CESAR LCPC

General overview

Version 2019

IFSTTAR

itech

Table of content

1. Introduction

1.1. About CESAR

CESAR-LCPC is an all-purpose software package based on the finite element method and adapted to resolving civil engineering and environmental problems: soil and rock mechanics, heat-related problems, hydrogeology, structural calculations, etc.

From the beginning of its development, the CESAR solver includes a wide variety of constitutive models and an extensive element library, results of the research effort of LCPC's teams.

Since version 5 of CESAR-LCPC, user benefits from a highly interactive graphical user interface and enriched meshing capabilities (tetrahedron meshes).

Civil engineering-based models assist the engineer in better understanding and mastering complex phenomena (explanatory models), as well as in designing and sizing structures (predictive decision-making models). Such models rely upon: an analysis of the phenomena under examination and their physical-mathematical representation, the numerical resolution of the corresponding set of equations, and comparisons with experimental results.

The main applications for CESAR-LCPC in geotechnical engineering are:

- Design of underground structures: tunnels, mines, storage facilities, deep excavations…
- Design of embankments, shallow and deep foundations (settlements and soil failure analysis),
- Slope stability analysis: cut or fill for highways, reinforced slopes, earth retaining structures…
- Seepage, dewatering and consolidation analyses.

The main applications for CESAR-LCPC in structural engineering are:

- Massive concrete structures (phenomena associated with young hardening concrete),
- Ancient buildings with masonry structures.
- Design of pavement structures for airport, tramway or other transport facilities.

1.2. History

The IFFSTAR, formerly known as the LCPC (Laboratoire Central des Ponts et Chaussées / Road and Public Works Research Institute), civil engineering research facility, has been involved with these model-building efforts since the end of the 1960's and over the past thirty years has been focusing on the development of its CESAR-LCPC software package.

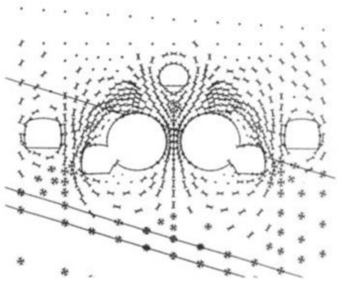
Development of the CESAR-LCPC product got underway towards the beginning of the 1980's as a definitive successor to the ROSALIE system (developed and revised by the LCPC between 1968 and 1983); Version 2.0 was initially released in 1986. CESAR-LCPC has been designed for continual revision and updating by virtue of its status as a research and development tool.

1.3. CESAR models through versions

We present below few models as illustrations of the version evolutions of CESAR from its early age until now. We focus here on **geotechnical applications** (foundations, excavation and tunnels).

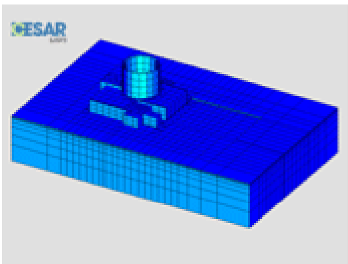
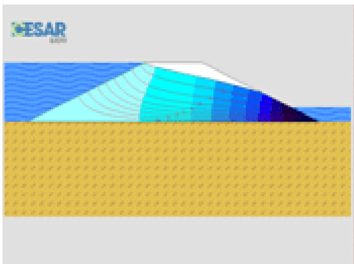
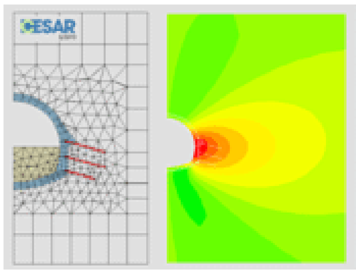
More references on other modelling capacities of CESAR could be found for illustration of analyses of structures of Civil Engineering as bridges, ancient buildings or other expert projects.

Version 2.x (1986 > 1992)

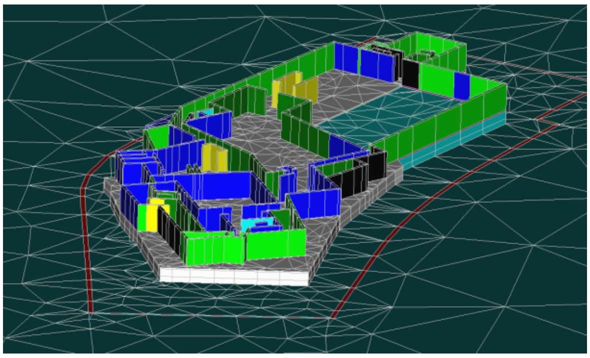
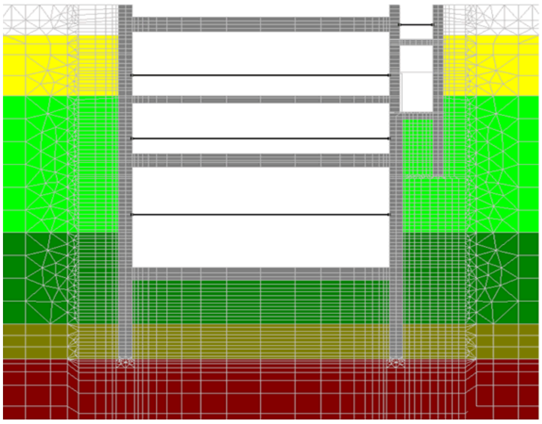
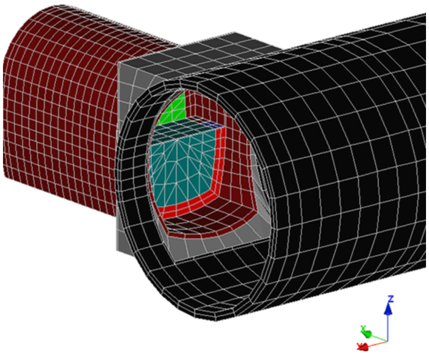
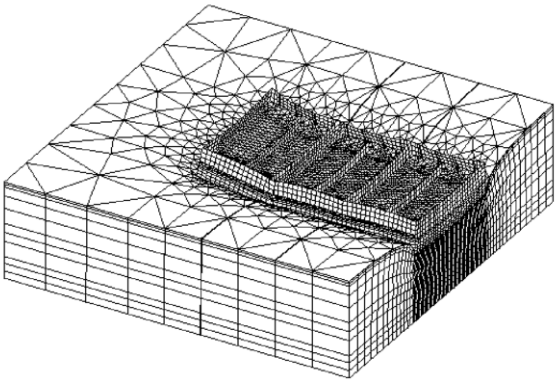


Channel Tunnel analysis in the 90's

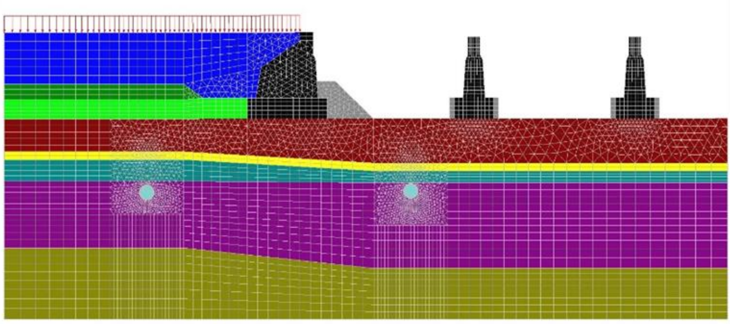
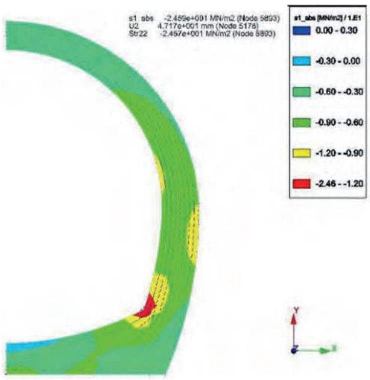
Version 3.x (1992 > 2003)

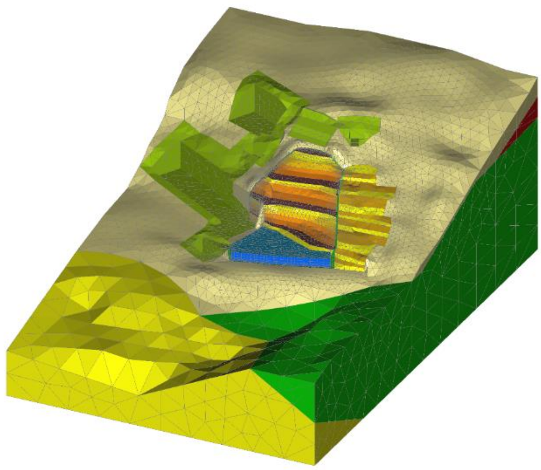
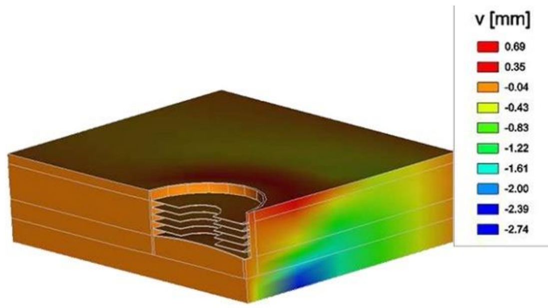
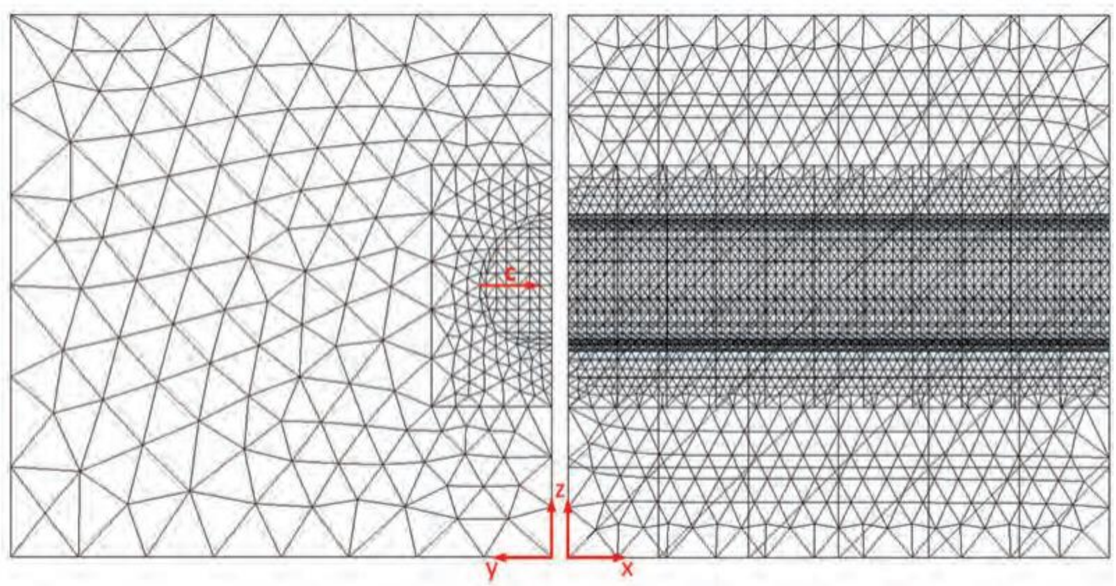
		
