to be included.
5. Identify the couples $(n_i, \Delta t_i)$ in the table reserved for this purpose.

 It should be noted that the sum of the n_i values must equal N_{pas} .

Tab 2: Iterative process

This tab enables defining parameters of the iterative method used for the computation.

Inputting parameters of the iterative process

1. Enter the maximum number of iterations authorized per time step.
2. Set the relative tolerance on the nodal temperature values obtained.

 A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.

3. Select the analysis algorithm:
 - "Multifrontal" method,
 - "Skyline" method.

Tab 3: Monitoring of temperature-related boundary conditions vs. time

This tab serves to define the time-based evolution of the eventual temperatures imposed. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i (defined in the "time step" tab).

The temperatures imposed within a set of boundary conditions at time t_i will be such that:

$$\Theta^j(t_i) = \bar{\Theta}^j \cdot f(t_i)$$

where:

- $\bar{\Theta}^j$: Temperature imposed at node j in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

i If the given model does not contain any imposed temperatures, the data associated with this tab are not to be used.

i If the imposed temperatures do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \ \forall t_i$

i In the special case where several temporal evolutions of temperature-related boundary conditions must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coef.").
3. For each time value, input the corresponding coefficient of "relaxation". This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

i The $N_{ps} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

Tab 4: Monitoring of "loads" vs. time

This tab enables defining the time-based evolution of the eventual loads (flows or flow rates) imposed. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i (defined in the "time step" tab).

The loads imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loads imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

-  If the given model does not contain any imposed loads, the data associated with this tab are not to be used.
-  If the imposed loads do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$
-  In the special case where several temporal evolutions of load conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the load monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
 2. Enter the values of the function $f(t_i)$ for each time value t_i considered.
 3. Repeat steps 2 and 3 for all load cases defined in the model.
-  The $N_{pas} + 1$ number of values characterizing function F may also be input from a file by activating the "Initialization by file input" button.

Tab 5: Storage for restart

This tab allows storing onto a file the set of elements (results obtained from the last time step) required for a subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 6: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the ultimate purpose of:

- producing graphs of the results,
- performing a mechanical analysis using the MEXO module.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" (with primary unknowns: temperature field and degree of hydration) or "Primary and secondary results" (gradients and flows included separately).

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
 2. Select from the proposed list those time steps to be stored using the left mouse button.
 3. Choose the storage type from either "Primary results" (with primary unknowns: temperature field and degree of hydration) or "Primary and secondary results" (gradients and flows included separately).
-  Selected time steps may be "deselected" using the right mouse button.

Tab 7: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Characteristics of the "monitoring" functions (time functions for temperature boundary conditions and imposed loads),
- Primary unknowns - temperature field and degree of hydration,
- Gradients,
- Heat flows.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

Tab 8: The QAB test

The set of computations performed with the TEXO module relies upon the results of a calorimetric test, which must be included as part of the input data. This test consists of recording, as a function of time, the temperature curve of a concrete sample placed in a calorimeter. The figure below indicates, for illustration purposes, this curve under adiabatic conditions ($-\text{div}\mathbf{q} = 0$) for both an ordinary concrete (B25, water/cement ratio of 0.6), and a high-performance concrete (B80, water/cement ratio = 0.4).

The results from this calorimetric test are thus characterized by:

- a set of value triplets (*Time*, *Teta*, *TetaE*),

where: *Time*: Time value at the time of each measurement,

Teta: Temperature value of the sample at the time of each measurement,

TetaE: Temperature value outside the calorimeter at the time of each measurement.

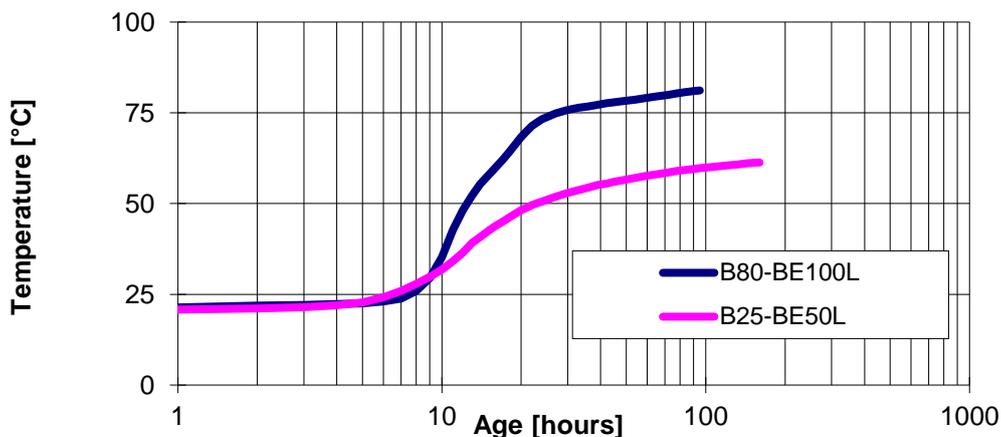
- Three coefficients *A*, *B* and *C* characterizing calorimeter heat losses.

The lost power can be expressed using these three coefficients as follows:

$$P = \frac{A}{C}(Teta - TetaE) + \frac{B}{C}(Teta - TetaE)^2$$

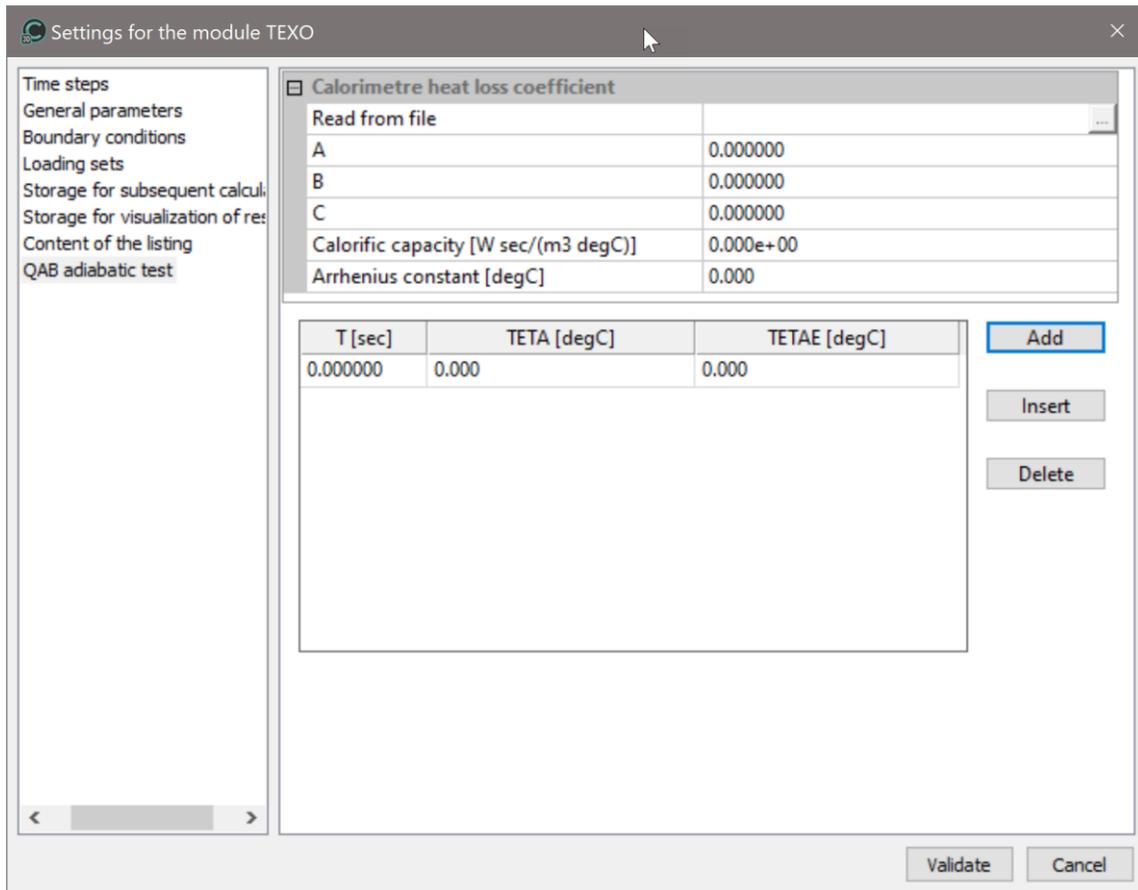
where: *CM*: the calorific capacity of the sample,

XK: Arrhenius Law's constant (if no information is available on this value, set *XK* = 5500).



Temperature histories $T^{ad}(t)$ measured during an adiabatic test
for both an ordinary concrete (B25) and a high-performance concrete (B80)

Defining the characteristics of the QAB test



1. Enter the three coefficients *A*, *B* and *C* characterizing calorimeter heat losses, the calorific capacity of the sample *CM*, and Arrhenius Law's constant *XK*.
 - *A,B,C*
 - *CM*
 - *XK*
2. Enter the triplet values *Time*, *Teta* and *TetaE* in the grid set aside for this purpose.
 - *NbTriplets*: The number of value triplets *Time*, *Teta* and *TetaE*
 - *Triplets*: The specific value triplets *Time*, *Teta* and *TetaE*

i QAB test characteristics may also be input from a file by activating the "Initialization by file input" button. This formatted file must contain the following information:

The MEXO module

The MEXO module is primarily an incremental elastic mechanical solution module with displacement boundary conditions (and eventually with loading as well), within which dilatation and shrinkage of either a thermal or chemical origin are incorporated. Upon completion of a computation using the TEXO module, MEXO is involved in two aspects: determining the mechanical effects (displacements, stresses) due to evolution in both temperature and degree of hydration within early-age concrete structures, and predicting cracking-related risks.

The data required to run the MEXO computation module are defined via a dialog box containing the following tabs.

Tab 1: General parameters

This tab basically serves to define the number of time values considered in the mechanical analysis. This number must equal that entered into the TEXO module used to conduct the corresponding thermal analysis.

Inputting the general parameters

1. Input the number of time values ($N_{pas} + 1$) included in the associated TEXO module.
2. Select the analysis algorithm:
 - "Multifrontal" method,
 - "Skyline" method.
3. Specify the name of the TEXO restart file; this file features syntax of the form "Etude_Mi.prtx", where "Etude" is the name of the project containing the TEXO module, "i" the TEXO module number and "prtx" the assigned file extension.

Tab 2: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the primary unknowns. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a set of boundary conditions at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\} f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

i If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.

2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").
3. For each time value, input the corresponding coefficient of relaxation. This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

 The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

Tab 3: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a load case at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

 If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

 If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

 In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of function $f(t_i)$ for each time value t_i considered.

 The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 4: Storage for restart

This tab allows storing onto a file the set of elements (results obtained from the last time step) required for a subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 5: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Activate the "Storage for visualization of results" tab.
2. Verify that the cell labeled "Storage for all time steps" has not been ticked.
3. Select from the proposed list those time steps to be stored using the left mouse button.



Selected time steps may be "deselected" using the right mouse button.

4. Choose the storage type from either "Primary results" or "Primary and secondary results".

Tab 6: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time steps,
- Characteristics of the "monitoring" functions (time functions for displacement boundary conditions and imposed loads),
- Displacements,
- Stresses.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

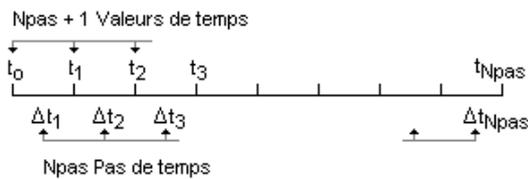
The CSNL module

The CSNL module serves to solve consolidation problems in saturated or quasi-saturated soils (degree of saturation greater than 80%).

The data required to run the CSNL computation module are defined via a dialog box containing the following tabs.

Tab 1: Time step

This tab enables defining the given time discretization.



Three methods may be employed in order to define the given time values.

1. Constant time step:

According to this method, the user identifies the initial value t_0 of time along with a time step value Δt . The time values considered herein are:

$$t_1 = t_0 + \Delta T,$$

$$t_2 = t_1 + \Delta T, \dots$$

2. Time defined by absolute value:

According to this method, the user directly indicates the $Npas+1$ time values to be taken into consideration.

3. "Group"-defined time steps:

According to this method, the user indicates the initial time value t_0 along with a set of value couples $n_i, \Delta t_i$. It is assumed herein that time step Δt_i remains constant for n_i number of time steps. The time values taken into consideration will be:

$$t_1 = t_0 + \Delta T1$$

$$t_2 = t_1 + \Delta T1$$

...

$$t_{n1} = t_{n1-1} + \Delta T2$$

$$t_{n1+1} = t_{n1} + \Delta T2$$

...

Defining time discretization by means of constant steps

1. Indicate the number of pertinent time values $Npas + 1$.
2. Enter the initial time t_0 value.
3. Select the "Constant time step" option.
4. Input the corresponding time step value.

Defining discretization by means of inputting absolute time values

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time given by absolute value" option.
4. Input the N_{pas} values of time that complete the initial value in the table reserved for this purpose.

Defining discretization by means of "grouping" time steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time step given by group" option.
4. Press the "Add" button as many times as there are time step groups to be included.
5. Identify the couples $n_i, \Delta t_i$ in the table reserved for this purpose.

 It should be noted that the sum of the n_i values must equal N_{pas} .

Tab 2 : Iterative process

This tab enables defining parameters of the iterative method used for the computation.

Inputting the iterative process parameters

1. Define the iterative process:
2. Input the maximum number of iterations.
3. Set the relative tolerance on the convergence.

Tab 3: Solution method

The computation of the consolidation problem may use one of the 2 following algorithms:

- "Pardiso" Method,
- "Skyline" method.

Tab 4: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the "primary unknowns". This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a "set of boundary conditions" at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\} f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

 If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

 In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary

conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").
3. For each time value, input the corresponding coefficient of "relaxation". This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

i The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

Tab 5: Monitoring of loading sets

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} \cdot f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given "load case",
- $f(t_i)$: Value of the function at time t_i .

i If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

i If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

i In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several "load cases" (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.

i The $N_{pas} + 1$ number of values characterizing the function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 6: Storage of displacements for restart

This tab allows storing onto a file the displacement computation results from the last time step for subsequent restart.

Defining storage of displacements for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 6: Storage of hydraulic loads for restart

This tab enables storing onto a file the results of hydraulic load-type computations from the last time step for subsequent restart.

