GEOMECHANICS
DR. A. FERRARI AND PROF. L. LALOUI

Project – Part 4 – 13.11.2025 (**due on 19.12.2025**)

Evaluation and design of a geotechnical project using finite element method

Problem statement

As a geotechnical engineer, you are responsible for designing an excavation in clay soil supported by two diaphragm walls, as shown in Figure 1. The soil properties have been determined in the previous project phases. The data and parameters that you have established will now be used in the geotechnical design of the structure.
In this phase, you will construct a finite element model (FEM). The results obtained will allow you to analyse the best sequence of works, the effect of the choice of constitutive model, and the importance of hydraulic flow during construction. Steady-state conditions are considered for each stage.

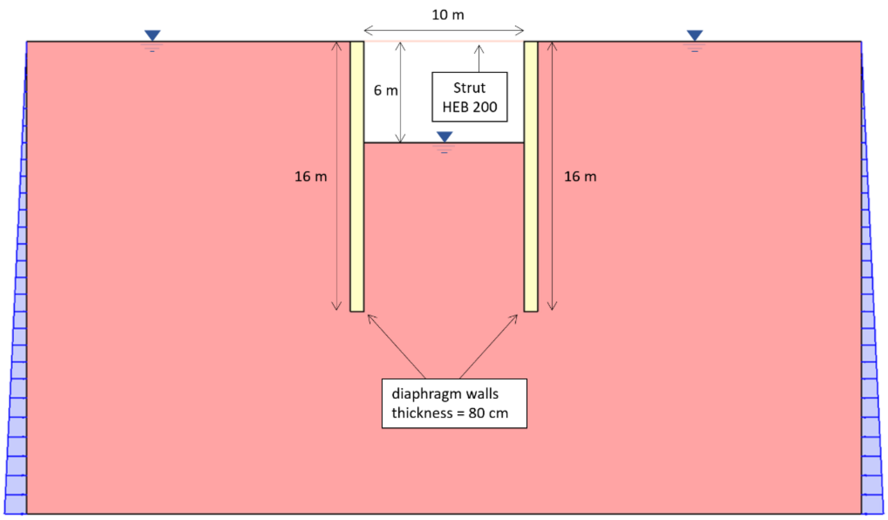


Figure 1: Geometry of the excavation problem (final stage)

This project will consider two different construction sequences: either the strut is placed at the end of the excavation (configuration 1) or at the beginning (configuration 2). This analysis demonstrates the significance of stress history during a project. You will choose the optimal configuration based on the model results.

Configuration 1

0. Initial conditions (fluid head at 0 m)
1. Installation of the diaphragm walls
2. Excavation to -1 m and fluid head at -1 m
3. Excavation to -3 m and fluid head at -3 m
4. Excavation to -5 m and fluid head at -5 m
5. Excavation to -6 m and fluid head at -6 m
6. Installation of the strut (Final step)

Configuration 2

0. Initial conditions (fluid head at 0 m)
1. Installation of the diaphragm walls and the strut
2. Excavation to -1 m and fluid head at -1 m
3. Excavation to -3 m and fluid head at -3 m
4. Excavation to -5 m and fluid head at -5 m
5. Excavation to -6 m and fluid head at -6 m
(Final step)

4. Implementation of FE model

The finite element calculation is performed with ZSoil. You will construct the model from scratch (geometry, boundary conditions, flow conditions, mesh, staging). Due to the symmetry of the structure, the model size can be reduced by modelling only half of the structure. A schematic representation of the problem is shown in Figure 2.