Nuclear plant	Dike	Tunnel

Version 4.x (2003 > 2009)

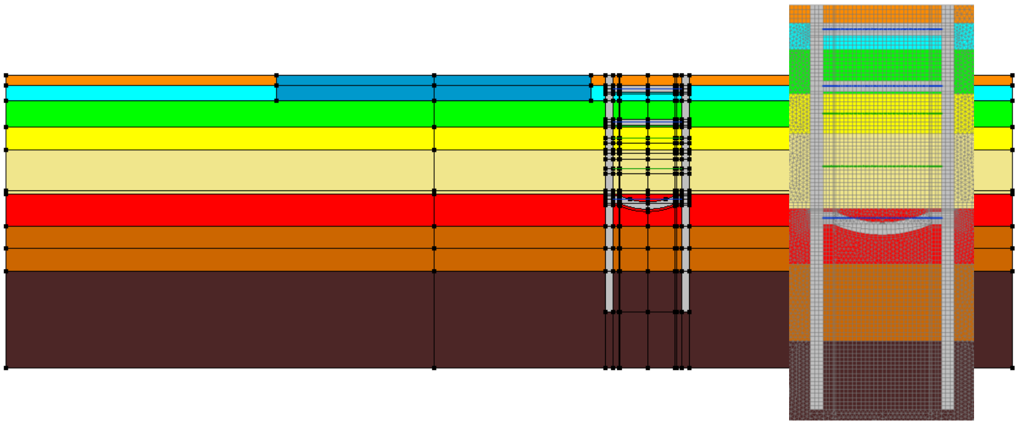
	
Foundation of Louis Vuitton building (Terrasol)	Deep excavation (Arcadis)
	
Tunnel intersection (Bouygues TP)	Water treatment plant (Terrasol)

Version 5.x (2009 > 2014)

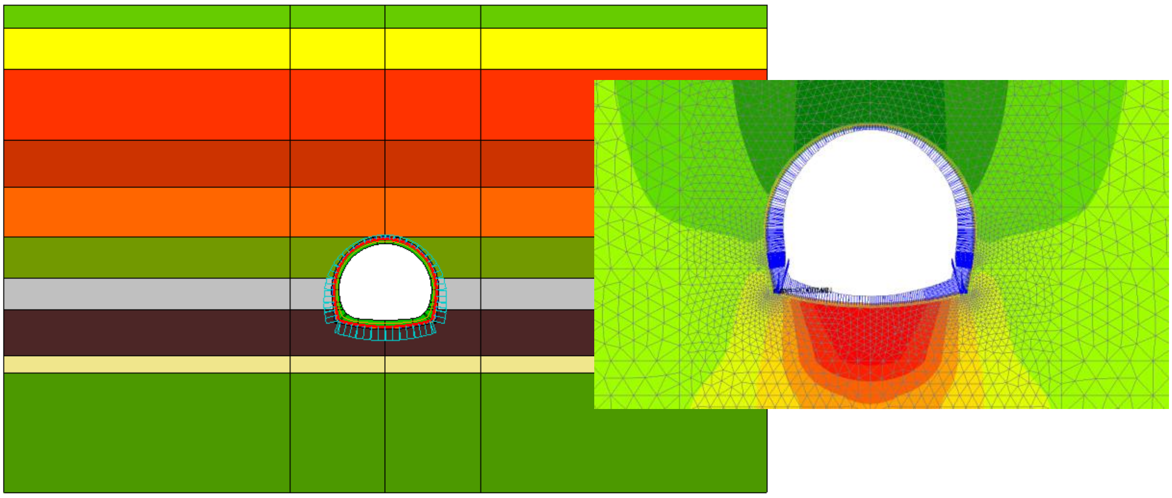
	
Excavations below existing bridge (Vinci)	Tunnel (Canal de Provence)

<table>
<tr><td></td><td>
</td></tr>
<tr><td>Odeon tower foundations (Terrasol)</td><td>Excavation (LCPC for Soletanche-Bachy)</td></tr>
<tr><td colspan="2">
</td></tr>
<tr><td colspan="2">3D Tunnel (Egis)</td></tr>
</table>

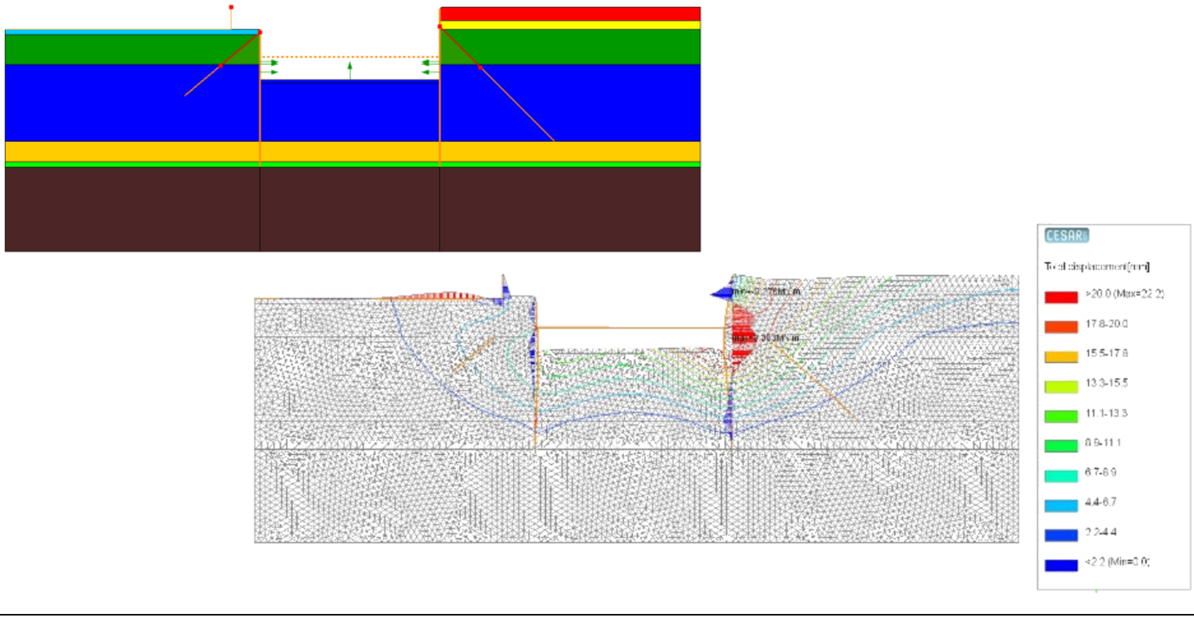
Version 6.x (2014 > 2019)



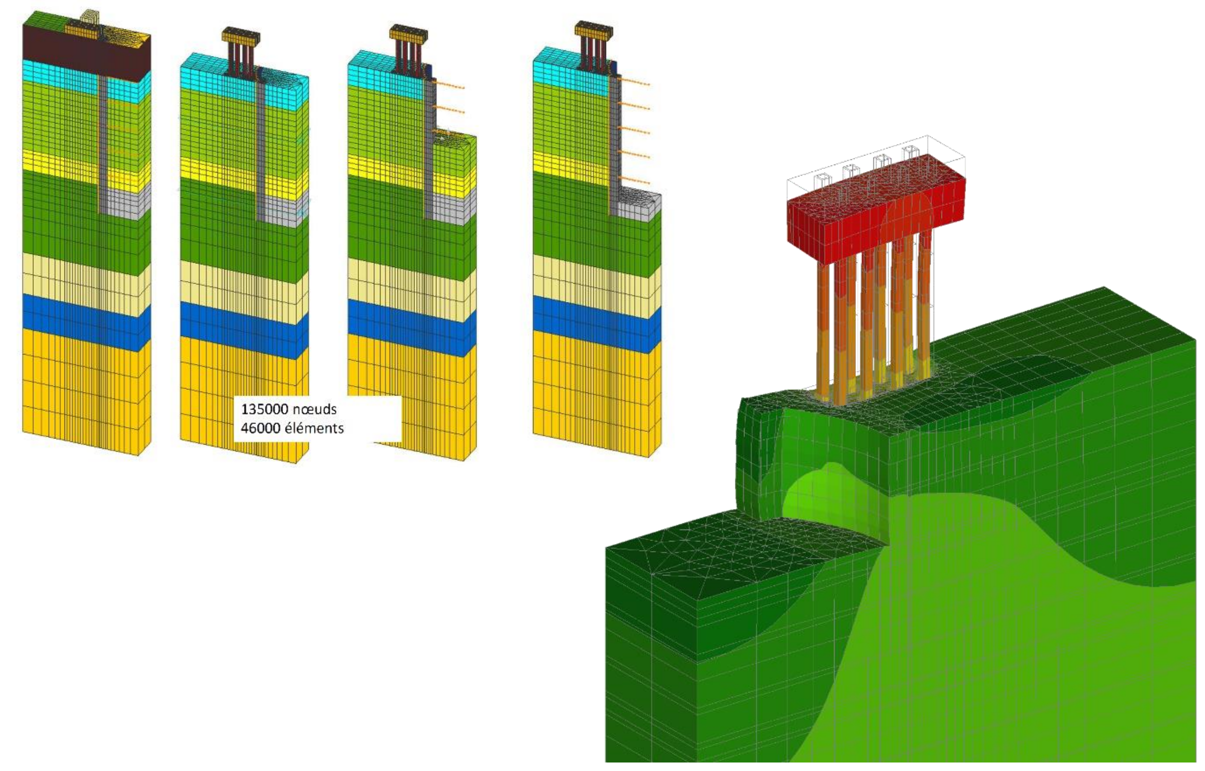
Excavation for a subway station (Setec)