Defining storage of hydraulic loads for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 8: Storage of stresses for restart

 In the special case where the current model is of the "Staged construction" type, this tab is deactivated, meaning that all storage and restart operations are conducted automatically.

This tab allows storing onto a file the set of elements (results obtained from the last increment) necessary for proceeding with a subsequent "restart".

Defining storage of stresses for subsequent calculation

1. Tick the cell labeled "Storage for subsequent calculation".
2. Enter the name of the storage file.

Tab 9: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.

 Selected time steps may be "deselected" using the right mouse button.

3. Choose the storage type from either "Primary results" or "Primary and secondary results".

Tab 10 : Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Characteristics of the "monitoring" functions (time functions for load boundary conditions),
- Primary results,
- Secondary results.

Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

The MPNL module

The MPLI and MPNL modules serve to solve linear (MPLI) or non-linear (MPNL) thermo-mechanical evolution problems within saturated porous media.

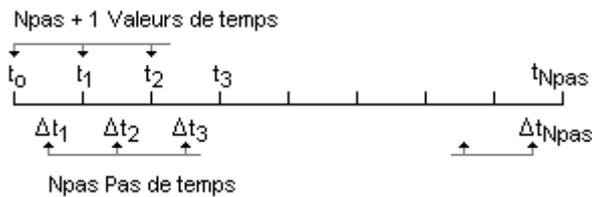
The behavioral laws associated with the MPNL module are as follows:

- thermo-poro-anelasticity (MPNL), isotropic or orthotropic thermo-poro-linear elasticity of revolution (MPLI),
- anisotropic fluid conduction (Darcy's) law,
- anisotropic heat conduction (Fourier's) law.

The data required to run both of these computation modules are defined via a dialog box containing the following tabs.

Tab 1: Time step

This tab enables defining the given time discretization.



Three methods may be employed in order to define the given time values.

1. Constant time step:

According to this method, the user identifies the initial value t_0 of time along with a time step value Δt . The time values considered herein are:

$$t_1 = t_0 + \Delta T,$$

$$t_2 = t_1 + \Delta T, \dots$$

2. Time defined by absolute value:

According to this method, the user directly indicates the $N_{pas}+1$ time values to be taken into consideration.

3. "Group"-defined time steps:

According to this method, the user indicates the initial time value t_0 along with a set of value couples $n_i, \Delta t_i$. It is assumed herein that time step Δt_i remains constant for n_i number of time steps. The time values taken into consideration will be:

$$t_1 = t_0 + \Delta T1$$

$$t_2 = t_1 + \Delta T1$$

...

$$t_{n1} = t_{n1-1} + \Delta T1$$

$$t_{n1+1} = t_{n1} + \Delta T2$$

...

Defining time discretization by means of constant steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Constant time step" option.
4. Input the corresponding time step value.

Defining discretization by means of inputting absolute time values

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time given by absolute value" option.
4. Input the N_{pas} values of time that complete the initial value in the table reserved for this purpose.

Defining discretization by means of "grouping" time steps

1. Indicate the number of pertinent time values $N_{pas} + 1$.
2. Enter the initial time t_0 value.
3. Select the "Time step given by group" option.
4. Press the "Add" button as many times as there are time step groups to be included.
5. Identify the couples $n_i, \Delta t_i$ in the table reserved for this purpose.

 It should be noted that the sum of the n_i values must equal N_{pas} .

Tab 2: Solution method

This tab enables choosing and defining parameters of the iterative method used for the computation.

Inputting parameters of the iterative process

1. Select the solution method applied (either the semi-implicit or implicit configuration).
2. Enter the maximum number of iterations authorized per time step.
3. Set the relative tolerance of the values obtained for the primary unknowns.

 A tolerance of $10^{-3} \leq \text{tolerance} \leq 10^{-2}$ will be classically employed.

Tab 3: Monitoring of boundary conditions on the primary unknowns

This tab serves to define the time-based evolution of the conditions imposed on the primary unknowns. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The conditions imposed on the primary unknowns within a set of boundary conditions at time t_i will be such that:

$$\{U(t_i)\} = \{\bar{U}\}, f(t_i)$$

where:

- \bar{U} : Vector of the primary unknowns imposed in the given set of boundary conditions,
- $f(t_i)$: Value of the function at time t_i .

 If the imposed primary unknowns do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \quad \forall t_i$

 In the special case where several temporal evolutions of boundary conditions on the primary unknowns must be taken into account, it would be necessary to define several sets of boundary

conditions (see chapter entitled: "Initialization of boundary conditions"). It thereby becomes possible to define a different time function for each of these sets.

Entering the boundary condition monitoring coefficients

1. Select the case for the target time function from among the sets of boundary conditions defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered (the column headed "Coefficients").

i The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. For each time value, input the corresponding coefficient of relaxation. This coefficient must equal either 1 (default value) if the boundary conditions are taken into account for the given time or 0 if not.
4. Repeat steps 2 through 4 for all sets of boundary conditions defined in the model.

Tab 4: Monitoring of loading conditions

This tab enables defining the time-based evolution of the loading conditions. This evolution is characterized by inputting a value $f(t_i)$ for each of the considered time values t_i .

The loading conditions imposed within a "load case" at time t_i will be such that:

$$\{F(t_i)\} = \{\bar{F}\} f(t_i)$$

where:

- $\{\bar{F}\}$: Vector characterizing the loadings imposed for the given load case,
- $f(t_i)$: Value of the function at time t_i .

i If the given model does not contain any imposed loadings, the data associated with this tab are not to be used.

i If the imposed loadings do not vary with respect to time, it would simply be necessary to retain the values proposed by default, i.e.: $f(t_i) = 1 \forall t_i$

i In the special case where several temporal evolutions of loading conditions must be taken into account, it would be necessary to define several load cases (see chapter entitled: "Initialization of load cases"). It thereby becomes possible to define a different time function for each of these cases.

Entering the loading monitoring coefficients

1. Select the load case for the target time function from among the load cases defined in the model.
2. Enter the values of the function $f(t_i)$ for each time value t_i considered.

i The $N_{pas} + 1$ number of values characterizing function f may also be input from a file by activating the "Initialization by file input" button.

3. Repeat steps 2 and 3 for all load cases defined in the model.

Tab 5: Storage of displacements for restart

This tab allows storing onto a file the displacement computation results from the last time step for subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 6: Storage of pressures for restart

This tab enables storing onto a file the results of "pressure"-type computations from the last time step for subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 7: Storage of temperatures for restart

This tab enables storing onto a file the results of "temperature"-type computations from the last time step for subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for restart".
2. Enter the name of the storage file.

Tab 8: Storage for subsequent calculation

This tab enables storing onto a file the results from the last time step for subsequent restart.

Defining storage for restart

1. Tick the cell labeled "Storage for subsequent calculation".
2. Enter the name of the storage file.

Tab 9: Storage for visualization of results

This tab basically serves to select the desired time steps with which results are to be saved for the production of subsequent graphics.

Storing results for all time steps

1. Tick the cell labeled "Storage for all time steps".
2. Choose the storage type from either "Primary results" or "Primary and secondary results".

Storing results for only selected time steps

1. Verify that the cell labeled "Storage for all time steps" has not been ticked.
2. Select from the proposed list those time steps to be stored using the left mouse button.
3. Choose the storage type from either "Primary results" or "Primary and secondary results".



Selected time steps may be "deselected" using the right mouse button.

Tab 9: Listing contents

This tab allows defining the component elements of the computation note, thereby making it possible to obtain all or part of the following:

- Time step characteristics,
- Characteristics of the "monitoring" functions (time functions for load boundary conditions),
- Primary results,
- Secondary results.

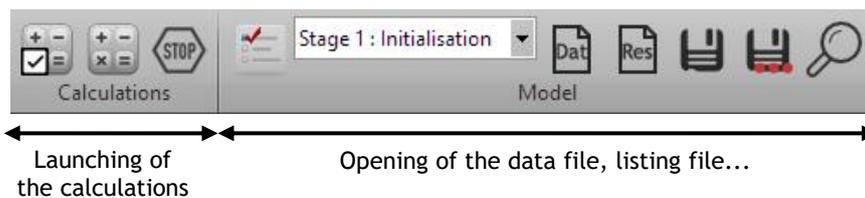
Defining listing contents

1. Tick those cells corresponding to the specific entities sought on the listing.

14. Computation management

Introduction

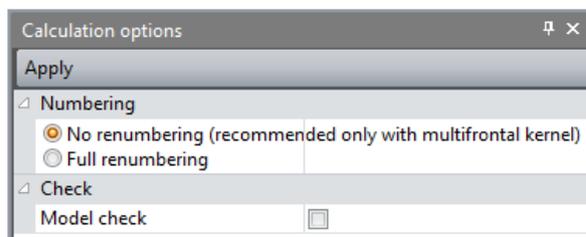
Activating the "Computation management" tab basically serves to execute computations for the specified models. The toolbar associated with this tab is displayed as follows:



Two options are proposed for the purpose of optimizing the amount of model preparation time.

Renumbering is only necessary in the case where the multifrontal solver has not been selected (see the previous chapter). If this solver is chosen, the dataset generation time becomes longer.

Model verification ensures that all model elements are correctly connected and moreover that no single element is isolated. Such a situation would cause a computational error (of the "Negative or zero pivot" type). This verification step may wind up being long; also, this option is not activated by default.



The various tools of the toolbar will be described below.



General analysis options

This tool serves to perform the following settings:

- Type of "renumbering": With or without renumbering. The renumbering process is basically used to minimize the bandwidth of the matrices introduced. Activating the "With renumbering" option is thus recommended for all "solvers" relying on a direct solution method (AXIF, LIGC, TACT, DTLI, SSTR, SURF, NAPP, LINC, LINH, SUMO, CSLI, MPLI, MPNL). In the case the analysis is being performed with the "Pardiso" or the "Multifrontal" solver, renumbering is of no utility; it is thus recommended to activate the "Without numbering" option in order to avoid executing an operation that often consumes considerable computation time. It is important to pay close attention to the special case where one model uses the results of another model. For example, let's cite the definition within a "static model" of thermal loadings by reading from a file the temperatures resulting from the "heat exchange model" computation. For this operation to be

valid, the two models would have to be "compatible", which requires making sure to activate the same "Type of renumbering" option for both models.

- Model verification: If this option is checked (default value), the software will verify model consistency before undertaking the computations.

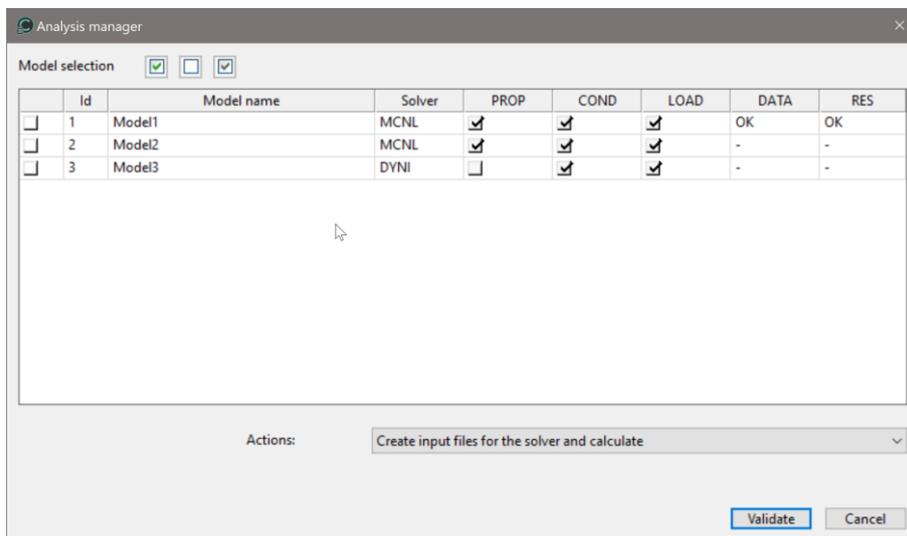


Analysis manager

This tool oversees the effective execution of computations for the selected models. Its activation displays a dialogue box of the type shown below.

When the model is calculated, its status is displayed:

- OK, when calculations are consistent with the iteration process,
- other message: erreur with iteration proces (non convergence), database...



As example below, we read:

- that the model « Model1 » is calculated,
- that the model « Model2 » is ready for calculation but is not calculated,
- that the model « Model3 » is not ready for calculation as the material properties are not completed.

This dialogue box presents the set of models created by the user. To perform analyses on any of these models, the following procedure must be implemented.

Performing computations

1. Check (left-hand column) the models for which an analysis is to be conducted.



Only completed models, those with validated « Properties », « Boundary conditions » and « Loadings », can be calculated.

2. In the "Actions" drop-down list, choose the "Data file creation and execution of computations" option. The "Data file creation" choice simply creates the CESAR solver "input" data file. This text file may be visualized and edited by the user (see the "Data file visualization" tool). In the special case where this file has been modified by the user, it would be necessary to select the "Solve with

the existing data file" option from the "Actions" drop-down list so that the user's modifications are taken into account.

3. Press "Validate".



Interrupting the computations

This tool allows the user to interrupt the computation sequence at any time.

The user must realize that using this tool results in the loss of all information that has not been saved.



Editing the data file

This tool enables visualizing and ultimately editing the data file corresponding to the current model selected from the "Models" drop-down list.



Editing the "listing"

This tool enables visualizing and ultimately editing the results listing generated by the solver and corresponding to the current model selected from the "Models" drop-down list.



Save, Save as, etc.

These two tools serve to save the file being visualized in the working window (i.e. data file or results listing).



Search, etc.

This tool is used to search for a character chain in the file being visualized in the working window (data file or results listing).



Show the mesh of the model

Feature in beta version.



Mesh display options

Feature in beta version.

15. Display of results

Introduction

This module serves to graphically visualize the primary results obtained for the given "current model". Results of the following types may be represented herein:

- Deformed mesh of the model (for the case of mechanical problems),
- Vectors,
- Tensors,
- Iso-surface,
- Forces within beam- and truss-type elements.

These results may be superimposed on the same view. It is thereby possible to visualize the iso-surfaces associated with a given scalar and primary stresses, on the structure represented in the deformed position.

The "Type of results to display" option enables selecting the types of results to be visualized. For each result type, it is possible to set a certain number of parameters, such as:

- Choice of entity to be visualized (type of vector, tensor, iso-surface scalar, etc.),
- Scales,
- Color palettes.