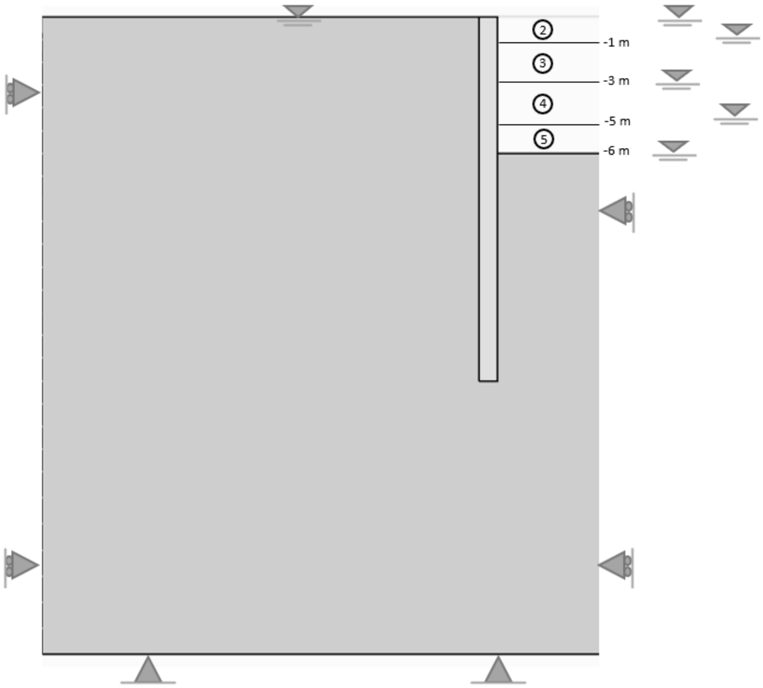


Figure 2: Schematic representation of the problem

The software can be downloaded at the following website: https://zsoil.com/student/. Follow the installation instructions on the website to obtain the free student version.

A tutorial is provided in the appendix to guide the student through the implementation steps of the software.

4.1. Data and model preparation

Follow the instructions given in the tutorial and build a first model using **Mohr-Coulomb** as the constitutive model for the soil, and **configuration 1** (installation of the strut at the end of the excavation).

One of the first steps is to define the material properties. The key soil parameters have been defined in the past phases of the project. The parameters for the concrete and the strut are given in the tutorial.

a) Given the relationship $K_0 = 1 - \sin(\phi')$, calculate the initial coefficient of lateral stress. Use this value to define the initial conditions in the FE model.

b) Which of the three triaxial tests performed in the previous phases is the most representative given the soil conditions at the bottom of the excavation? Explain your choice by comparing the pre-consolidation pressures. Express the corresponding elastic parameters (E_0 and ν_0) that will be used in the FE model.

4.2. Effect of sequence

To demonstrate the significance of stress history during a project, build a second model using **Mohr-Coulomb** as the constitutive model for the soil, and **configuration 2** (installation of the strut at the beginning of the excavation). It is recommended that the model created in the previous section be copied and the desired parameters (in this case, the existing functions) modified.

Run both models (configurations 1 and 2) considering the Mohr-Coulomb constitutive model. Compare the results and answer the following questions.

a) Plot the in-situ state of stress at the initial conditions. Explain analytically the values for effective stresses (σ'_x, σ'_y), the total stresses (σ_x, σ_y), and the pore water pressure p_w.

b) For the final stage, plot the displacements (absolute, horizontal, and vertical) in the soil. In the excavation zone, compare and comment on the two configurations.

c) For the final stage, plot the effective stresses (σ'_x, σ'_y), the total stresses (σ_x, σ_y) in the soil. In the excavation zone, compare and comment on the two configurations.

d) According to the previous results, which configuration is the most suitable? Explain why.

4.3. Effect of the constitutive model

To demonstrate the effect of the constitutive model, build two models considering **Mohr-Coulomb** and **Modified Cam-Clay** as the constitutive models for the soil, under the **best configuration** defined previously (section 4.2, question d)). Please note that in ZSoil, 'Modified Cam-Clay' is called 'Cam-Clay'.

a) For the final stage, plot the displacements (absolute, horizontal, and vertical) in the soil. In the excavation zone, compare and comment on the role of the constitutive model.

b) For the final stage, plot the effective stresses (σ'_x, σ'_y), the total stresses (σ_x, σ_y) in the soil. In the excavation zone, compare and comment on the role of the constitutive model.

4.4. Effect of the flow

As the flow is calculated separately from the mechanical model in ZSoil, the constitutive model does not influence the flow calculation. Similarly, the two configurations considered do not affect the flow distribution. Therefore, you may select any model you wish.

a) Plot the total head for all the excavation stages (-1 m, -3 m, -5 m and -6 m). Draw the flow net on the total head plots for all the excavation stages by hand. Check the consistency of the flow by looking if the flow lines are in agreement with the flow net.

b) Visualize the flow velocities for all the excavation stages (-1 m, -3 m, -5 m and -6 m). For the worst-case scenario, you are asked to design the pumping system at the bottom of the excavation. Provide an estimate of the flow in liters per meter per day [l/m.day] for the entire structure. Based on this value, do you think it is needed to install a pumping system?

c) A new parking lot has just been built thanks to your excavation. A slab has been cast and the bottom of the excavation is now impermeable. What will be the consequences and how it will affect the general distribution of effective stresses (σ'_x, σ'_y) and the pore water pressure (p_w)? Please provide a reason response to this question, it is not necessary to create a new model.