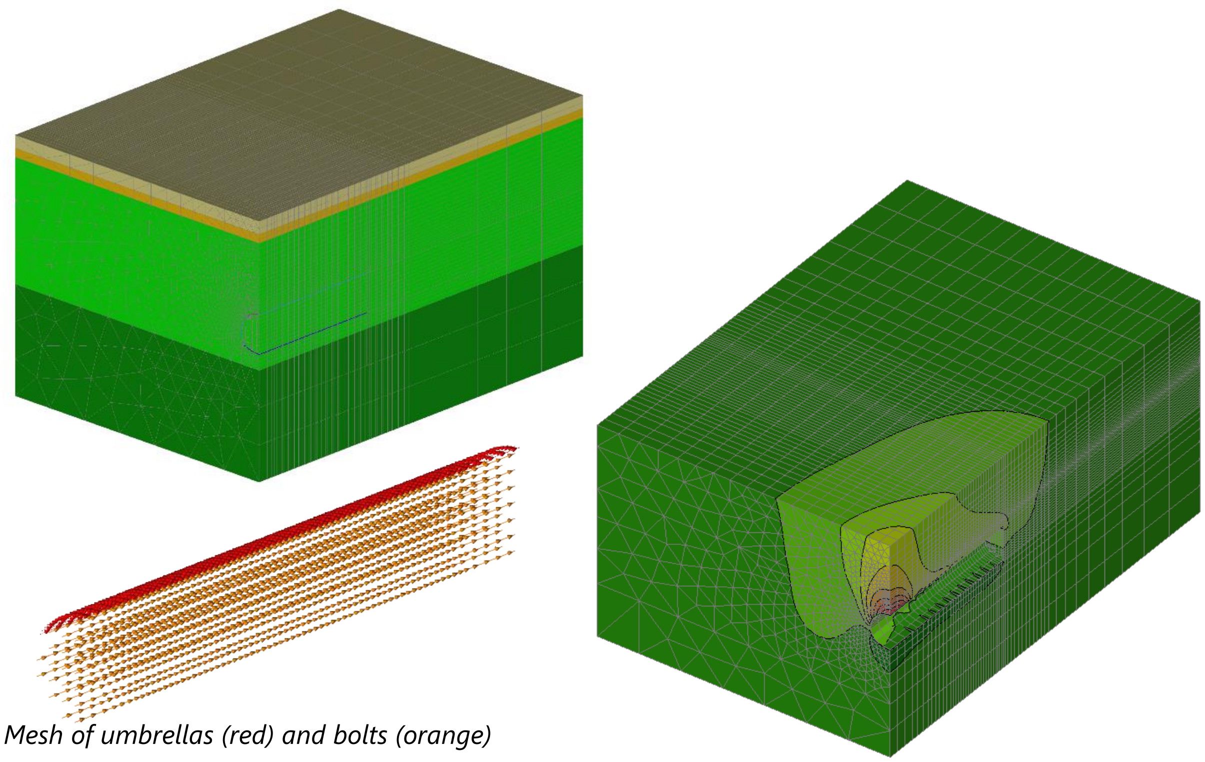
Subway section analysis (Vinci)



Excavation for building foundations (Centrale Nantes)

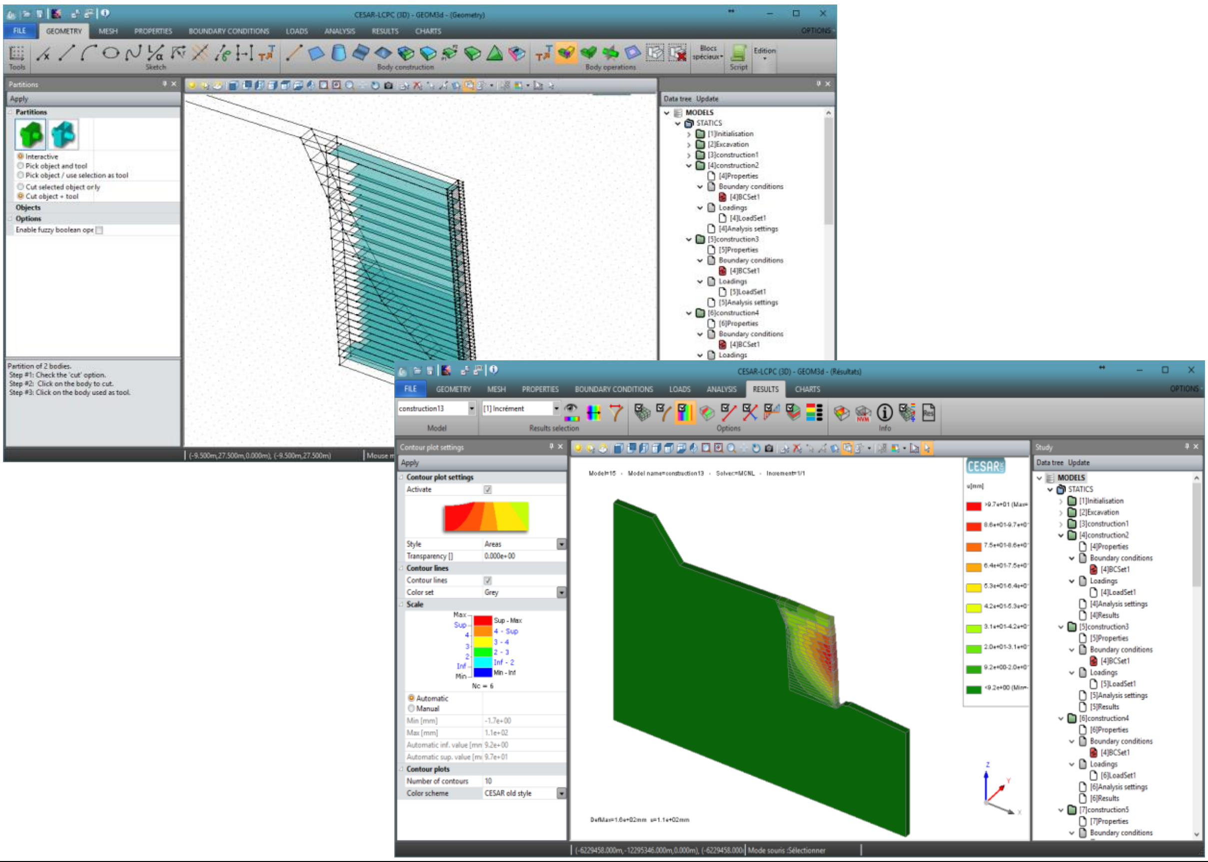


3D modelling of temporary supports for the project of the station *La Défense* – Line 15 of the Grand Paris Express (Société du Grand Paris)

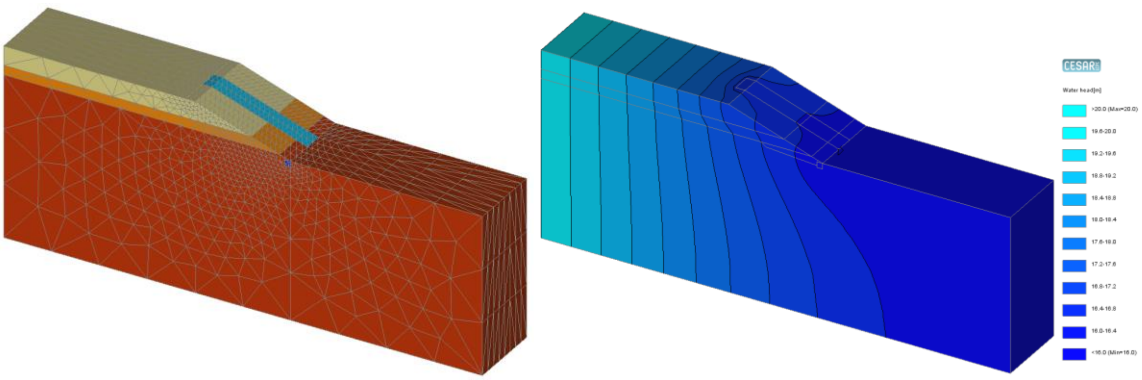


Mesh of umbrellas (red) and bolts (orange)

Tunnel 3D with umbrella and front reinforcement (Newtun consortium)



MSE wall (Cerema)