A specialized tool is dedicated to defining the parameters associated with each type of result. These settings will thus be entered by means of the following tools:

- Deformed mesh options (case of mechanical problems),
- Vector options,
- Tensor options,
- Iso-surface options,
- Options relevant to forces within beam- and truss-type elements.

Tools used to visualize results

The visualization of results associated with the current model requires the following tools:

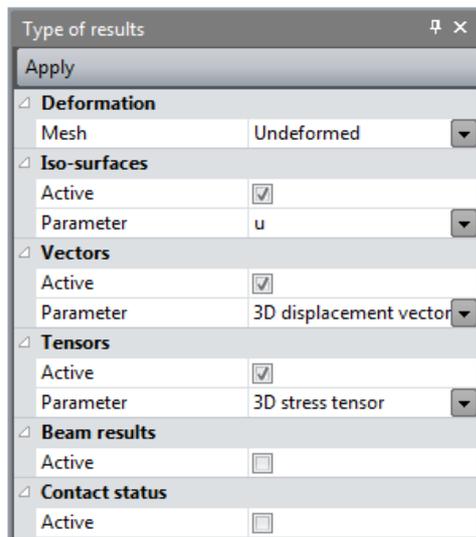
-  Choice of type of results to be displayed
-  Envelope of iso-surface scalar
-  Animation or choice of "entity" (increment, time step, etc.) to be displayed
-  Mesh settings
-  Deformed mesh settings
-  Iso-surface settings

-  Vector settings
-  Tensor settings
-  Beam and truss result settings
-  Contact state display
-  Legend settings
-  Cutting plane
-  Plane section analysis
-  Informations
-  Results displayed by body



Type of results to display

Activating this tool displays a dialog box that enables selecting the entities to be visualized. This dialog box contains six "sections".



The first allows the user to choose the mesh visualization mode. The following choices are proposed:

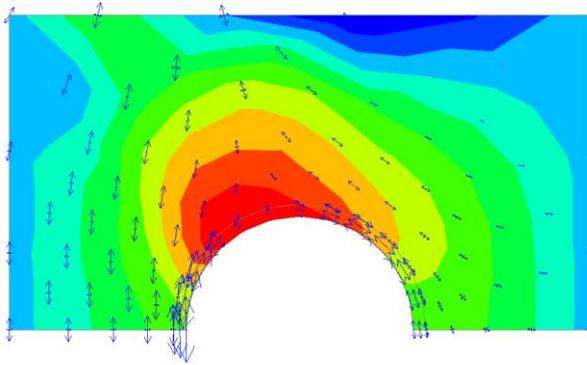
- Display mesh in its original shape,
- Display mesh in its deformed shape.

The five following sections serve to define the types of results to be displayed. Five types are proposed herein:

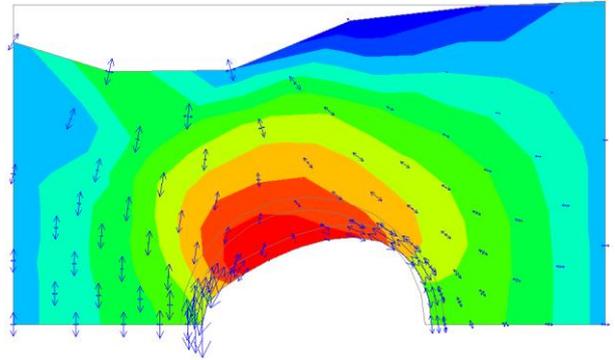
- Iso-surfaces,
- Vectors,
- Tensors,
- Beam and truss results,
- Contact status (if the model contains contact elements).



It should be pointed out that these results may be superimposed. For example, it is possible to visualize the isovalues associated with a given criterion by superimposing the primary stresses.



Isovalues + primary stresses



Isovalues + primary stresses on the deformed mesh

Defining the type of results to be displayed

1. Activate the "Choice of type of results for display" tool.
2. Indicate the mesh visualization mode.
3. Tick the types of results to be visualized concurrently.
4. "Apply" to enter these selections.



Envelope of iso-surface results

Activating this tool displays a dialogue box that allows visualizing the "envelope" iso-surface results for the selected scalar on the set of "increments" or "time steps" of the given model.

Display the iso-surface envelope

1. Activate the "Envelope of iso-surface results" tool.
2. Check the box labeled " iso-surface envelope".
3. Choose one of the two options "Envelope of minimum values" or "Envelope of maximum values".
4. Press "Apply" to record the choices made.



Choice of entity

This tool enables choosing the entity (increment, time step, etc.) targeted by the results visualization sequence. The following table indicates the generic entity notion for each of the CESAR-LCPC computation modules eligible to be applied herein.

Field of application:	Module	Entities =
Statics	LINE	Loading set
	MCNL	Load increments
	TCNL	Load increments
Dynamics	DYNI	Time steps
	MODE	Eigenmodes
	SUMO	Time steps
	FLAM	Buckling modes
	LINC / LINH	-
Hydrogeology	DTNL	Time steps
	NSAT	Time steps
Heat transfer	DTNL	Time steps
Early age concrete	TEXO	Time steps
	MEXO	Time steps
Consolidation	CSNL	Time steps
Coupled problems	MPNL	Time steps

If the model has been analyzed for several entities, then user has the choice between two types of animation:

- animation showing evolution of one result for the current entity;
- animation showing evolution of one result in function of the set of entities.

Animation of the results

1. Choose the type of animation.
2. If choice is « Current entity », animation is defined by two parameters: number of period (duration of the animation) and number of drawings per period (flow of the animation). The animation is commanded by following buttons:



with:

- ▶ Start of the animation
- End of the animation

3. If choice is « All entities », animation is commanded by following buttons:



with:

- ▶ Start of the animation
- End of the animation

Button-assisted entity selection

1. Choose the type of animation "All entities"
2. Use the commands:



with:

- ⏪: Return to the first entity
- ⏮: Return to the previous entity
- ⏭: Continue to the next entity
- ⏩: Select the last entity

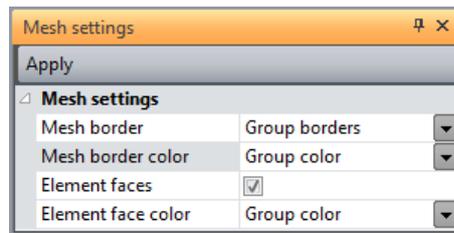
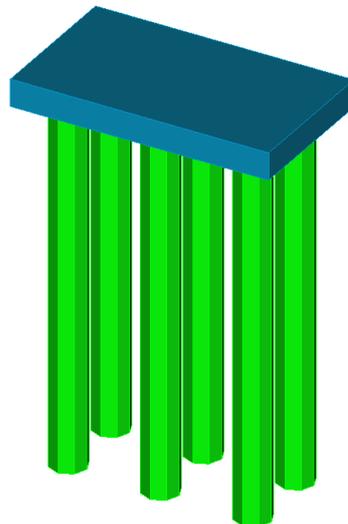
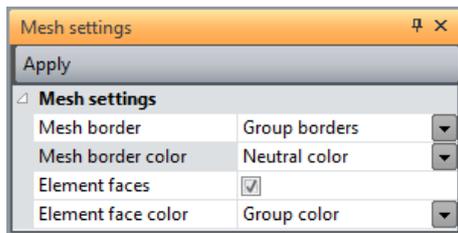
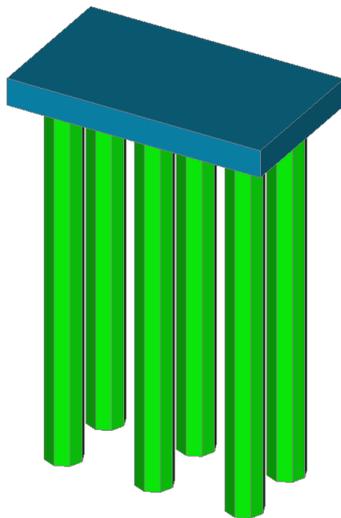


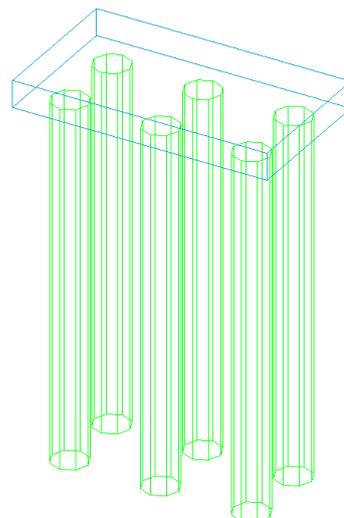
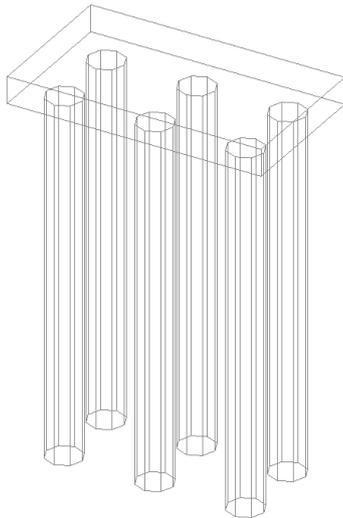
Mesh settings

This option allows setting the mesh display mode and makes it possible to display the following:

- Mesh borders,
- Element borders,
- Faces (colored elements).

Each of these entities may be displayed either in a neutral color (light gray) or in its defined color. The table below indicates of the possible combinations.





Mesh settings ⌵ ✕

Apply

▾ Mesh settings

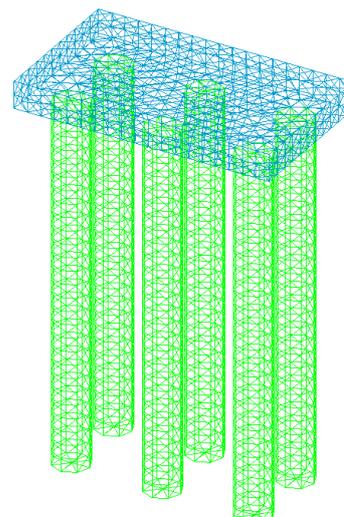
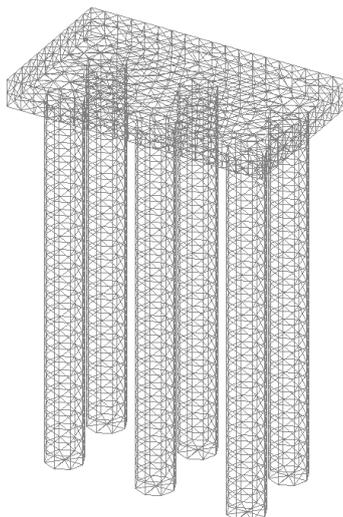
Mesh border	Group borders
Mesh border color	Neutral color
Element faces	<input type="checkbox"/>
Element face color	Group color

Mesh settings ⌵ ✕

Apply

▾ Mesh settings

Mesh border	Group borders
Mesh border color	Group color
Element faces	<input type="checkbox"/>
Element face color	Group color



Mesh settings ⌵ ✕

Apply

▾ Mesh settings

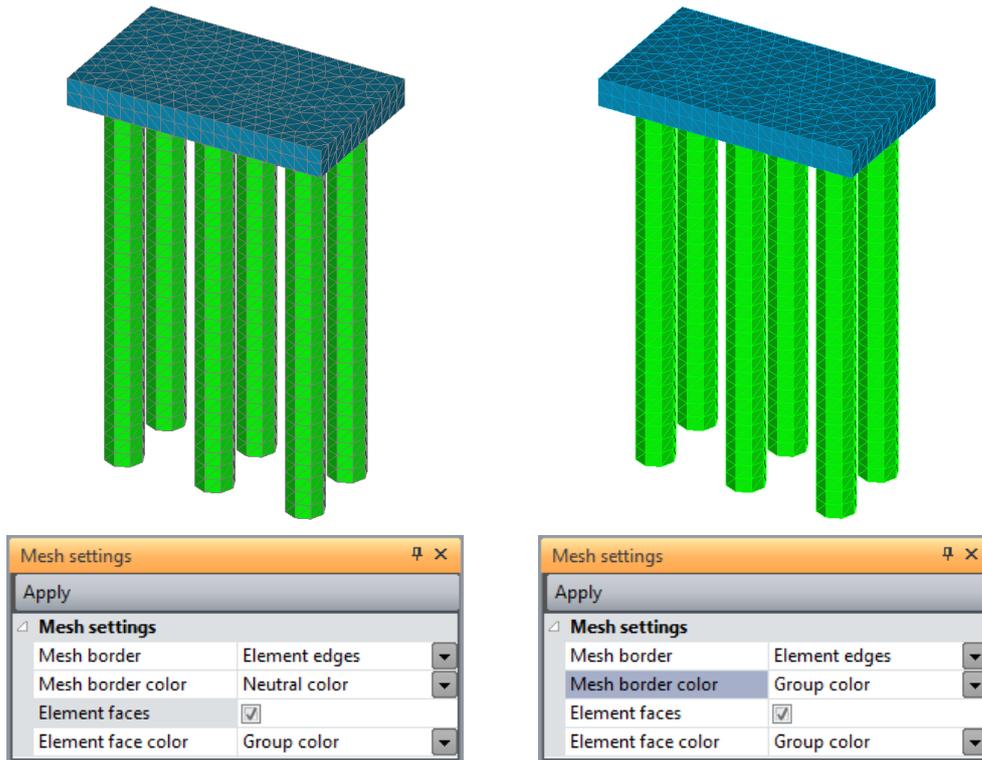
Mesh border	Element edges
Mesh border color	Neutral color
Element faces	<input type="checkbox"/>
Element face color	Group color

Mesh settings ⌵ ✕

Apply

▾ Mesh settings

Mesh border	Element edges
Mesh border color	Group color
Element faces	<input type="checkbox"/>
Element face color	Group color



Defining the mesh settings

1. Activate the "Mesh settings" tool.
2. Select the entities (group boundaries, element boundaries, facets) to be displayed.
3. Choose the colors to be associated with each entity.
4. "Apply" to enter these choices.



Deformed mesh settings

This tool allows setting the display parameters for the model's deformed mesh. Its utility is limited to those modules giving rise to displacement-based results.

Three parameters may be defined herein:

- the deformed mesh scale,
- the preference whether or not to display the initial position of the model as represented by body boundaries.
- The "reference model", set to display the increment of displacement between this model and the active one.