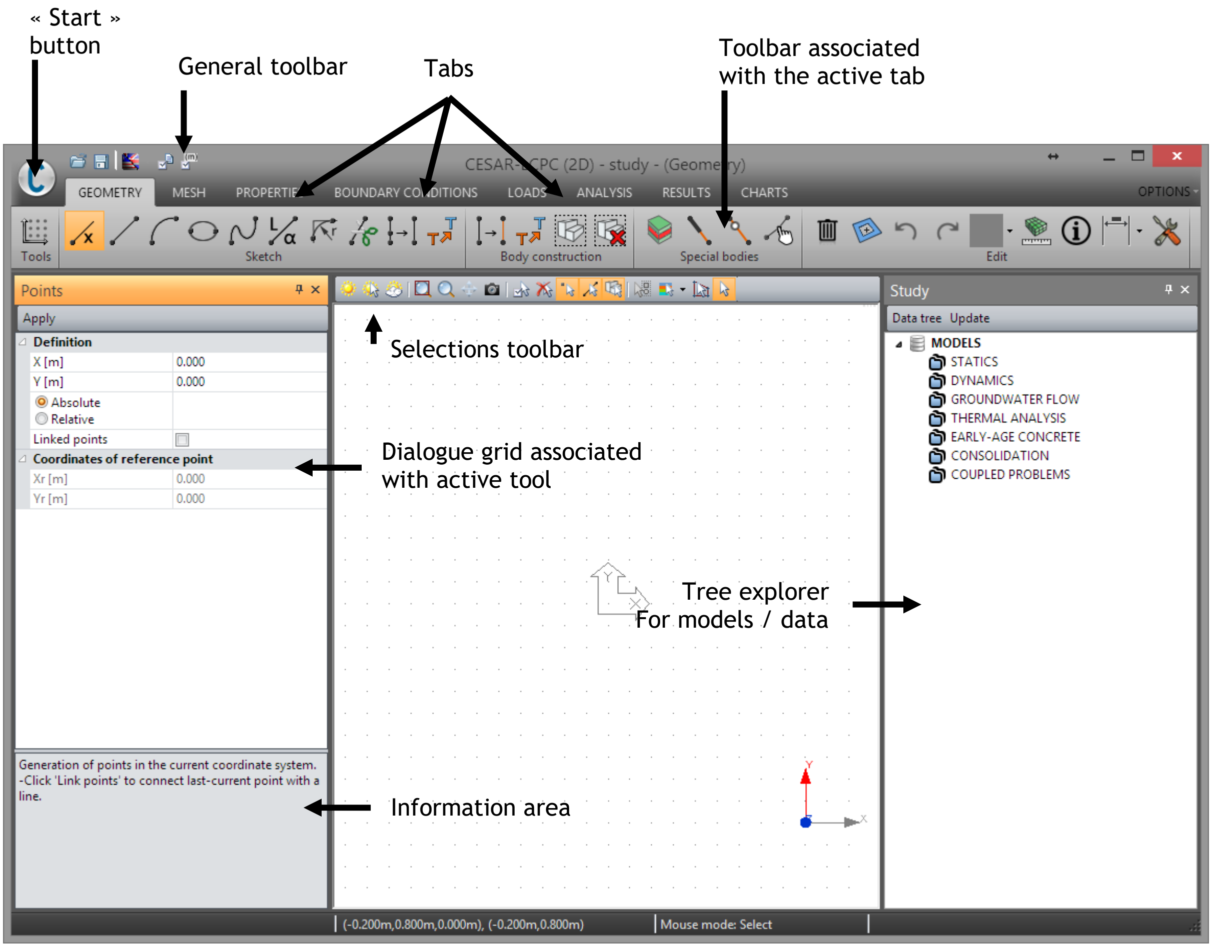


Influence of gravel inserts on water flow in an embankment slope (Guintoli)

2. Graphical User Interface

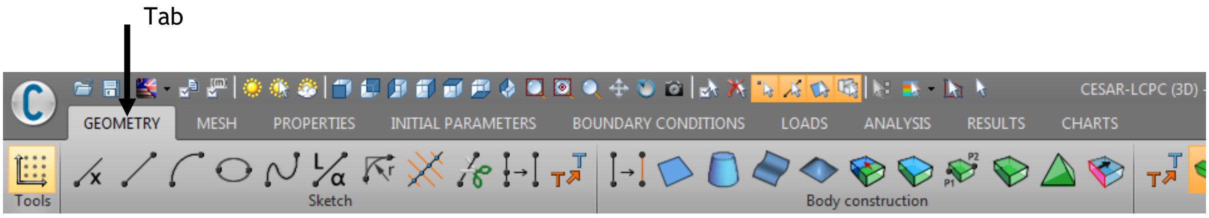
2.1. General overview

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



2.1.1. 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).

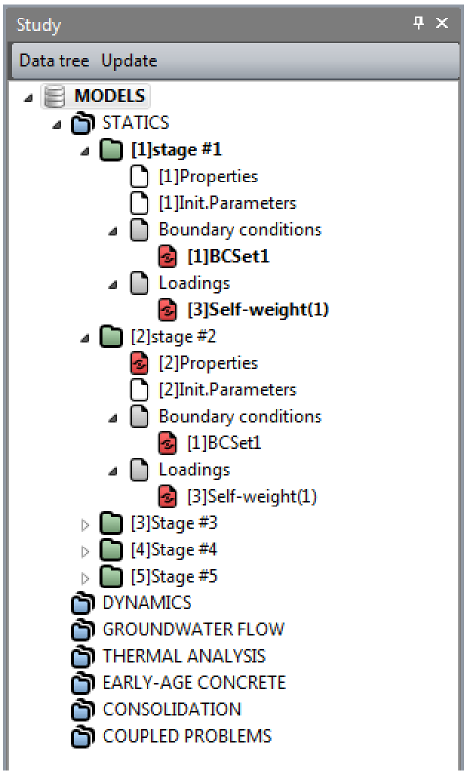
As a default, CESAR 3D proposes the following tabs:

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

2.1.2. 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.

The diagram beside lists 5 Mechanics models (stages #1 to #5).

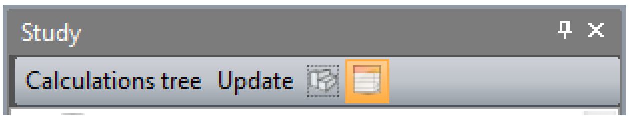


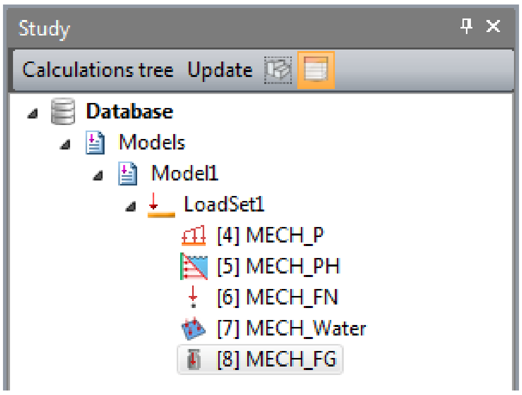
2.1.3. Data tree diagram

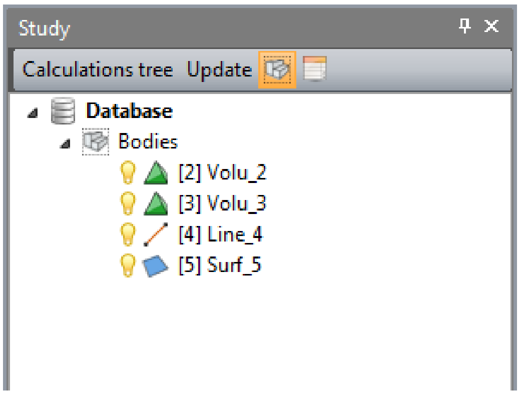
A single click shifts from the model tree diagram to the data tree diagram.

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:

- A uniform pressure type of mechanical loading (MECH_P),
- A hydrostatic pressure type of mechanical loading (MECH_PH),
- A gravity force type of mechanical loading (MECH_FG).
- -...

Example 2: List of the model's constitutive bodies:

- 2 volume bodies,
- 1 one-dimensional body,
- 1 surface body.

2.2. CAD tools

Using CESAR-LCPC, user will access advanced geometry modelling features. These are necessary to achieve 2D as well as 3D complex analysis.

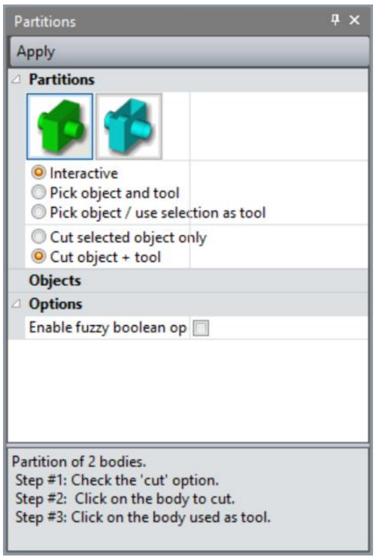
Data exchange is also possible with other CAD software (dxf or LandXML imports for example).

Standard tools enable the user to easily define and edit the structure's geometry.

- Definition of lines, circles, ellipses, splines, predefined structures,
- Definition of surfaces: planes, cylinders, Coons patches ...,
- Translation, rotation and symmetry operations,
- Intersection of lines, surfaces...

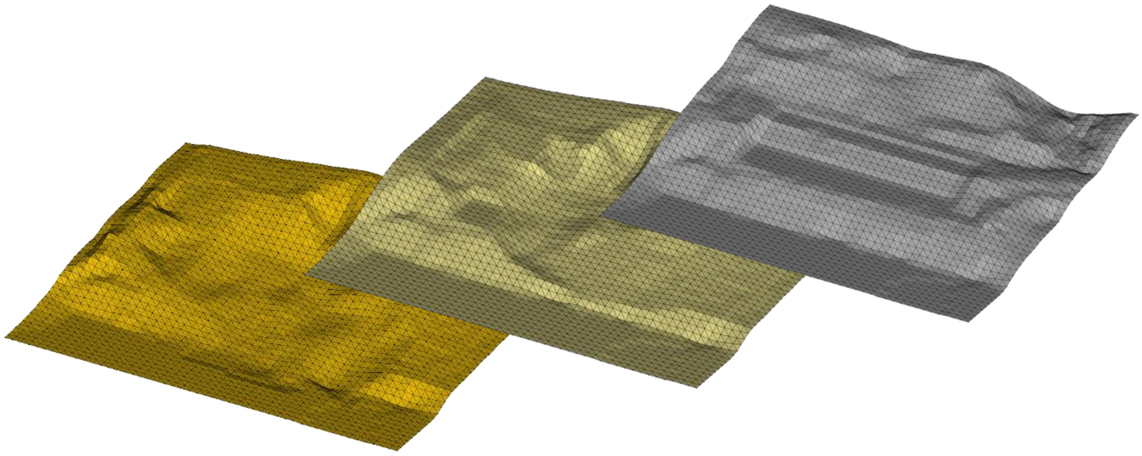
Intersections

- Volume/volume
- Volume/surface
- Surface/surface
- Surface/line
- Line/line
- Line/surface



LandXML import

With development of 3D models and investigations of landslides, accurate representation of the site geometry is becoming important. The LandXML format is helpful to import this relevant information.



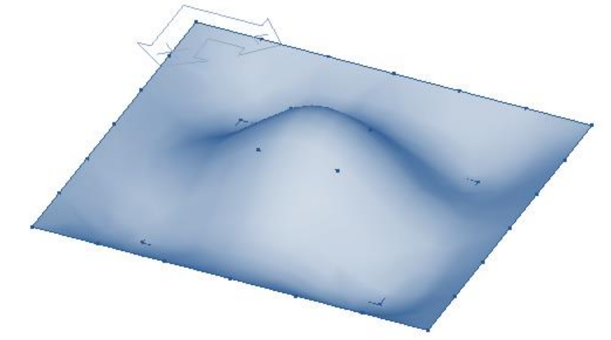
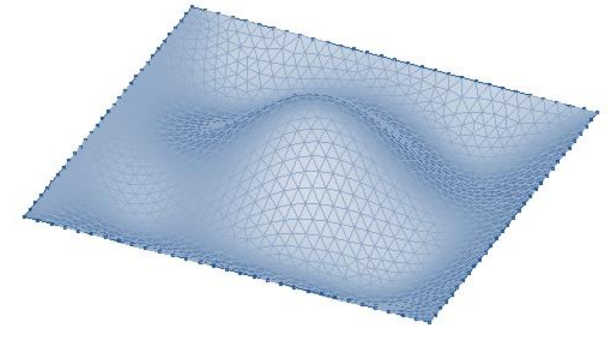
Example of imported LandXML files (3 surfaces)

B-Spline Surfaces

Non-uniform rational basis spline is a mathematical model commonly used in computer graphics for generating and representing curves and surfaces which offers great flexibility and precision for handling both analytic and freeform shapes.

In CESAR-LCPC, NURBS are controlled by an external boundary and inner points.

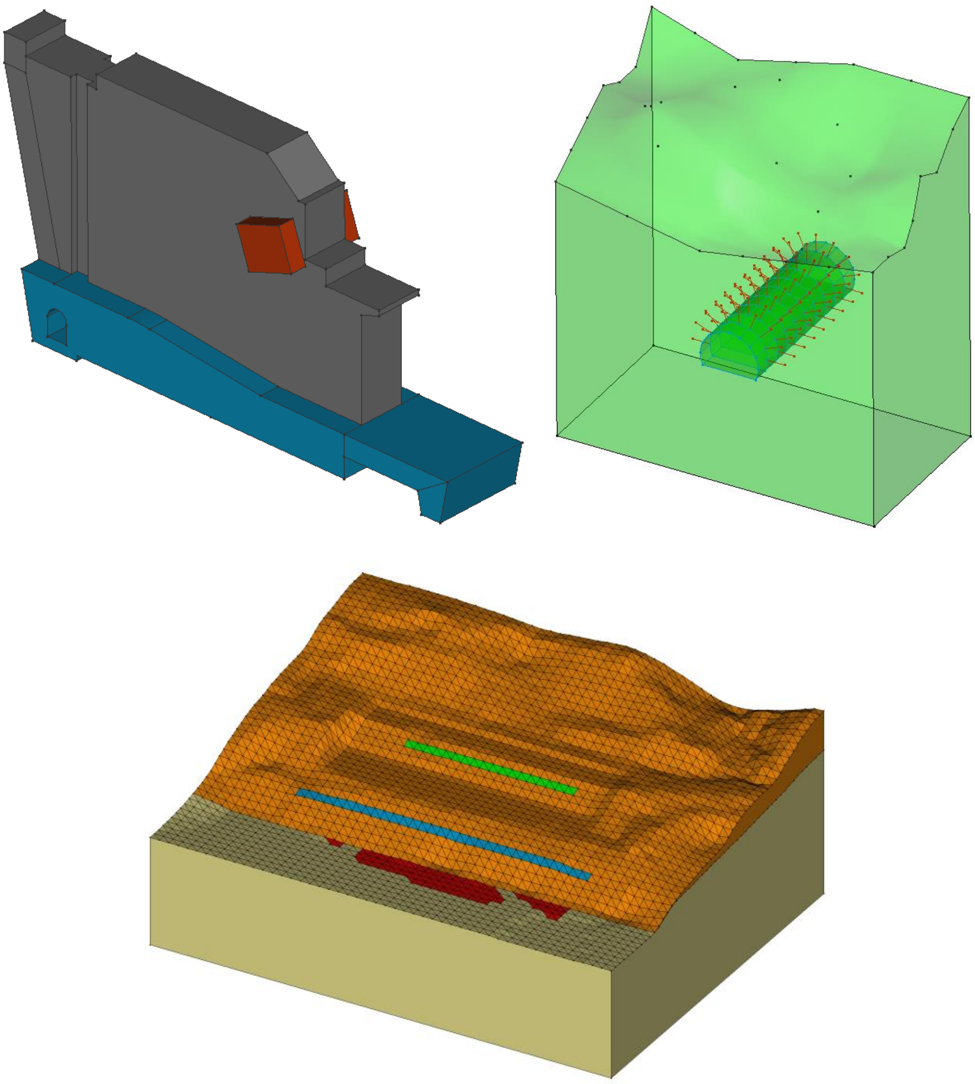
Using these NURBS is useful for modelling of complex geotechnical or structural problems.

	
B-spline surface region	*Example of the associated mesh*

Volumes of any shape

With the same objectives of the NURBS, volumes of any shapes are fundamental for complex geotechnical or structural models.

CESAR-LCPC enables the generation of volumes closed by surfaces of any type.



Example of volumes

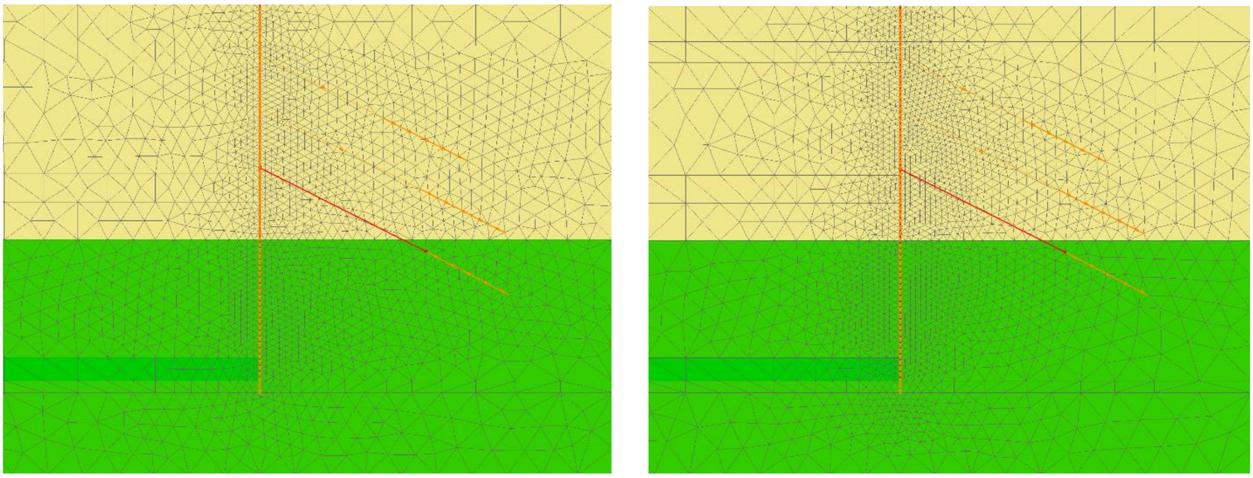
2.3. Meshing features

2.3.1. Surface meshing

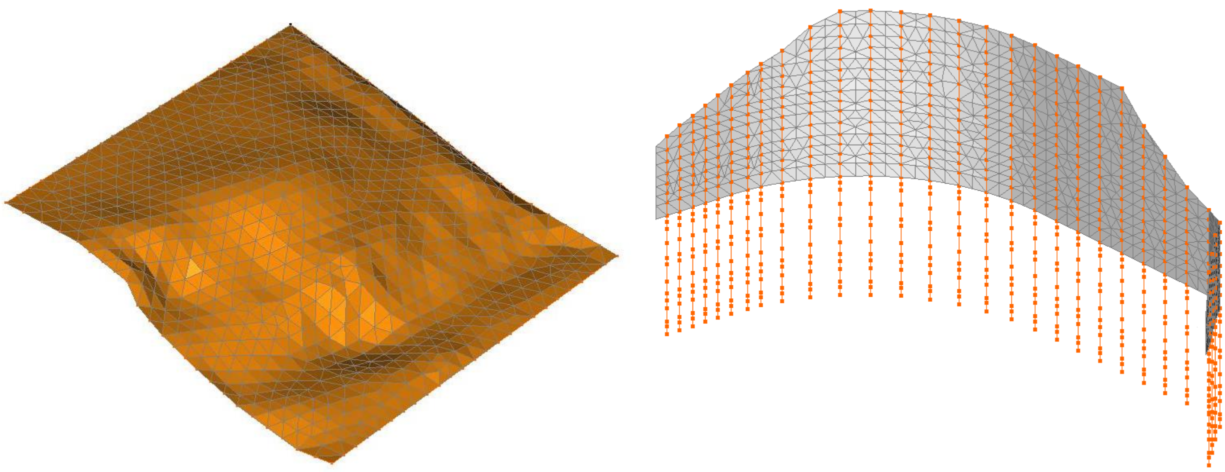
Surface meshing is the common procedure for 2D models as well for shell elements generation. Surface meshes are quadrangular or triangular, depending of the geometry of the supporting surface.