Two methods may be used to set the deformed mesh scale. According to the first or so-called "automated" method, the magnitude of the maximum displacement is represented by a standard length equal to a percentage of the model size. With this method, representation of the maximum deformed mesh magnitude is therefore independent of the actual value itself. The user may then adjust the percentage of the size of the model used to represent the deformed mesh.

According to the second or so-called "manual" method, the user defines the correspondence existing between a given displacement value and its graphical representation. It may be stated, for example, that a 10-mm displacement will be represented by a 1-m length. This method may be used to graphically compare the results of several computations.

In case of "staged construction" analysis, the last section proposes to display incremental deformed mesh by comparing current stage to one of the preceding ones.

This option is also useful for displaying the incremental value of the current displacement.

Setting parameters for displaying the deformed mesh

1. Activate the "Deformed mesh settings" tool.
2. Choose the applicable scale-definition mode (automated or manual).
3. Specify the parameters associated with the given scale-definition mode.
4. Tick the option labeled "Group boundaries" should it also be desired to visualize the initial model shape.
5. Press the "Apply" button to enter these choices.

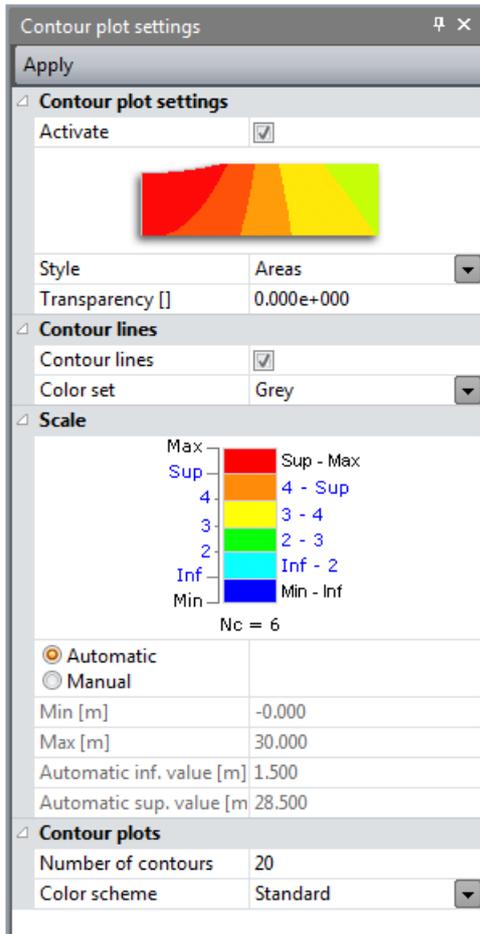
Setting parameters for displaying the incremental deformed mesh

1. Activate the "Deformed mesh settings" tool.
2. Choose the applicable scale-definition mode (automated or manual).
3. Specify the parameters associated with the given scale-definition mode.
4. Tick the option labeled "Group boundaries" should it also be desired to visualize the initial model shape.
5. Activate "Incremental displacement". Specify the reference model from the list.
6. Press the "Apply" button to enter these choices.



Contour plot settings

This tool serves to set the parameters characterizing the display of contour plots. Activating this option generates a dialog box that contains four sections.

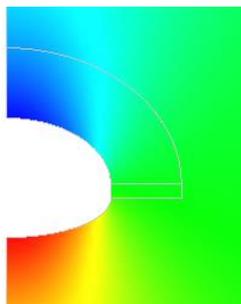


The first section allows setting the display mode for the given scalar plot and, as an initial step, defining the presentation "style" of these scalar plots. Three styles are proposed herein:

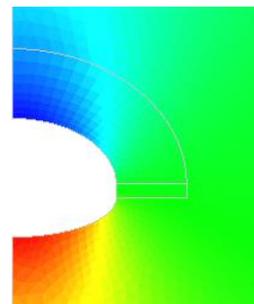
- Constant color per area,
- Gradient scale of colors,
- Average color per element.



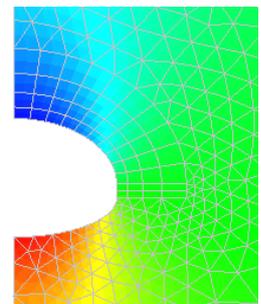
Constant color per areas



Gradient color

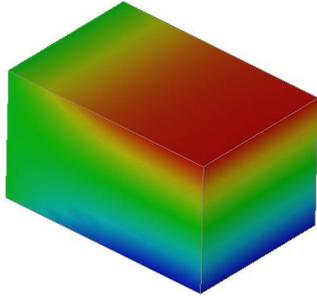


Average color per sub-element

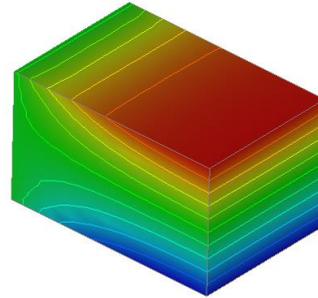


Contour plots + Contour lines

In addition to the contour plot display, it is possible to add contour-lines of the scalar. These will clearly show the limit between two of the scalar values.

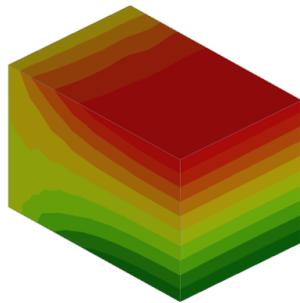


Gradient color contour plots

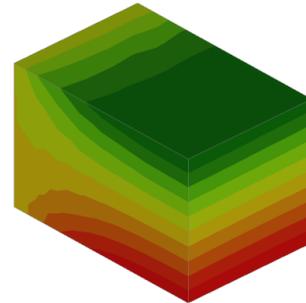


Contour plots + Contour lines

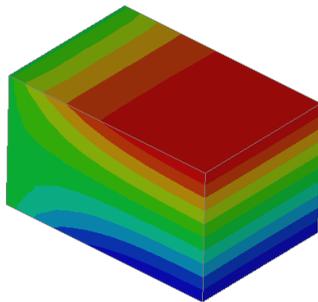
This option also makes it possible to choose the "color palette" for representing contour plots, by means of a drop-down list proposing a set of predefined palettes.



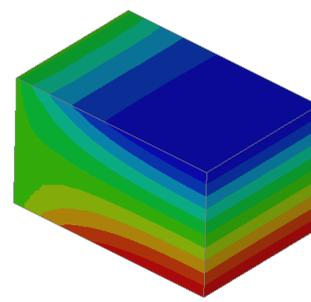
"CESAR style" Palette



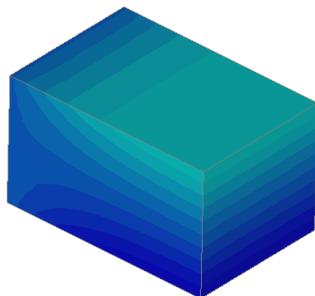
"CESAR style"(inversed) Palette



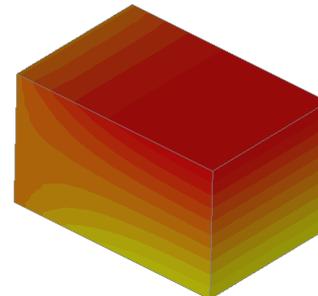
"Standard" Palette



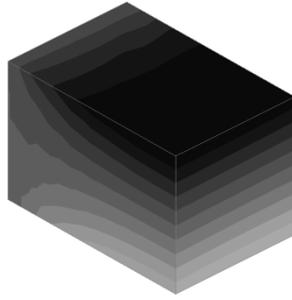
"Standard (inversed)" Palette



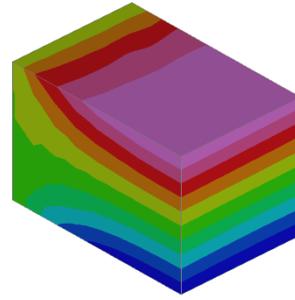
"Groundwater" Palette



"Temperature" Palette



"Black & White" Palette

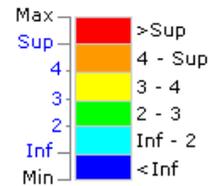


"Mechanical" Palette

In addition to the style defined above, it is possible to adjust the "Inf" and "Sup" values that characterize both the extreme values of the scalar to be represented and the number of intervals taken into consideration. Below, we have presented a characteristic scale of values comprising six color intervals.



Continuous representation of colors



Zone-by-zone representation

For a given scalar, the software computes the "Min" and "Max" values corresponding to the extreme points of the target scalar value. As a default, the "Min-Max" interval is divided into n (with n being defined by the user) equidistant intervals. Within the value levels defined in this manner, we are able to distinguish the "Inf" and "Sup" values positioned above the minimum value and below the maximum value, respectively. Regardless of the style chosen to represent the isovalues, the "Min-Inf" and "Sup-Max" zones display uniform colors.

Both the "Inf" and "Sup" values may be modified by the user; it is thereby straightforward to visualize, for example, those zones for which the target scalar exceeds the given "Sup" value.

Setting the scalar to display

1. The scalar displayed as a contour plot is associated to the one defined in "Type of results to display".

Setting the contour plots display

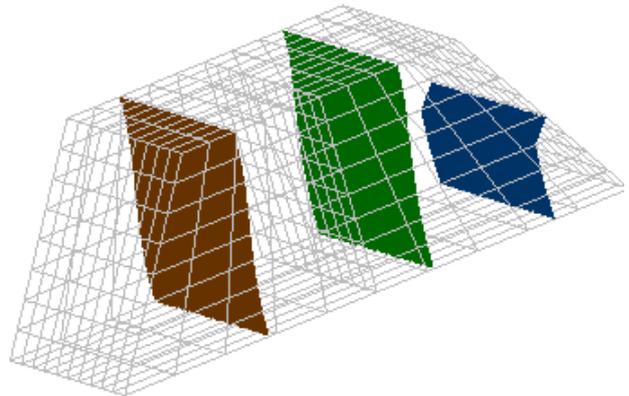
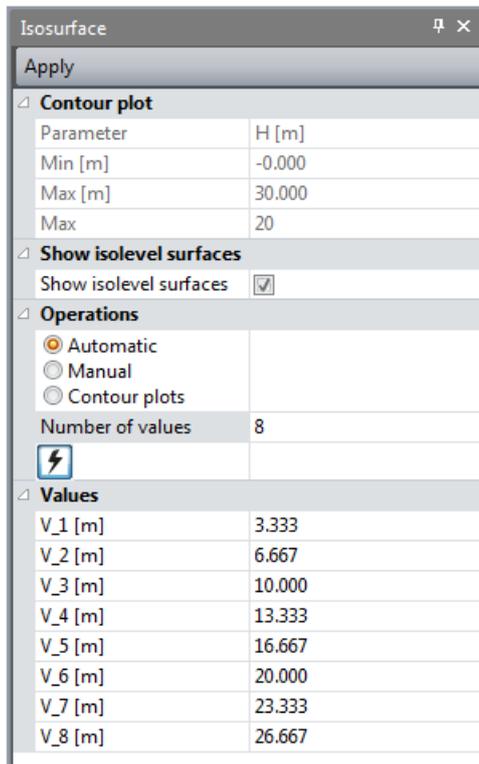
1. Tick "Activate" for activation des isovaleurs.
2. Select a display style (areas, gradient or average).
3. Define a level of transparency (0 = opaque ; 1 = transparent)
4. Tick the "Contour lines" for activation.
5. Choose the appropriate color palette.
6. Select the "Automatic" or "Manual" scale.
7. If "Manual scale", input the "Inf" and "Sup" values that characterize the color representation scale.
8. Specify the number of (color) intervals to be included.

9. Choose the appropriate color palette.
10. "Apply" to enter these choices.



Isosurfaces

This tool serves to set "isolevel surfaces". As example on the figure below, three isolevel surfaces relative to three hydraulic load values are displayed.



Isosurfaces in a dike

Setting the scalar to display

1. The scalar displayed as an isolevel surface is associated to the one defined in "Type of results to display" with settings defined in "Contour plot settings".

Setting the isolevel surfaces visualization parameters

1. Activate the tool "Isosurfaces". Contour plots settings are recalled.
2. Tick "Show isolevel surfaces" for activation.
3. Select one of the isolevel surfaces definition method:
4. If "Automatic", define the number of isolevel surfaces. Use the tool ⚡ to generate the corresponding number of surfaces, consistent with the contour plot settings and then number of isolevel surfaces.
5. If "Manual", user defines the values of isosurfaces, using (+) for adding, or (-) for deleting. The eraser tool 🗑️ deletes the list.
6. If "Contour plots", settings in "Contour plots settings" are used Use the tool ⚡ to generate the corresponding number of surfaces, consistent with the contour plot settings.
7. "Apply".

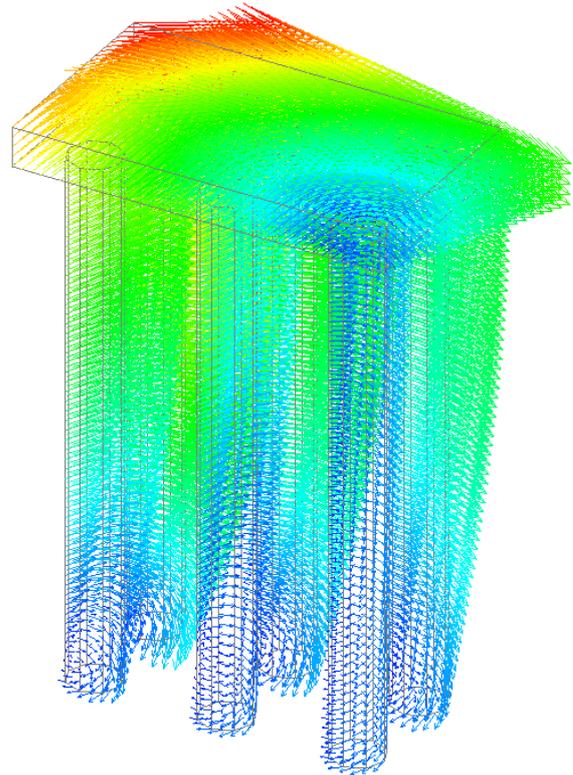
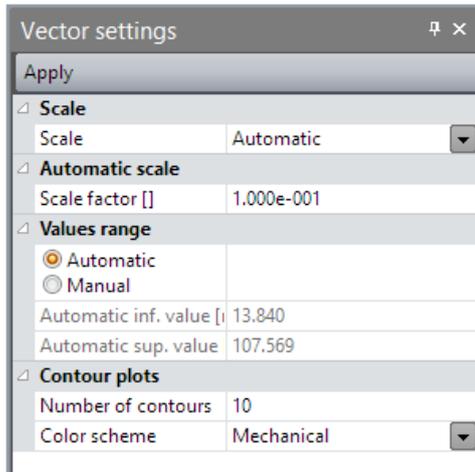
Hiding the isosurfaces

1. Activate the tool "Isosurfaces".
2. Untick " Show isolevel surfaces".
3. "Apply".



Vector options

This tool allows setting the parameters that characterize the display of vector-type results.