CESAR-LCPC proposes 3 levels of filling surfaces using mesh algorithms: linear, quadratic or cubic. These levels are useful to generate transition meshes from refined areas to lose areas.

These algorithms are also used for surface mesh generation in 3D (shell and plates elements).



Example of 2D mesh evolution depending on the algorithm (from linear to cubic)



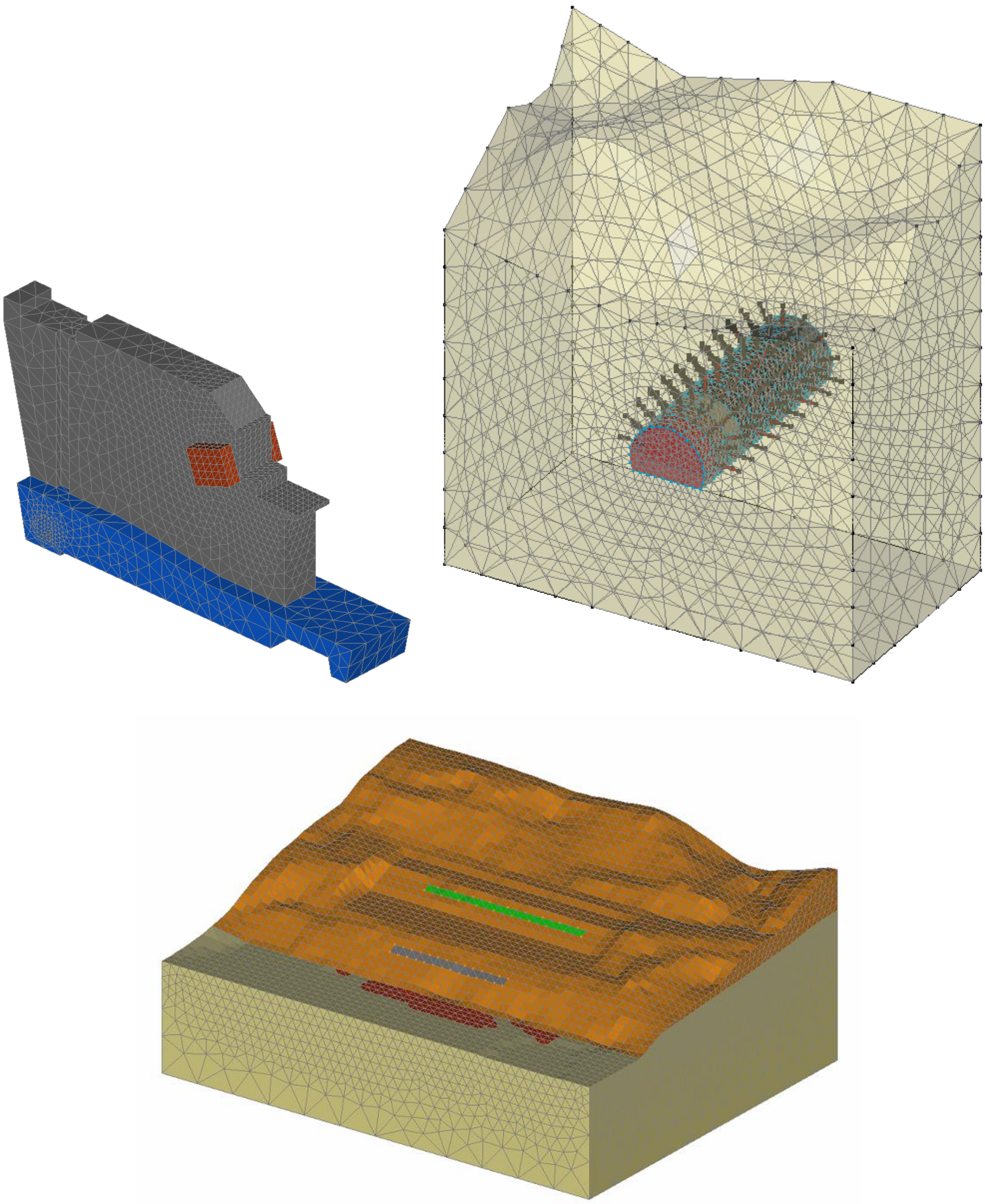
3D Surface meshes

2.3.2. 3D mesh - Tetrahedron mesh generator

In CESAR-LCPC, the user is free of geometrical constraints. This is made possible by using automatic tetrahedron mesh generators. Those fill the volume from its external surface mesh.

CESAR-LCPC offers 2 types of mesh generators:

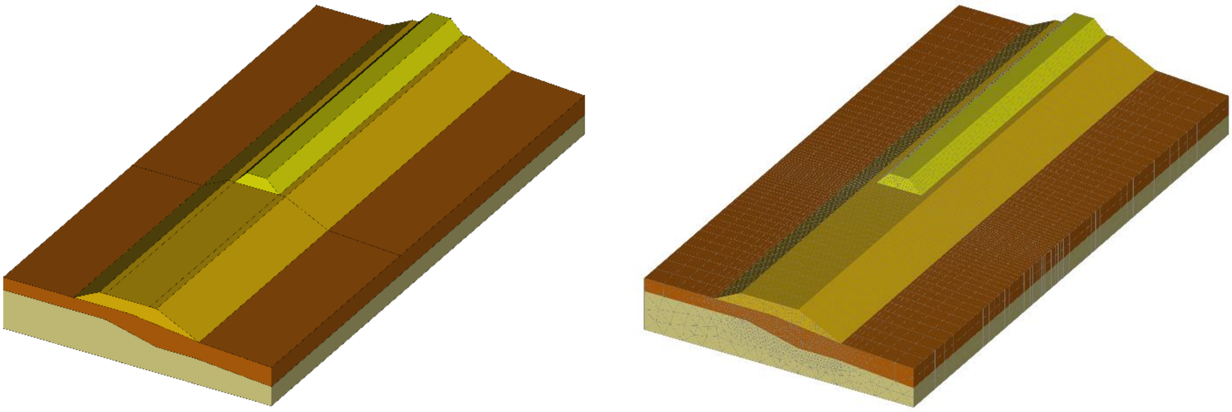
- An integrated one, with option to define a mesh coarseness;
- An external one, "TETMESH-GHS3D", developed by INRIA and SIMULOG.



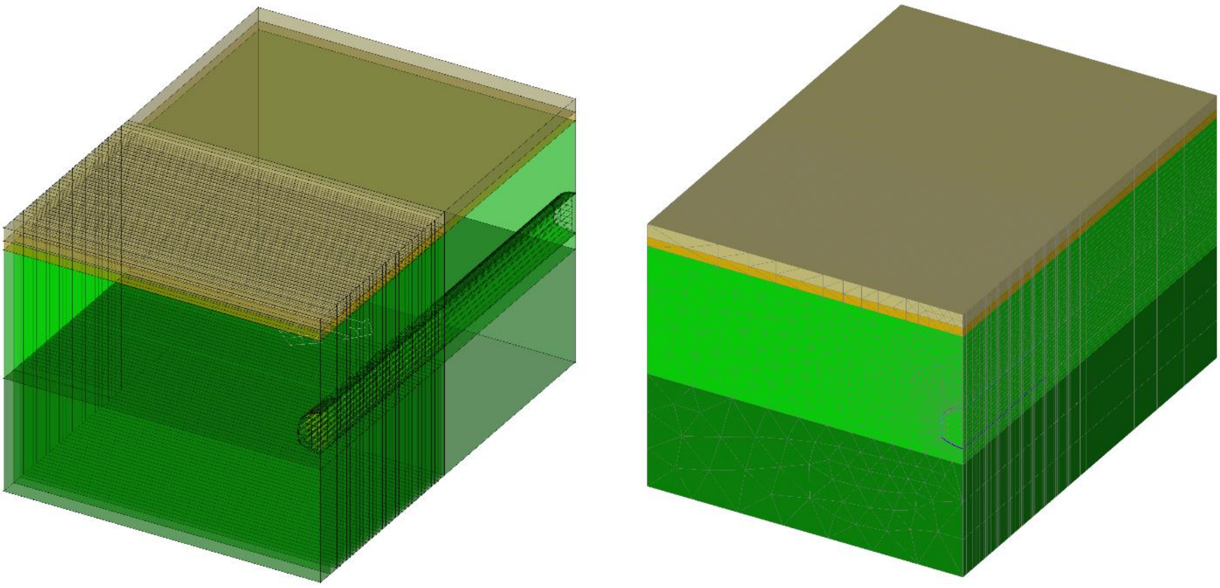
Example of 3D meshes

2.3.3. 3D mesh - Extrusion procedure

This procedure is useful when user designs models with a principal direction. In the GEOMETRY stage, a plane surface (horizontal or vertical) is designed and extruded. In the MESH stage, the user sets the density on this surface and all the extruded surfaces parallel to it. Then a density is set on all edges joining these parallel planes. This way, user forces the mesher to generate a regular mesh along the principal direction.



embankment – regular extruded geometry and the resulting 3D mesh



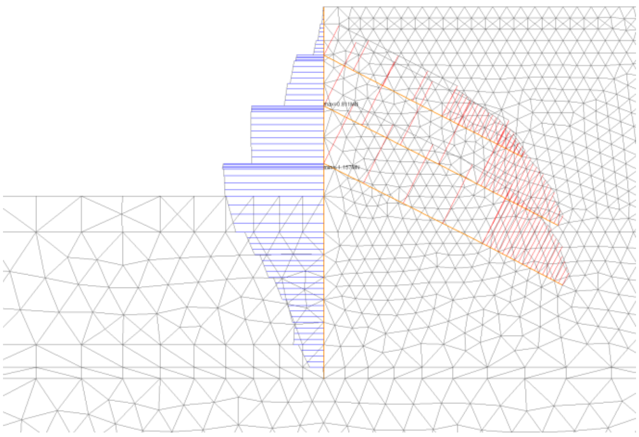
tunnel – regular extruded geometry and the resulting 3D mesh