Displacement vector within a pile group

Activating this option generates a dialog box that serves to enter any eventual adjustments to the scale used for representing vector magnitude.

Two methods may be used to set this scale. According to the first or so-called "automated" method, the maximum vector magnitude is represented by a standard length equal to a percentage of the model size. With this method, representation of the maximum vector magnitude is therefore independent of the actual value itself. The user may then adjust the percentage of the size of the model used to represent these vectors.

According to the second or so-called "manual" method, the user defines the correspondence existing between a given vector magnitude and its graphical representation. This method may be used to graphically compare the results of several computations.

Next, it is possible to adjust the colors used for representing vectors. Just like the case for iso-surfaces (see the section on "Iso-surfaces options"), it is possible herein to define the following parameters:

- Color palette,
- The inferior and superior limits, "Inf" and "Sup", of the values characterizing the color representation scale,

- The number of color levels.

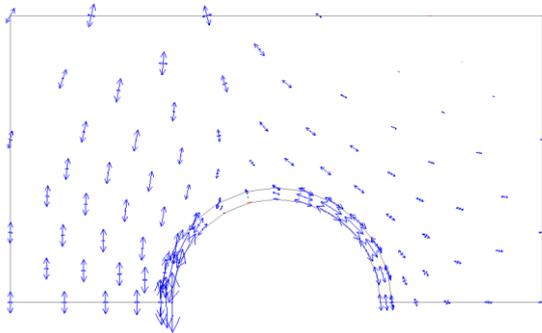
Setting the vector visualization parameters

1. Activate the "Vector options" tool.
2. Indicate the scale-definition mode (automated or manual).
3. Enter the parameters associated with the particular scale-definition mode:
4. If "Automatic", set the "Scale factor". The value is the ratio between the display length and the maximal dimension of the model.
5. If "Manual", set the display length for a given value of the scalar.
6. Input the "Inf" and "Sup" values that characterize the color representation scale.
7. Specify the number of (color) intervals to be included.
8. Choose the appropriate color palette.
9. Press the "Apply" button to enter these choices.



Tensors settings

This tool allows setting the parameters that characterize the display of tensor-type results.



Stress tensors around an excavated area

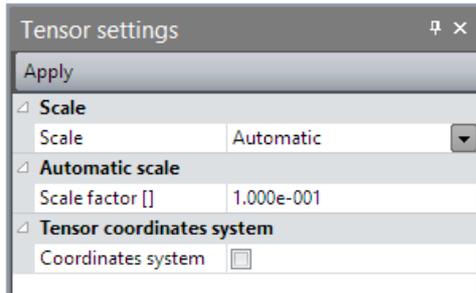
Activating this option generates a dialog box that allows entering any eventual adjustments to the scale used for representing the magnitude of tensor components.

Two methods may be used to set this scale. According to the first or so-called "automated" method, the maximum magnitude of the tensor components is represented by a standard length equal to a percentage of the model size. With this method, representation of the maximum magnitude is therefore independent of the actual value itself. The user may then adjust the percentage of the size of the model used to represent these tensors.

According to the second or so-called "manual" method, the user defines the correspondence existing between a given tensor component magnitude and its graphical representation. This method may be used to graphically compare the results of several computations.



For the tensors, the negative components (compression) will be represented in blue and the positive components (tension) in red. This solution makes it possible to easily distinguish compressed zones from tensioned zones.



Setting the tensor visualization parameters

1. Activate the "Tensor options" tool.
2. Indicate the scale-definition mode (automated or manual).
3. Enter the parameters associated with the particular scale-definition mode
4. If "Automatic", set the "Scale factor". The value is the ratio of the maximum display length to the largest dimension of the model.
5. If "Manual", set the display length of a specified value.
6. Activate the coordinate system settings for representing the tensor (either the primary coordinate system or the local coordinate system). If the local system is preferred, enter the angle existing between the X axis of the global coordinate system and the x axis of the local system.
7. Press the "Apply" button to enter these choices.



Beam results settings

With this option, the user is able to set the results visualization parameters (N, V, M) for beam-type structural elements.



It should be noted herein that these forces will only be represented on the structure in the undeformed position.

Activating this option produces a dialog box that serves to enter any eventual adjustments to the scale used for representing the forces.

Two methods may be used to set this scale. According to the first or so-called "automated" method, the maximum magnitude of the target force is represented by a standard length equal to a percentage of the model size. With this method, representation of the maximum magnitude is therefore independent of the actual value itself. The user may then adjust the percentage of the size of the model used to represent these forces.

According to the second or so-called "manual" method, the user defines the correspondence existing between a given force magnitude and its graphical representation. This method may be used to graphically compare the results of several computations.

Setting the beam results visualization parameters

1. Activate the "Beam results options" tool.
2. Indicate the scale-definition mode (automated or manual).
3. Enter the parameters associated with the particular scale-definition mode
 - If "Automatic", set the "Scale factor". The value is the ratio of the maximum display length to the largest dimension of the model.
 - If "Manual", set the display length of a specified value.
4. Specify if the minimal and maximal values of the selected result are to be displayed.
5. Press the "Apply" button to enter these choices.

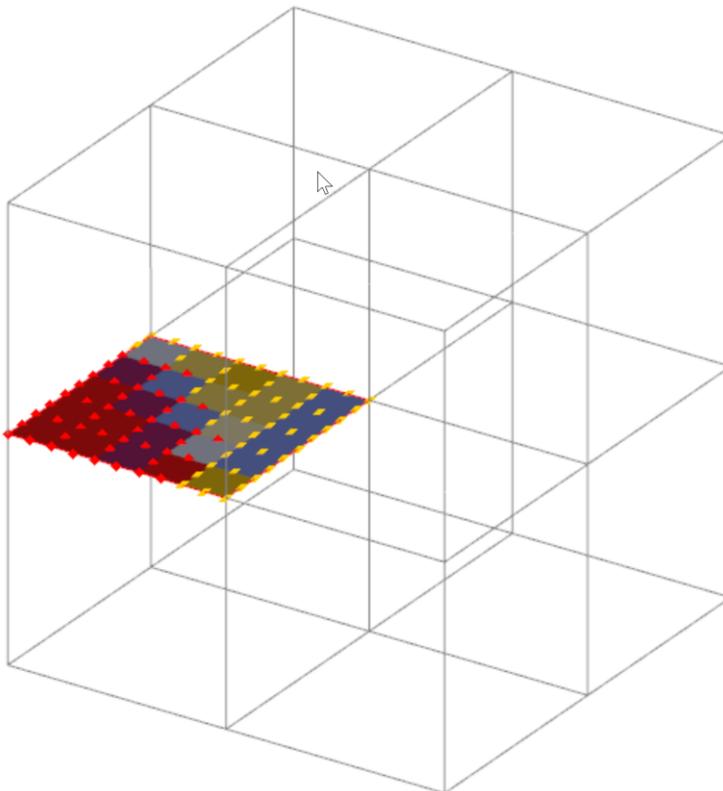
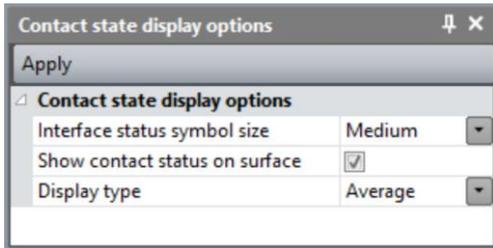


Contact state display options

This tool allows setting the display of the contact elements state (adherent, sliding, open).

Display contact state

1. Activate the tool « Display options of contact state ».
2. Set the size of the elements.
3. Set the display of the contact surfaces.



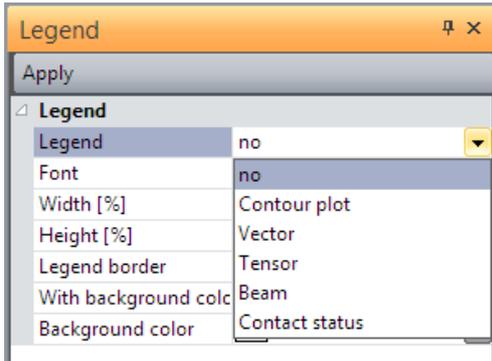


Legend

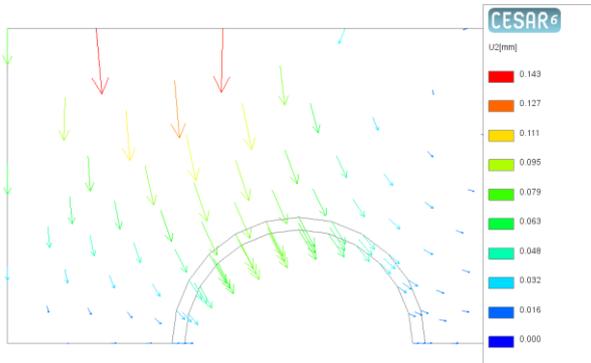
This tool allows setting the legend associated to the displayed results.

Display a legend

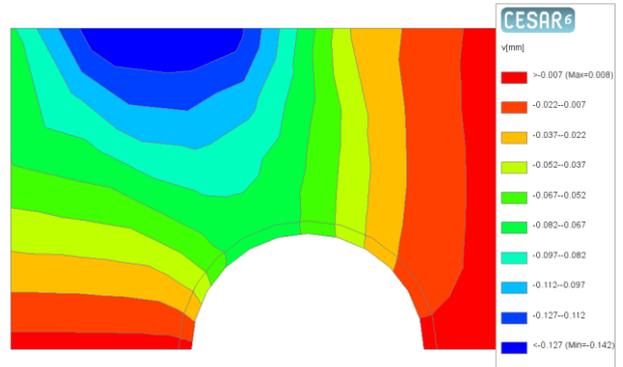
1. Activate the « Legend » tool.
2. Select the type of legend to display.
3. Set the display : font, background, border, height and width.



The display of the legend is linked with the choice of the scalar set in « Type of results to display ».



Legend associated with the displacement vector plot

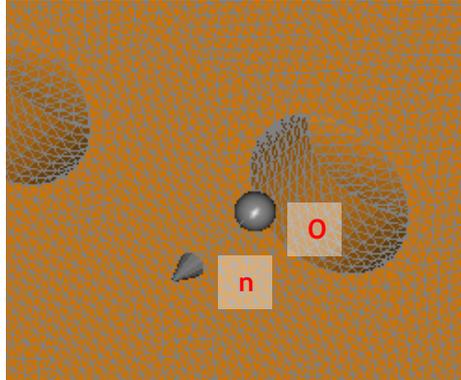


Legend associated with the isovalue plot



Cutting plane

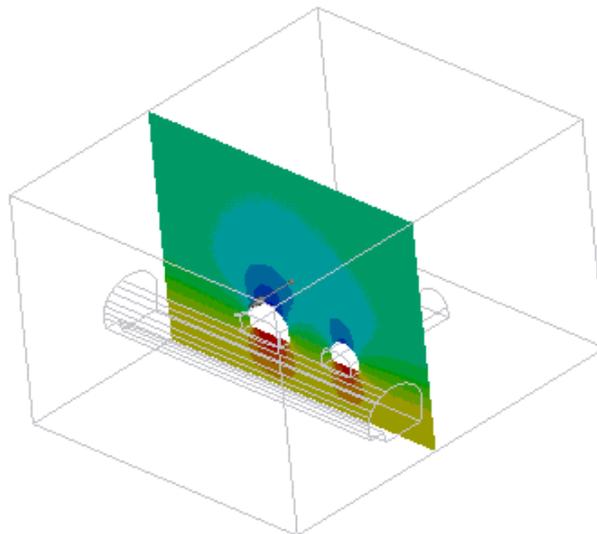
This tool basically enables visualizing results inside the structure. Its activation generates a dialog box that first serves to define a plane by both the position of an origin point O and a normal \vec{n} to the plane.



This plane features two properties that the user may elect to activate.

- Cutting plane: if this property is activated, the contour plot of the current scalar will be represented within the cutting plane.
- Clip plane: if this property is activated, the model portion located in front of the plane will be considered invisible.

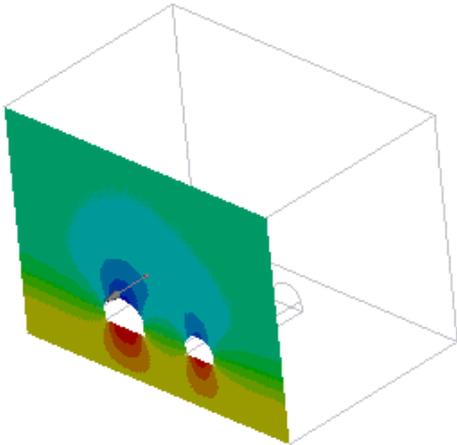
The combination of these two properties allows obtaining several types of potential representations of great utility in proceeding with an accurate interpretation of the results. We will display a few of these possibilities below.



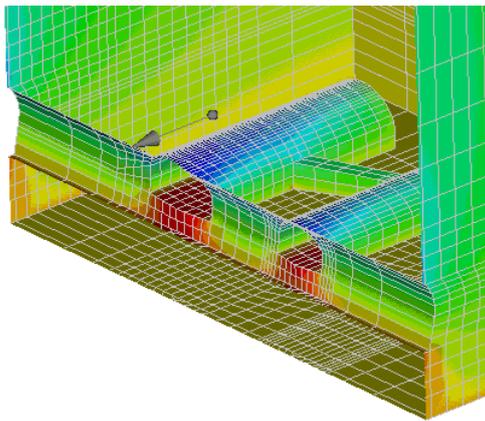
With cutting plane - without clip plane

In the above example, the mesh has been simply represented by the group edges. It is used here to ease the visual understanding of the plane position. The plane has been ascribed the "cutting plane" property; the scalar plot of the current scalar are thus represented in this same plane. Moreover, the "Clip plane" property has not been applied herein: the portion of the structure located in front of the plane has therefore remained visible.