2.4. Result analysis

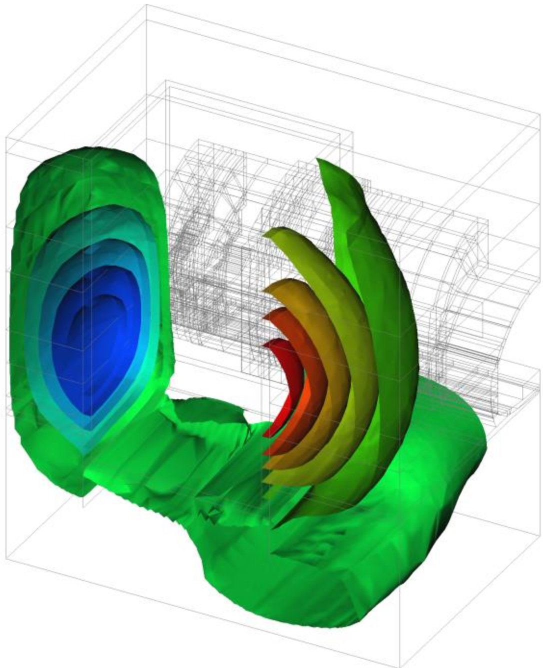
2.4.1. Result display

The visualization of results associated with the current model requires the following tools:

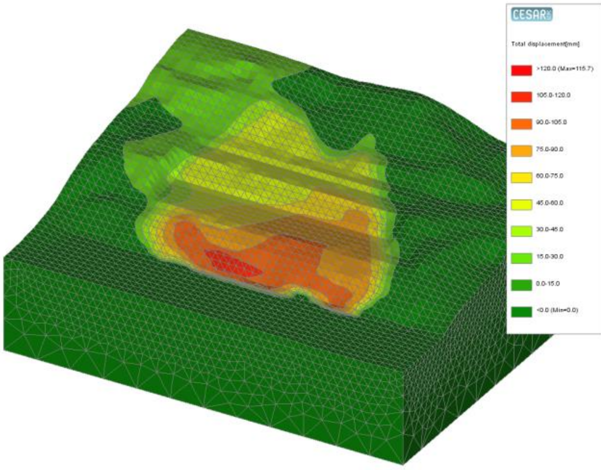
- Choice of type of results to be displayed
- Envelope of iso-surface scalar
- Animation or choice of "entity" (increment, time step, etc.) to be displayed
- Mesh settings
- Deformed mesh settings
- Contour plot settings
- Iso surfaces settings
- Vector settings
- Tensor settings
- Beam and truss result settings
- Contact state display
- Legend settings
- Cut plane
- Plane section analysis
- Information
- Results displayed by body



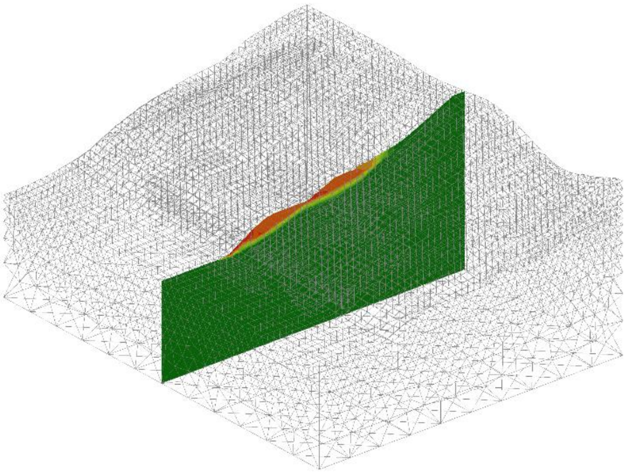
Forces in the structural element (truss and beams)



Iso-surfaces of displacement in a 3D modle of tunnel



Contour plots on a 3D slope model

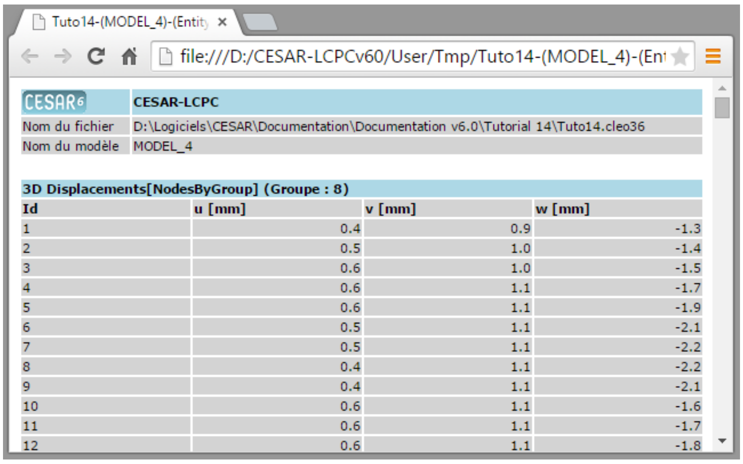


Cut plane in a 3D slope model

2.4.2. Result tables and charts

In addition to views of the results, user will access and analyse data through extensive tables and charts.

For every scalar result of the calculations, it is possible to access full tables per element or body. It is also easy to export these tables for analysis with external tools (MS Excel for example).

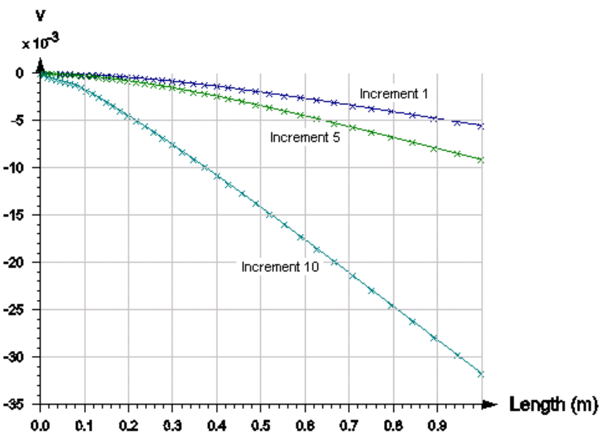


CESAR6	CESAR-LCPC
Nom du fichier	D:\Logiciels\CESAR\Documentation\Documentation v6.0\Tutorial 14\Tuto14.cleo36
Nom du modèle	MODEL_4

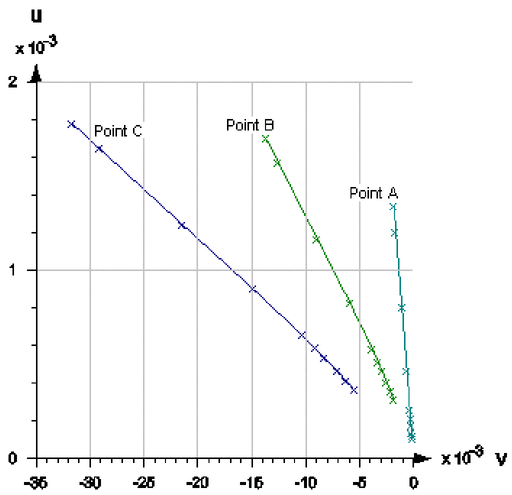
3D Displacements[NodesByGroup] (Groupe : 8)

Id	u [mm]	v [mm]	w [mm]
1	0.4	0.9	-1.3
2	0.5	1.0	-1.4
3	0.6	1.0	-1.5
4	0.6	1.1	-1.7
5	0.6	1.1	-1.9
6	0.5	1.1	-2.1
7	0.5	1.1	-2.2
8	0.4	1.1	-2.2
9	0.4	1.1	-2.1
10	0.6	1.1	-1.6
11	0.6	1.1	-1.7
12	0.6	1.1	-1.8

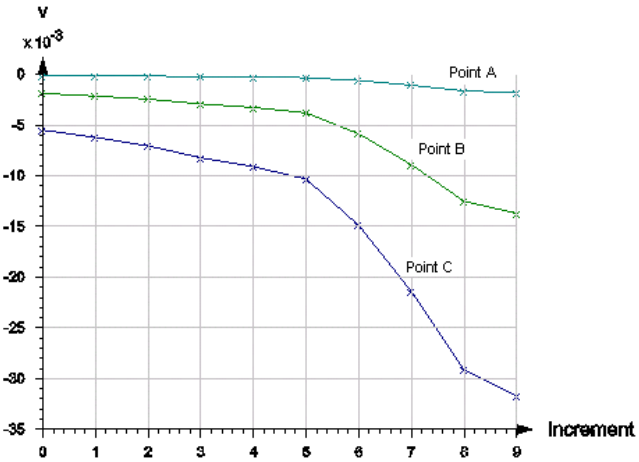
Three types of charts are available.



Charts for a line set



Charts for a set of points



Parametric charts

3. Analysis features

We present here the main components of the solver. User will refer to theoretical manual or reference manual to get a complete and detailed information.

3.1. 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 truss elements
Family 8	Three-dimensional truss elements
Family 11	Axisymmetrical isoparametric displacement-type elements for the computation of structures with a revolution-based geometry submitted to any loading

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)

Coupling

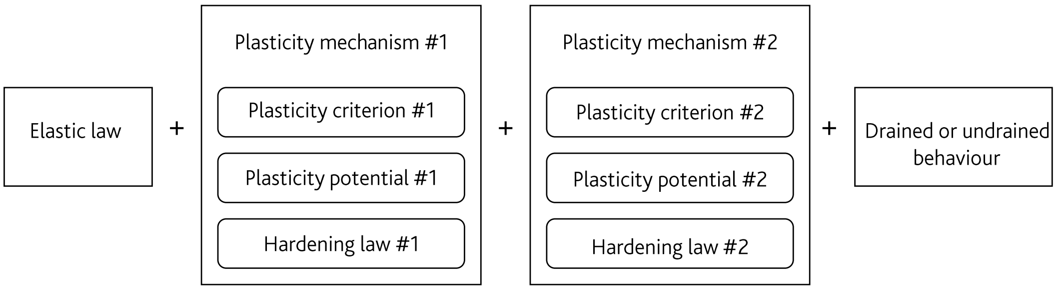
Family 41	Two-dimensional isoparametric elements with three degrees of freedom per node (two displacements, hydraulic load) for consolidation problems
Family 42	Three-dimensional isoparametric elements with four degrees of freedom per node (three displacements, hydraulic load) for consolidation problems
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

3.2. Constitutive models

3.2.1. Constitutive laws for the element families used in mechanics and geomechanics.

Family	Constitutive model	Type of material
Family 1	*Elasticity:*	
	– isotropic linear	
	– orthotropic linear	
	– with isotropic dilatancy	
	Elastoplasticity, criterion:	
	– 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
	Other:	
	early-age concrete	concrete
Family 2	*Elasticity:*	
	– isotropic linear	
	– orthotropic linear	
	– with isotropic dilatancy	
	Elastoplasticity, criterion:	
	– 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
	Other:	
	– early-age concrete	concrete
Families 3, 4, 5, 7, 8 and 11	*Linear isotropic elasticity*	
Family 6	*Contact laws:*	
	– bonding	
	– Coulomb friction	
	– perfect slippage	

3.2.2. Toolbox for user constitutive model definition



Elastic laws

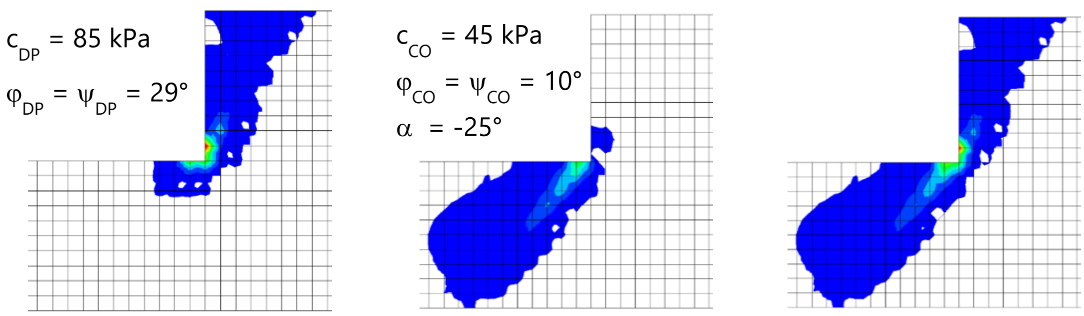
- Homogeneous isotropic linear elasticity
- Homogeneous orthotropic linear elasticity
- Isotropic linear elasticity with E and ν varying linearly with depth
- Isotropic linear elasticity with E = f(z)
- Non-linear elasticity of the Hardening Soil Model
- Non-linear elasticity of Cam-Clay
- Non-linear elasticity of Duncan
- Non-linear elasticity of Fahey-Carter

Plasticity criterions

- Coulomb
- Mohr-Coulomb
- Hardening Soil Model
- Hoek-Brown
- Mohr-Coulomb with c and φ = 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)

Example 1: Simple non-supported excavation

With this simple model, an excavation without supports is analysed. The soil is modelized by an elastic law with a Drücker-Prager criterion, then by an elastic law with a Oriented criterion and finally by an elastic law with both Drücker-Prager and Oriented criterions. Results presented below show that the influence of both criterions is well taken into account.

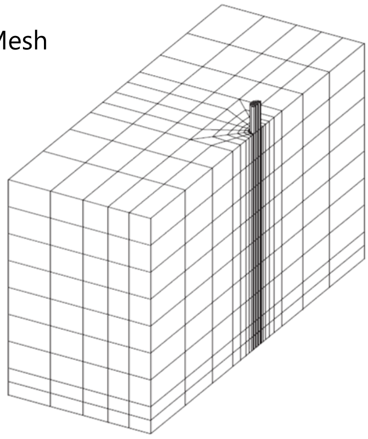


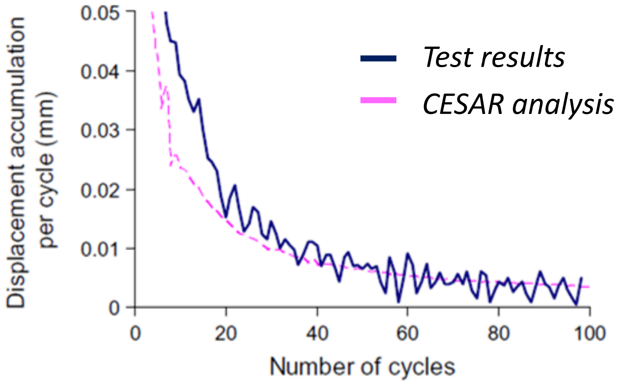
Drücker-Prager alone | Oriented criterion alone | DP + oriented criterion

Example 2

A pile is driven in a uniform soil mass and a cyclic loading is applied at its top. The soil is modelled by a combination of an elastic law with a Drücker-Prager criterion and a nonlinear kinematic hardening. Results presented below show the ability of this model to represent the displacements accumulation with number of cycles.

Mesh





3.2.3. Constitutive laws for the element families used in Coupled analysis.

Family	Constitutive model
Family 41	*Mechanics:* – linear isotropic elasticity – linear orthotropic elasticity *Hydraulics:* – anisotropic behaviour
Family 42	*Mechanics:* – linear isotropic elasticity *Hydraulics:* – anisotropic behaviour
Family 45	*Thermo-poro-elasticity:* – isotropic linear – orthotropic linear – with isotropic dilatancy *Thermo-poro-elastoplasticity, criterion:* – 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	***Thermo-poro-elasticity:*** – isotropic linear – orthotropic linear – with isotropic dilatancy ***Thermo-poro-elastoplasticity, criterion:*** – Von Mises (with or without strain hardening) – Drücker-Prager (with or without strain hardening) – parabolic – Vermeer – Nova – modified Cam-Clay – Prevost and Hoëg – oriented

3.3. Computation modules

We list here the computation modules, which are available in the present version of CESAR. Every module is characterized by a key word based on acronyms.

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 behaviour	**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	**Heat transfer**
LINE	Resolution of a thermal problem in permanent state	**Heat transfer**
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)	**Heat transfer**
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** **Heat transfer**

3.3.1. Mechanics: stage construction

3.3.2. Mechanics: c-phi reduction analysis

3.3.3. Mechanics: Safety factor analysis

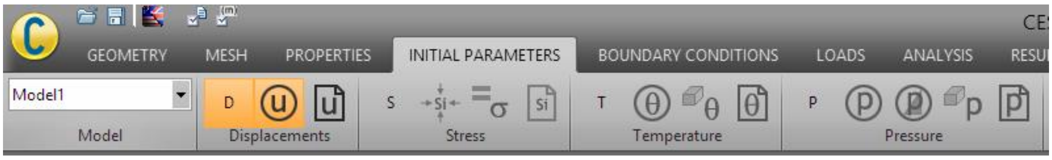
3.3.4. Hydrogeology: flow in unsaturated soils

3.3.5. Consolidation

3.4. Parameter initialization

Activating the "Initial Parameters" tab has the following effect:

- displays a toolbox to allow initializing the parameters included in the current model;
- displays 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 present 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
Hydraulic 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

3.4.1. Initial stress field

For geotechnical analysis, initial stress field is a key step. User will take benefit of 3 methods of initialisations:

1. Geostatic stress field
2. Gravity stress field
3. Uniform stress field

Geostatic stress field

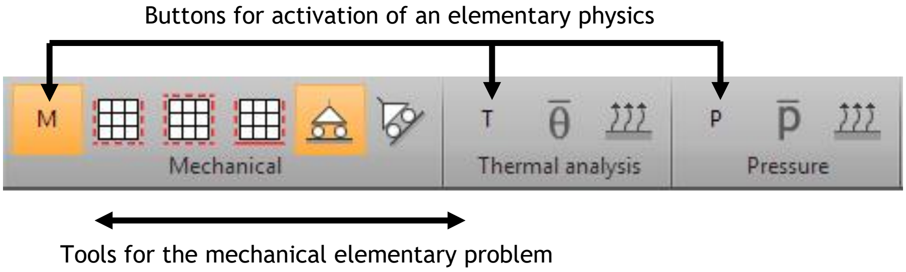
- Ko
- OCR

Gravity stress field

Uniform stress field

3.5. 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:

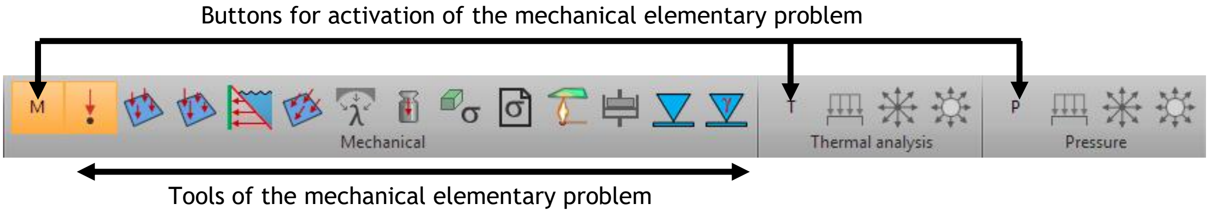


The set of tools associated with each of the potentially-applicable elementary physical problems have been listed below.

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 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

3.6. Loading features

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:



Elementary physical problem	Tools
Mechanics (M)	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
Heat transfer (T)	Uniformly-distributed flow Point flow rate Volumic flow rate
Pressure (P)	Uniformly-distributed flow Point flow rate Volumic flow rate

www.cesar-lcpc.com
Contact: cesar-lcpc@itech-soft.com
Tel.: +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