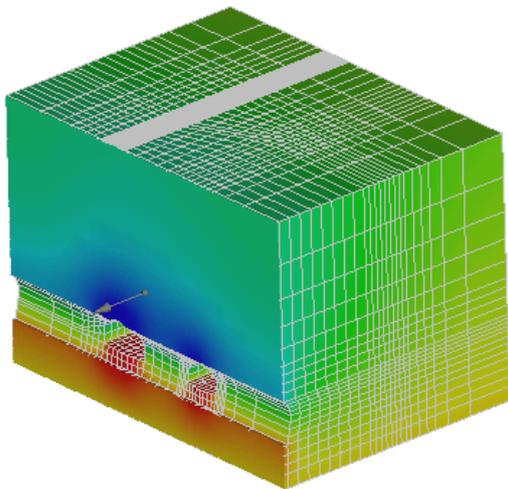
In holding all other parameters unchanged, we would obtain the following view if the "Clip plane" property were to be activated.



It is also quite helpful to combine the "Clip plane" property with a complete representation of the mesh (i.e. drawing of the facets). In the view displayed below, we have modified the mesh visualization options in order to activate both the facet layout and element edges.

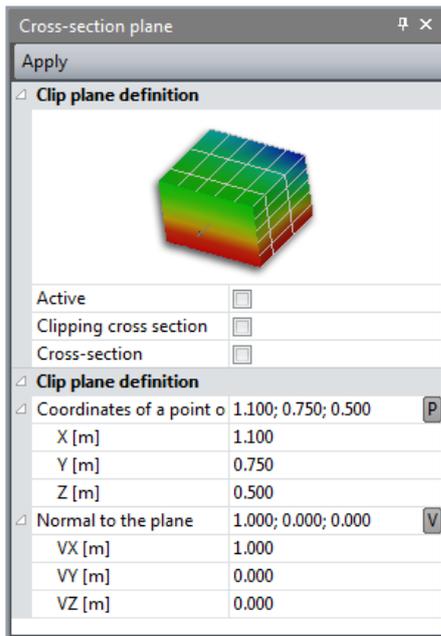


If we were to deactivate the "cutting plane" property while maintaining the "clip plane" property active, the plane under investigation would become transparent, which would then allow visualizing results on the internal facets composing the model's group boundaries. The following view shows the results obtained using this type of configuration.



Defining a cutting plane

1. Activate the "Cutting plane" tool.
2. Set both the "Cutting plane" and "Clip plane" parameters in order to obtain the desired type of visualization.
3. Indicate the position of the plane by inputting the coordinates of an origin point and the components of a normal vector.
4. Press the "Apply" button to enter these choices.



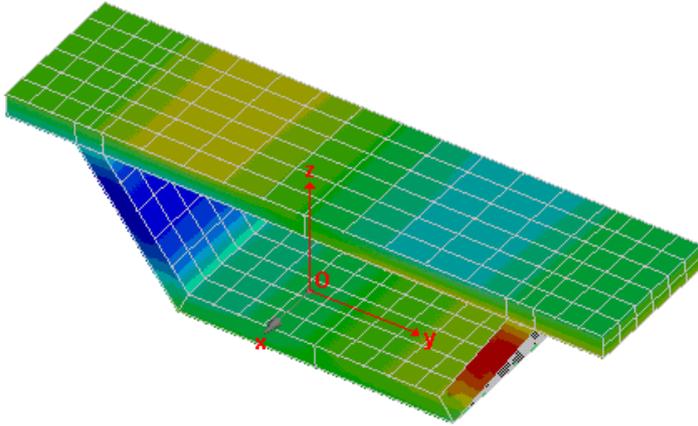
Modifying the cutting plane position interactively

1. Activate the "Cutting plane" tool.
2. Tick the cell labeled "Active" to visualize the arrow that symbolizes the cutting plane.
3. Set both the "Cutting plane" and "Clip plane" parameters to obtain the desired type of visualization.
4. Indicate the position of the plane by inputting the coordinates of an origin point and the components of a normal vector.
5. Press the "Apply" button to enter these choices.
6. Activate the "Selection" option.
7. Position the mouse on the origin point of the plane. While holding down the left mouse button, move the position so as to modify the origin point characterizing the cutting plane.



Plane section analysis

This tool serves to compute the reduction forces N , V_y , V_z , M_y , M_z and M_t being exerted in a plane defined by the user on the model's visible volume bodies.



The considered plane is defined by the position of both an origin point and the components of the two vectors y and z .

Computing the reduction forces within a given plane

1. Activate the "Plane section analysis" tool.
2. Specify the position of the origin point O and the components of the two vectors V_y and V_z .
3. Tick the cell marked "Update the cutting plane" if the indicated plane is to be overlapped with the cutting plane, thereby allowing for visualization of its position.
4. Activate the "Calculate" button to obtain the reduction forces within the given plane.

Saving the resultant information in a text file

1. Activate the "Plane section analysis" tool.
2. Specify the position of the origin point O and the components of the two vectors V_y and V_z .
3. Activate the "Calculate" button to obtain the reduction forces within the given plane.
4. Click the "Save as" button. A dialog box enables you to enter the name of the destination file in which the reduction force data is to be stored. The extension of this newly-created file will be set as a default to ".info".
5. Define a new plane position and once again activate the "Calculate" button.
6. Click the "Save" button. The information corresponding to the new plane will then be saved in the previously-created file.



The stored information may be saved in ASCII files, with rows by using the separator character "," (comma). It is thereby very simple to recover this information with any spreadsheet application in order to perform special processing.



Information

This tool generally serves to recover information pertaining to model results associated with nodes and beam elements. Activating this tool displays a dialog box whose content depends on the nature of the results being visualized. When this dialog box is open, a right mouse click on the particular node or beam element suffices to recover the pertinent information.

When using the "Choice of result types to be displayed" tool, if node results are entered, the information dialog box will produce all or part of the following output:

- Number of the clicked node,
- Node coordinates,
- Current vector components (selected under "Vector options"),
- Current tensor components (selected under "Tensor options"),
- Value of the current scalar used for representing isovalues (selected under "Isovalue options").

Should beam element results be entered, the following information will be produced:

- Number of the clicked element,
- Origin N, extreme N, middle N,
- Origin Vy, extreme Vy, middle Vy,
- Origin Vz, extreme Vz, middle Vz,
- Origin My, extreme My, middle My,
- Origin Mz, extreme Mz, middle Mz,
- Origin Mt, extreme Mt, middle Mt.

Recording information

1. Activate the "Information" tool.
2. Right click on the node or beam element of the given mesh. The results associated with the selected node will then be displayed in the dialog box.

Saving information in a text file

1. Activate the "Information" tool.
2. Right click on the node or beam element of the given mesh. The results associated with the selected node will be displayed in the dialog box.
3. Click the "Save As" button. A dialog box will then allow entering the name of the file in which the target information is to be stored. As a default, the ".info" file extension will be assigned.
4. Right click on another mesh node or beam element.
5. Now click the "Save" button. The information corresponding with the clicked node or element will be saved in the previously-created file.
6. Click the "Close" button to end the prompt sequence and close the dialog box.



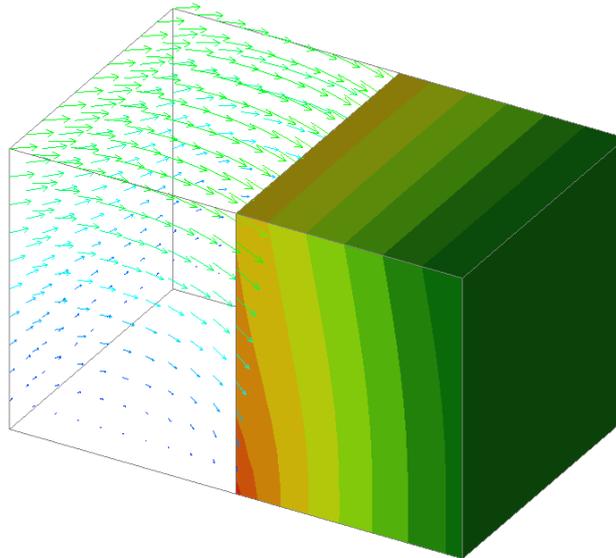
The stored information may be saved in ASCII files, with rows by using the separator character "," (comma). It is thereby very simple to recover this information with any spreadsheet application in order to perform special processing.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Noeud	170	x	0.647	m	y	-0.47	m	z	0	m	u	0.25	mm
2	Noeud	138	x	0.761	m	y	-0.247	m	z	0	m	u	0.28	mm
3	Noeud	141	x	0.713	m	y	-0.363	m	z	0	m	u	0.35	mm
4	Noeud	170	x	0.647	m	y	-0.47	m	z	0	m	u	0.42	mm



Results displayed by group

This tool serves to specify the display of certain types of results for one or more mesh groups.



Example: Displacement vectors in the left-hand volume, displacement isovalues in the right-hand volume

Display specific results on a selected group

1. Activate the "Displayed results" tool.
2. Record the volume data by means of a right-click.
3. Check the type or types of results to display on the selected group: isovalue, vector, tensor.
4. Press "Apply".



Listing of results

This tool displays the results file generated by the "solver" for the current model. Activating any other tool in the toolbar returns to the graphic visualization of results.

16. Charts of results

Introduction

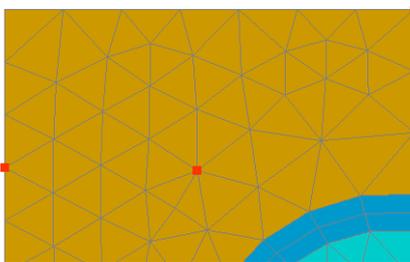
In addition to the set of graphic results displayed directly on the model (isovalues, vectors, tensors, etc.), it is possible to obtain results in the form of "x-y charts". The three following types of charts are available herein:

- Chart for line sets: evolution of a parameter along a line set. These curves are of the type $u = f(x)$, where u is the considered parameter and x the curvilinear abscissa along the line set;
- Chart for a set of points: evolution of a parameter as a function of time (or increment) for a given set of points;
- Parametric chart: evolution of a scalar as a function of another scalar for a given set of points.

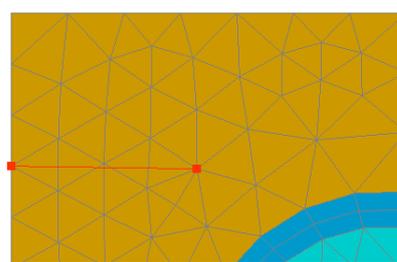
We can observe that the plot of these charts presupposes the definition of the two following entities:

- Line set,
- Set of points.

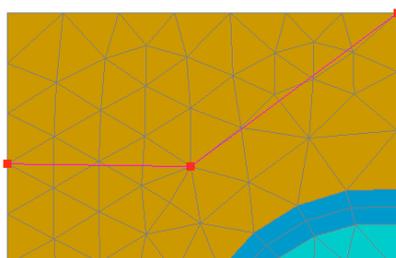
A line set is composed of a set of lines and edges, both of which are necessarily continuous. The term "line" implies the straight-line segments created within the present module that serve to connect model points or nodes. "Edges" refer, in classical terms, to the external element boundaries.



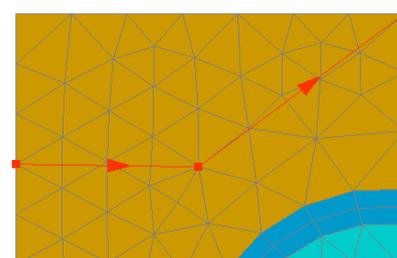
Points



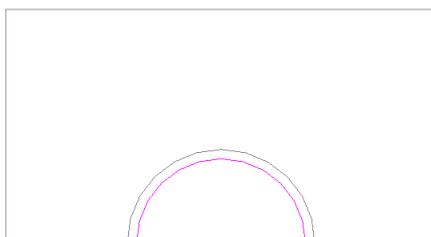
Line



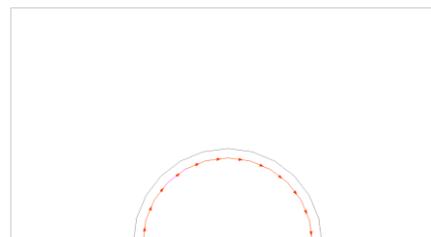
Selected lines



« Line set » created from selected lines



Selected edges



« Line set » created from selected edges



Points

This tool allows defining one or several points capable of being used either for integration within a set of points or for the construction of cross section support lines.

Defining points

1. Activate the "Points" tool.
2. Enter a name for the point to be defined. This name is then used in the chart reference system. For example, when we plot the evolution of a parameter as a function of time for a set of points. This type of chart consists of N curves, where N is the number of points contained within the set. In order to distinguish these various curves, the name of the point will be used.
3. Enter the X, Y and Z coordinates of the point.
4. Press the "Apply" button to create the defined point.
5. Repeat steps 2 and 3 for the entire set of points to be defined.
- 6.



If the coordinates of the point coincide with those of an existing node, it is then possible to recover these coordinates by right-clicking on the node.



Set of points

This tool serves to define one or several sets of points.



In order to designate a group of points as belonging to a set, it is necessary to have predefined the points using the "Points" tool.

Defining a set of points

1. Activate the "Set of points" tool.
2. Select those points intended to belong to the target set.
3. Assign a name to the target set of points.
4. Press the "Apply" button to create the defined set.
5. Repeat steps 1 through 4 for any new set of points.

Visualizing a set of points

1. Activate the "Set of points" tool.
2. Select the set of points from the proposed list.
3. Click the " Show" button. All points belonging to the given set are then highlighted within the graphics window.



Line

This tool enables defining the lines capable of being used in the generation of cross sections. The term "line" implies a continuous set of straight-line segments connecting either points created within the present module or existing model nodes.

Defining a line

1. Activate the "Lines" tool.
2. Click on the successive set of points or nodes constituting the line (or broken line).
3. Double click on the last point or node in order to end the line-definition sequence. This final point is acknowledged by a green coloring.



Line set

This tool enables defining one or several cross sections. A cross section is based on existing edges of the model or lines as defined in previous section.

Defining a line set

1. Activate the "Line set" tool.
2. Select the lines and/or segments constituting the line set. To run this tool, these lines and segments must be pointwise continuous.
3. Assign a name to the target line set.
4. Press the "Apply" button to create the defined line set. This definition is acknowledged by an arrow indicating the direction of each element composing the path.
5. Repeat steps 1 through 4 for any new line set.

Visualizing a line set

1. Activate the "Line set" tool.
2. Select the line set from the proposed list.
3. Click the "Show" button. The target line set is then highlighted within the graphics window.



Reverse the line set orientation

Each line set includes a "direction", which is used to define the increasing curvilinear abscissas x of the chart. The present tool enables reversing the default direction assigned to the selected line sets.

Reversing the cross-section direction

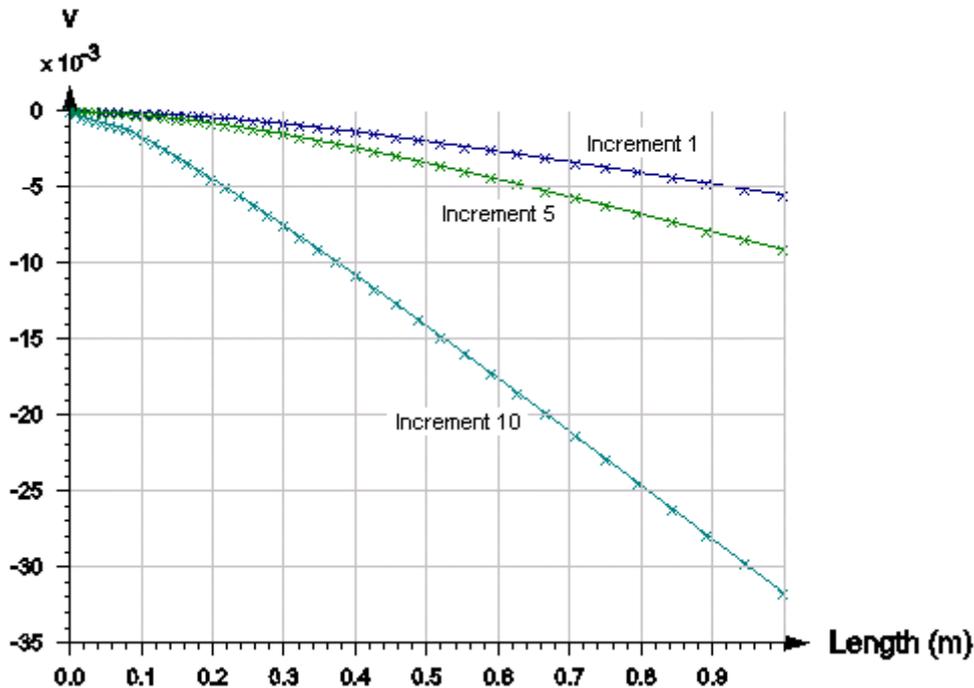
1. Select those line sets whose direction is to be reversed.
2. Activate the "Inverse the line set orientation" tool.



Charts for a line set

This tool allows obtaining x-y charts that reveal the evolution of a given parameter along a line set, one of those previously defined. On this series of curves, the x axis reflects the curvilinear abscissa developed along the line set, while the y axis displays the chosen parameter.

In the special case where the given model corresponds with a time- or increment-based computation, the evolution of the target parameter may be obtained for a list of user-defined time steps (or increments).



The parameters of the depicted chart may be adjusted by running the "Chart settings" tool.

i In order to run the present tool, it is necessary to have predefined at least one line set.

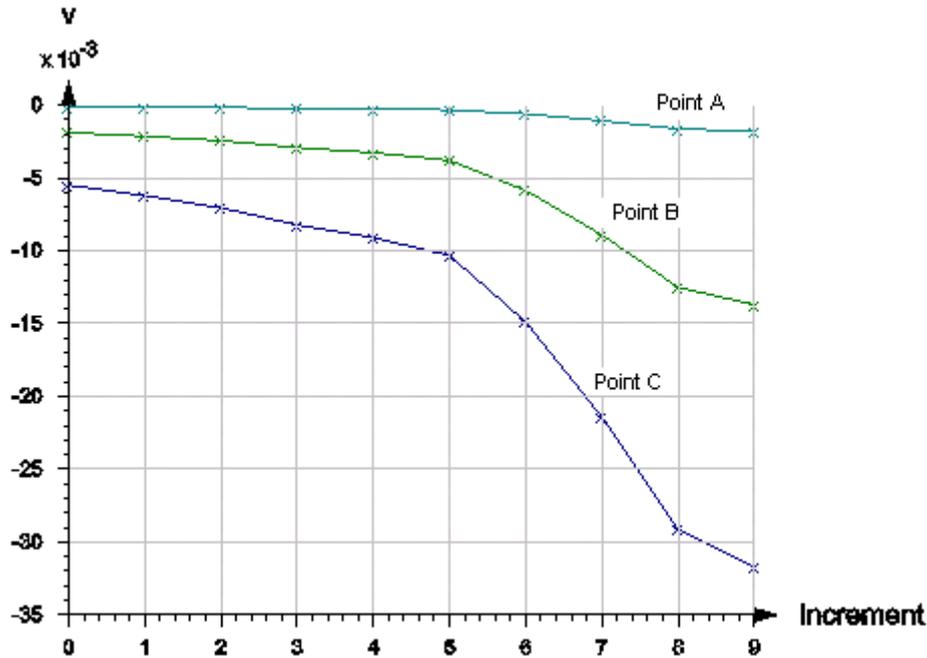
Defining "Charts for a line set"

1. Activate the "Charts for a line set" tool.
2. Select the parameter whose evolution along the line set is to be studied from the drop-down list entitled "Choice of parameter".
3. Choose the target line set from the corresponding drop-down list.
4. Select the increments from the list of available.
5. Press the "Apply" button. The chart is now generated.



Charts for a set of points

This tool allows obtaining a chart that displays the evolution of a parameter as a function of time (increments) for a set of points. A point-by-point curve is plotted when the tool is run.



In order to run the present tool, it is necessary to have predefined at least one set of points.

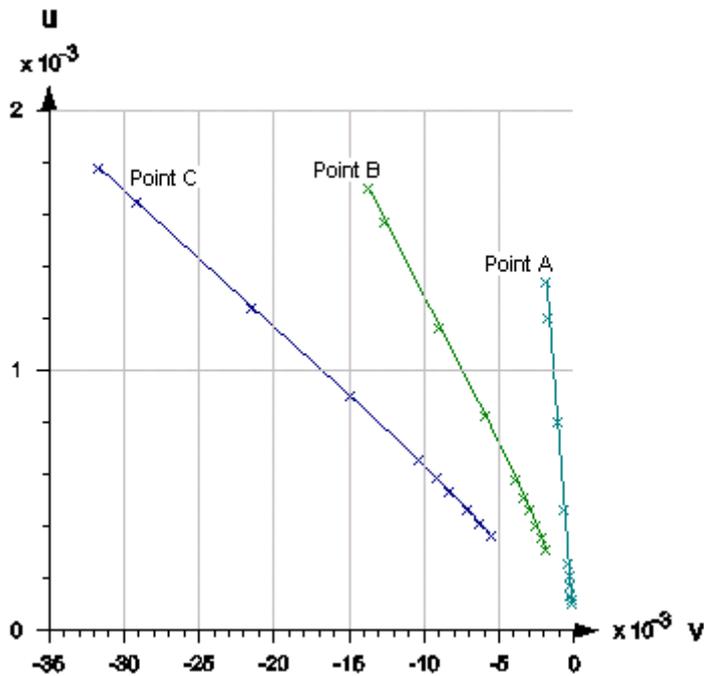
Defining "Charts for a set of points"

1. Activate the "Charts for a set of points" tool.
2. Select the given parameter from the proposed drop-down list.
3. Choose the target set of points from the second available drop-down list.
4. Press the "Apply" button. The curve is now generated.
5. Repeat steps 2 to 4 to modify the selected parameter or set of points.



Parametric charts

This tool allows obtaining a chart that displays the time-based (or increment-based) evolution of a parameter as a function of another parameter for a given set of points. A point-by-point curve is plotted when this tool is run.



In order to run the present tool, it is necessary to have predefined at least one set of points within the "Definition of point and cutting path sets" module.

Defining "Parametric charts"

1. Activate the "Parametric charts" tool.
2. Select the x-axis parameter from the first drop-down list presented.
3. Select the y-axis parameter from the second drop-down list.
4. Choose the target set of points.
5. Press the "Apply" button. The curve is now generated.
6. Repeat steps 2 to 4 to modify the selected parameters or set of points.

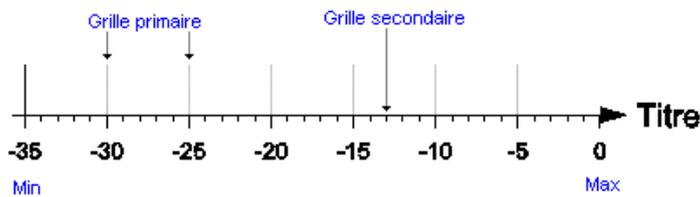


Chart settings

This tool serves to adjust the settings of the previously-defined charts elements (axes, curves, etc.). Activating this tool displays a grid with several sections.

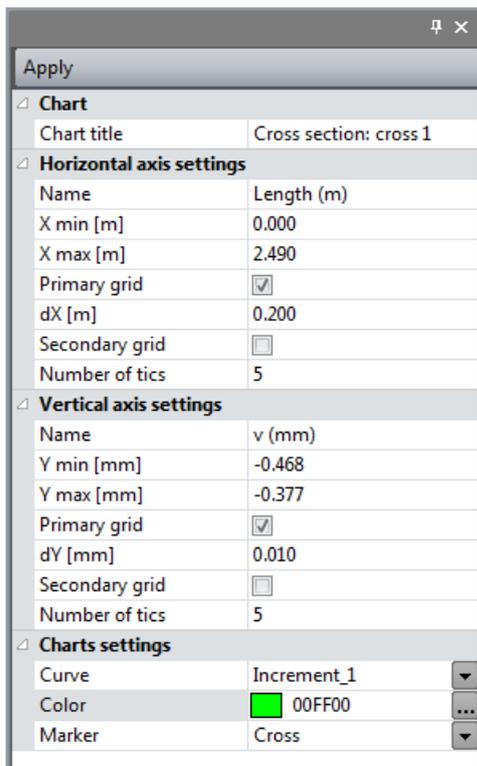
Two sections serve to parameterize the x and y axes. For each axis, it is possible to specify the following elements:

- Axis title,
- The Min and Max values that define the value variation interval along the given axis,
- The interval separating two lines of the primary "grid",
- Definition of the secondary grid by inputting the number of intervals (tick marks) between two primary grid lines,
- Visibility of both the primary and secondary grids.



The third section allows setting the representation mode for each curve of the chart, thereby making it possible to adjust the following:

- Curve plot color,
- Type of "marker" used for curve points.



Adjusting charts settings

1. Activate the "Charts settings" tool.
2. Modify the chart title
3. Set the parameters settings for the X axis.
4. Set the parameters settings for the X axis.
5. Set the parameters (colors, marker type) of each curve.
6. Click the "Apply" button to terminate and record the inputted settings.



Export charts data

This tool enables storing the (x,y) information used for the curves of the current chart onto a text file.

Export charts data

1. Activate the "Export charts data" tool.
2. Enter the name of the storage file for saving the pertinent information.

i The stored information will be saved in an ASCII format that allows for easy "recovery" with any spreadsheet application for special processing.

	A	B	C	D
1	Length (m) - v			
2	X	Incr 1	Incr 5	Incr 10
3	1	-0.005502	-0.00911	-0.0317
4	0.94661	-0.005114	-0.00849	-0.0298
5	0.89322	-0.004726	-0.00786	-0.02791
6	0.84468	-0.004373	-0.00729	-0.02619

A 1. Elements library

All the elements in CESAR are grouped in “families”. The following table describes the available element families in the current version of the software.

Mechanics

Family 1	Two-dimensional isoparametric displacement-type elements
Family 2	Three-dimensional isoparametric displacement-type elements
Family 3	Two-dimensional beam element
Family 4	Three-dimensional beam element
Family 5	Shell elements
Family 6	(2D, 3D) Contact elements
Family 7	Two-dimensional bar elements
Family 8	Three-dimensional bar elements
Family 11	Axisymmetrical isoparametric displacement-type elements for the computation of structures with a revolution-based geometry submitted to any loading
Family 16	(2D, 3D) Joint elements
Family 435	(2D, 3D) Bar elements with friction

Diffusion

Family 21	Two-dimensional isoparametric diffusion-type elements
Family 22	Three-dimensional isoparametric diffusion-type elements
Family 23	Two-dimensional exchange elements
Family 24	Three-dimensional exchange elements
Family 25	Discontinuous elements for identifying a free surface (plane problems)
Family 27	Elements for compatibility of interface elements (F6 and F16) in diffusion analysis
Family 28	Elements for compatibility of structural elements (F3, F4, F5, F7 and F8) in diffusion analysis

Coupling

Family 41	Two-dimensional isoparametric elements with three degrees of freedom per node (two displacements, hydraulic load) for consolidation problems
Family 42	Three-dimensional isoparametric elements with four degrees of freedom per node (three displacements, hydraulic load) for consolidation problems
Family 45	Two-dimensional isoparametric elements with four degrees of freedom per node (two displacements, pore pressure, temperature) for thermomechanical problems in porous media
Family 46	Three-dimensional isoparametric elements with five degrees of freedom per node (three displacements, pore pressure, temperature) for thermomechanical problems in porous media

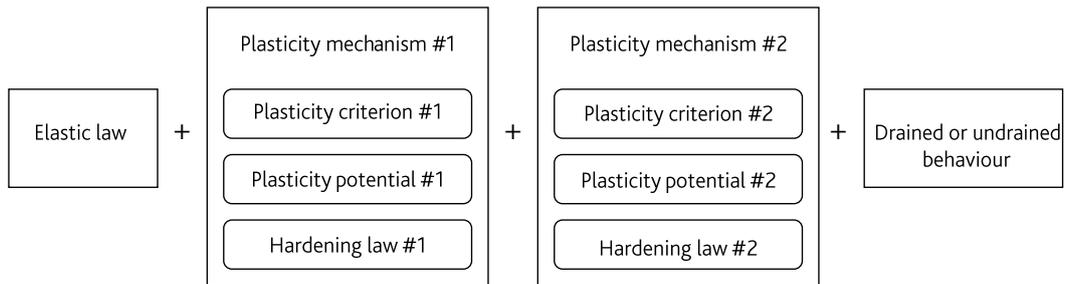
A2. Constitutive models

Constitutive laws for the element families used in mechanics and geomechanics.

Family	Constitutive model	Type of material
Family 1	<i>Elasticity:</i>	
	– isotropic linear	
	– orthotropic linear	
	– with isotropic dilatancy	
	<i>Elastoplasticity, criterion:</i>	
	– Mohr-Coulomb (with or without strain hardening, with or without orthotropic elasticity)	sand, clay, rock
	– Tresca	clay
	– Von Mises (with or without strain hardening)	metals
	– Drücker-Prager (with or without strain hardening)	sand, clay, rock
	– parabolic	concrete
	– Vermeer	sand
	– Nova	sand
	– modified Cam-Clay	clay
	– Prevost and Hoëg	rock
	– oriented	stratified medium
	– Melanie	clay
	– Willam - Warnke	concrete
– Hoek - Brown	rock	
<i>Other:</i> early-age concrete	concrete	
Family 2	<i>Elasticity:</i>	
	– isotropic linear	
	– orthotropic linear	
	– with isotropic dilatancy	
	<i>Elastoplasticity, criterion:</i>	
	– Mohr-Coulomb	sand, clay, rock
	– Von Mises (with or without strain hardening)	metals
	– Drücker-Prager (with or without strain hardening)	sand, clay, rock
	– parabolic	concrete
	– Vermeer	sand
	– Nova	sand
	– modified Cam-Clay	clay
	– Prevost and Hoëg	rock
	– oriented	stratified medium
	– Willam - Warnke	concrete
	– Hoek - Brown	rock
	<i>Other:</i> early-age concrete	concrete
Families 3, 4, 435	<i>Linear elasticity</i>	
	+ plasticity	
	+ brittle failure	
	+ perfect plasticity + brittle failure	
	+ loss of stiffness	
	+ perfect plasticity	

Families 5, 7, 8 and 11	<i>Linear isotropic elasticity</i>
Family 6	<i>Contact laws:</i> <ul style="list-style-type: none"> - bonding - Coulomb's friction - perfect sliding
Family 26	<i>Interface laws:</i> <ul style="list-style-type: none"> - bonding - Coulomb friction - perfect sliding - linear elasticity - elastoplasticity with traction limit
Family 435	<i>Interface laws:</i> <ul style="list-style-type: none"> - linear elasticity - linear elasticity + perfect plasticity

Toolbox for user constitutive model definition (only for families 1 and 2)



Elastic laws

- Homogeneous isotropic linear elasticity
- Homogeneous orthotropic linear elasticity
- Isotropic linear elasticity with E and ν varying linearly with depth
- Isotropic linear elasticity with $E = f(z)$
- Non-linear elasticity of the Hardening Soil Model
- Non-linear elasticity of Cam-Clay
- Non-linear elasticity of Duncan
- Non-linear elasticity of Fahey-Carter

Plasticity criterions

- Coulomb
- Mohr-Coulomb
- Hardening Soil Model
- Hoek-Brown
- Mohr-Coulomb with c and $\varphi = f(z)$
- Drucker-Prager
- Modified Cam-Clay
- Tresca
- von Mises
- Parabolic criterion
- Oriented criterion
- Hill
- Hill modified
- HiSS model (Desai)
- Prager (for non-linear kinematic hardening)
- S-Clay 1
- Drucker Prager (non-linear kinematic hardening)
- Hill - Lourenço
- Hill - Lourenço (variant)
- Hiss model (variant)

Constitutive laws for the element families used in Coupled analysis.

Family	Constitutive model
Family 41	<p><i>Mechanics:</i></p> <ul style="list-style-type: none"> - linear isotropic elasticity - linear orthotropic elasticity <p><i>Hydraulics:</i></p> <ul style="list-style-type: none"> - anisotropic behaviour
Family 42	<p><i>Mechanics:</i></p> <ul style="list-style-type: none"> - linear isotropic elasticity <p><i>Hydraulics:</i></p> <ul style="list-style-type: none"> - anisotropic behaviour
Family 45	<p><i>Thermo-poro-elasticity:</i></p> <ul style="list-style-type: none"> - isotropic linear - orthotropic linear - with isotropic dilatancy <p><i>Thermo-poro-elastoplasticity, criterion:</i></p> <ul style="list-style-type: none"> - Mohr-Coulomb - Tresca - Von Mises (with or without strain hardening) - Drücker-Prager (with or without strain hardening) - parabolic - Vermeer - Nova - modified Cam-Clay - Prevost and Hoëg - oriented
Family 46	<p><i>Thermo-poro-elasticity:</i></p> <ul style="list-style-type: none"> - isotropic linear - orthotropic linear - with isotropic dilatancy <p><i>Thermo-poro-elastoplasticity, criterion:</i></p> <ul style="list-style-type: none"> - Von Mises (with or without strain hardening) - Drücker-Prager (with or without strain hardening) - parabolic - Vermeer - Nova - modified Cam-Clay - Prevost and Hoëg - oriented

A3. Identification of scalar results

Volume bodies

Displacements

u_x	Horizontal displacement, direction Ox
u_y	Horizontal displacement, direction Oy
u_z	Vertical displacement, direction Oz
θ_x	Rotation around Ox-axis
θ_y	Rotation around Oy-axis
θ_z	Rotation around Oz-axis
$ u $	Total displacement

Strains

ϵ_{xx}	Strain
ϵ_{yy}	Strain
ϵ_{zz}	Strain
ϵ_{xy}	Strain
ϵ_{yz}	Strain
ϵ_{zx}	Strain
ϵ_1	Principal strain
ϵ_2	Principal strain
ϵ_3	Principal strain
$\epsilon_{xx,p}$	Plastic strain
$\epsilon_{yy,p}$	Plastic strain
$\epsilon_{zz,p}$	Plastic strain
$\epsilon_{xy,p}$	Plastic strain
$\epsilon_{yz,p}$	Plastic strain
$\epsilon_{zx,p}$	Plastic strain
$\epsilon_{1,p}$	Principal plastic strain
$\epsilon_{2,p}$	Principal plastic strain
$\epsilon_{3,p}$	Principal plastic strain
$\epsilon_{xx,t}$	Total strain
$\epsilon_{yy,t}$	Total strain
$\epsilon_{zz,t}$	Total strain
$\epsilon_{xy,t}$	Total strain
$\epsilon_{yz,t}$	Total strain
$\epsilon_{zx,t}$	Total strain

$\varepsilon_{1,t}$	Principal total strain
$\varepsilon_{2,t}$	Principal total strain
$\varepsilon_{3,t}$	Principal total strain
$ \varepsilon_p $	Norm of plastic strain

Stresses

σ_{xx}	Stress
σ_{yy}	Stress
σ_{zz}	Stress
σ_{xy}	Stress
σ_{yz}	Stress
σ_{zx}	Stress
σ_1	Principal stress
σ_2	Principal stress
σ_3	Principal stress
$ \sigma_1 $	Principal stress (absolute value)
$ \sigma_2 $	Principal stress (absolute value)
$ \sigma_3 $	Principal stress (absolute value)
σ_s	Shear stress
σ_t	Traction
σ_c	Compression

Stresses in reinforced bodies

$\sigma_{xx,m}$	Stress in the matrix
$\sigma_{xx,r1}$	Stress in reinforcement #1
$\sigma_{xx,r2}$	Stress in reinforcement #2
$\sigma_{xx,sum}$	Sum of stresses in both reinforcements
$\sigma_{yy,m}$	Stress in the matrix
$\sigma_{yy,r1}$	Stress in reinforcement #1
$\sigma_{yy,r2}$	Contrainte dans renforcement 2
$\sigma_{yy,sum}$	Sum of stresses in both reinforcements
$\sigma_{zz,m}$	Stress in the matrix
$\sigma_{zz,r1}$	Stress in reinforcement #1
$\sigma_{zz,r2}$	Contrainte dans renforcement 2
$\sigma_{zz,sum}$	Sum of stresses in both reinforcements
$\sigma_{xy,m}$	Stress in the matrix
$\sigma_{xy,r1}$	Stress in reinforcement #1
$\sigma_{xy,r2}$	Contrainte dans renforcement 2
$\sigma_{xy,sum}$	Sum of stresses in both reinforcements
$\sigma_{yz,m}$	Stress in the matrix
$\sigma_{yz,r1}$	Stress in reinforcement #1
$\sigma_{yz,r2}$	Contrainte dans renforcement 2
$\sigma_{yz,sum}$	Sum of stresses in both reinforcements
$\sigma_{zx,m}$	Stress in the matrix

$\sigma_{zx,r1}$	Stress in reinforcement #1
$\sigma_{zx,r2}$	Contrainte dans renforcement 2
$\sigma_{zx,sum}$	Sum of stresses in both reinforcements
$\sigma_{1,m}$	Principal stress in the matrix
$\sigma_{1,r1}$	Principal stress in reinforcement #1
$\sigma_{1,r2}$	Principal stress in reinforcement #2
$\sigma_{1,sum}$	Sum of principal stresses in both reinforcements
$\sigma_{2,m}$	Principal stress in the matrix
$\sigma_{2,r1}$	Principal stress in reinforcement #1
$\sigma_{2,r2}$	Principal stress in reinforcement #2
$\sigma_{2,sum}$	Sum of principal stresses in both reinforcements
$\sigma_{3,m}$	Principal stress in the matrix
$\sigma_{3,r1}$	Principal stress in reinforcement #1
$\sigma_{3,r2}$	Principal stress in reinforcement #2
$\sigma_{3,sum}$	Sum of principal stresses in both reinforcements

Criteria

σ_{tre}	Tresca
σ_{vm}	Von Mises
σ_{dp}	Drucker Prager
σ_{sm}	Maximal shear stress
σ_{com}	Stress combination
σ_{cp}	Parabolic criterion
σ_{cg}	General criterion
p	Mean stress
p	Deviatoric stress
σ_{rup}	Failure stress

Hydrogeology

H	Hydraulic head
P	Pressure
w	Water content
ΔH_x	Gradient of hydraulic head, x
ΔH_y	Gradient of hydraulic head, y
ΔH_z	Gradient of hydraulic head, z
ΔP_x	Gradient of pressure, x
ΔP_y	Gradient of pressure, y
ΔP_z	Gradient of pressure, z
vh_x	Fluid velocity, x
vh_y	Fluid velocity, y
vh_z	Fluid velocity, z
vp_x	Fluid velocity, x
vp_y	Fluid velocity, y
vp_z	Fluid velocity, z

Temperature

T°	Temperature
ΔT°_x	Gradient of temperature, x
ΔT°_y	Gradient of temperature, y
ΔT°_z	Gradient of temperature, z
ξ	Hydration degree
vT°_x	Temperature velocity, x
vT°_y	Temperature velocity, y
vT°_z	Temperature velocity, z

1D bodies (bars, beams)

N	Normal force
Vy	Shear force
Vz	Shear force
Mt	Torsion moment
My	Bending moment
Mz	Bending moment
δf	Relative displacement (friction bar)

Shells

Stresses

$\sigma_{_1}$	Principal stress in shell
$\sigma_{_2}$	Principal stress in shell
$\sigma_{_3}$	Principal stress in shell
$ \sigma_{_1} $	Principal stress (absolute value)
$ \sigma_{_2} $	Principal stress (absolute value)
$ \sigma_{_3} $	Principal stress (absolute value)
$\sigma_{_1,sup}$	Principal stress, upper face
$\sigma_{_1,inf}$	Principal stress, lower face
$\sigma_{_2,sup}$	Principal stress, upper face
$\sigma_{_2,inf}$	Principal stress, lower face
$\sigma_{_3,sup}$	Principal stress, upper face
$\sigma_{_3,inf}$	Principal stress, lower face
σ_{xx}	Stress in shell
σ_{yy}	Stress in shell
σ_{zz}	Stress in shell
σ_{xy}	Stress in shell
σ_{yz}	Stress in shell
σ_{zx}	Stress in shell
$\sigma_{xx,sup}$	Stress in shell, upper face
$\sigma_{yy,sup}$	Stress in shell, upper face
$\sigma_{zz,sup}$	Stress in shell, upper face
$\sigma_{xy,sup}$	Stress in shell, upper face

$\sigma_{yz,sup}$	Stress in shell, upper face
$\sigma_{zx,sup}$	Stress in shell, upper face
$\sigma_{xx,inf}$	Stress in shell, lower face
$\sigma_{yy,inf}$	Stress in shell, lower face
$\sigma_{zz,inf}$	Stress in shell, lower face
$\sigma_{xy,inf}$	Stress in shell, lower face
$\sigma_{yz,inf}$	Stress in shell, lower face
$\sigma_{zx,inf}$	Stress in shell, lower face
σ_{vm}	von Mises criterion in shell

Forces

N_{xx}	Force in shell
N_{yy}	Force in shell
N_{zz}	Force in shell
N_{xy}	Force in shell
N_{yz}	Force in shell
N_{zx}	Force in shell
N_1	Principal force in shell
N_2	Principal force in shell
$ N_1 $	Principal force in shell (absolute value)
$ N_2 $	Principal force in shell (absolute value)
M_{xx}	Moment in shell
M_{yy}	Moment in shell
M_{zz}	Moment in shell
M_{xy}	Moment in shell
M_{yz}	Moment in shell
M_{zx}	Moment in shell
M_1	Principal moment in shell
M_2	Principal moment in shell
$ M_1 $	Principal moment in shell (absolute value)
$ M_2 $	Principal moment in shell (absolute value)

Vectors

Mechanics

3D displacements
 3D rotations
 3D reaction forces
 3D reaction moments

Hydrogeology

Pressure gradient 3D
 Water head gradient 3D
 Fluid velocity 3D
 Fluid velocity 3D

Temperature

Temperature gradient 3D

Temperature velocity 3D

Tensors

Stresses

Stresses 3D

Stresses 3D, matrix

Stresses 3D, reinforcement 1

Stresses 3D, reinforcement 2

Stresses 3D, sum of 2 reinforcements

Strains

Strains 3D

Plastic strains 3D

Total Strains 3D

Forces

Forces 3D

Shell stresses

Shell forces

Shell moments

Site web : www.cesar-lcpc.com
Contact : cesar-lcpc@itech-soft.com
Tél : +33 (0)1 49 76 12 59



8 quai de Bir Hakeim
F-94410 Saint-Maurice
www.itech-soft.com

14-20 Boulevard Newton
Cité Descartes, Champs sur Marne
F-77447 Marne la Vallée Cedex 2
www.ifsttar.fr