d) When the water flows in the soil, it induces a seepage force on the particles in the direction of its motion. If this force is high, it can displace the soil particles and particles may get carried away by the flowing water. This phenomenon leads to a serious problem in soil engineering, which is known as piping. The safety factor for piping F is defined as follows and should be higher than 2.5:

$$F = \frac{i_c}{i} > 2.5$$

where i_c is the critical hydraulic gradient and i is the actual hydraulic gradient. For low cohesive soils ($c \approx 0\ kPa$), the critical gradient is $i_c = \frac{\gamma'}{\gamma_w}$. The hydraulic gradient $\left(i = \frac{h_2 - h_1}{L}\right)$ is computed close to the zone of excavation (in orange in Figure 3).

Compute the safety factor for piping. Does piping is verified for this project?

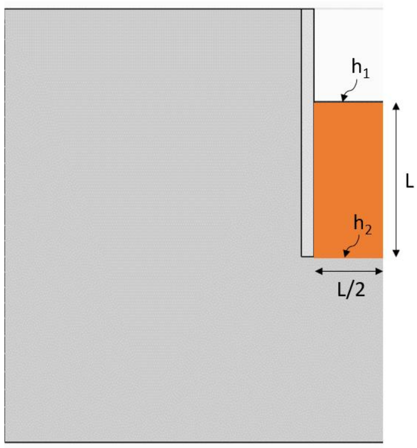
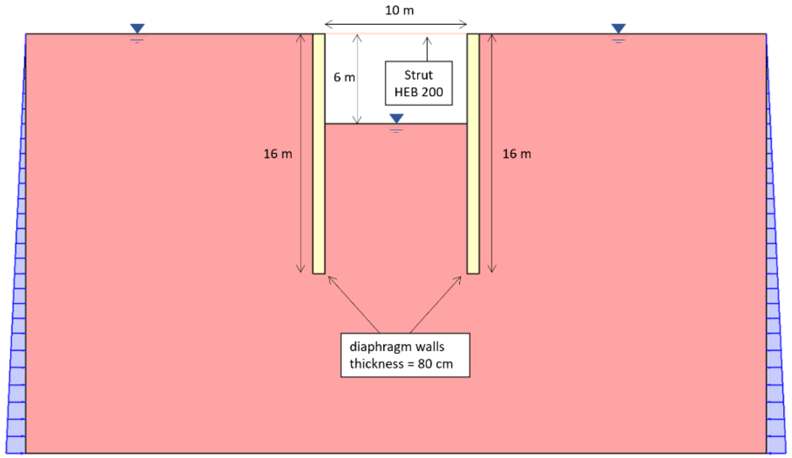


Figure 3: Area of interest for piping verification

APPENDIX

ZSoil - Tutorial



Geomechanics

IMPLEMENTATION OF FINITE ELEMENT MODEL WITH ZSOIL

Semester project – Part 4

Dr. ALESSIO FERRARI AND PROF. LYESSE LALOUI

Laboratory of soil mechanics – Fall 2025

13.11.2025

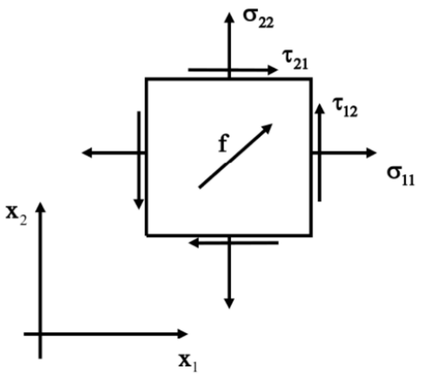
Introduction ZSoil

In the following sections a step-by-step tutorial is presented to solve the proposed problems with the use of the software ZSoil v.25.04. Three main steps will be accomplished:

- a preprocessing phase, where all the input data are introduced (problem geometry, material properties, mesh, boundary conditions, driver definition, ...);
- the calculation step, where the boundary value problem is solved with the finite element method;
- a postprocessing phase, in which the obtained results are displayed and analyzed.

<u>Remarks:</u>
To indicate a command to be selected from a menu, the notation 'Menu > Submenu > Command' will be used.

Please take note of the positive sign convention for the stress field, defined with respect to a Cartesian coordinate system.



Positive stresses (2D case)

Content

- Preselection of the analysis
- The preprocessing phase
- The calculation step
- The postprocessing phase

Preselection of the analysis

IMPLEMENTATION OF FINITE ELEMENT MODEL WITH ZSOIL

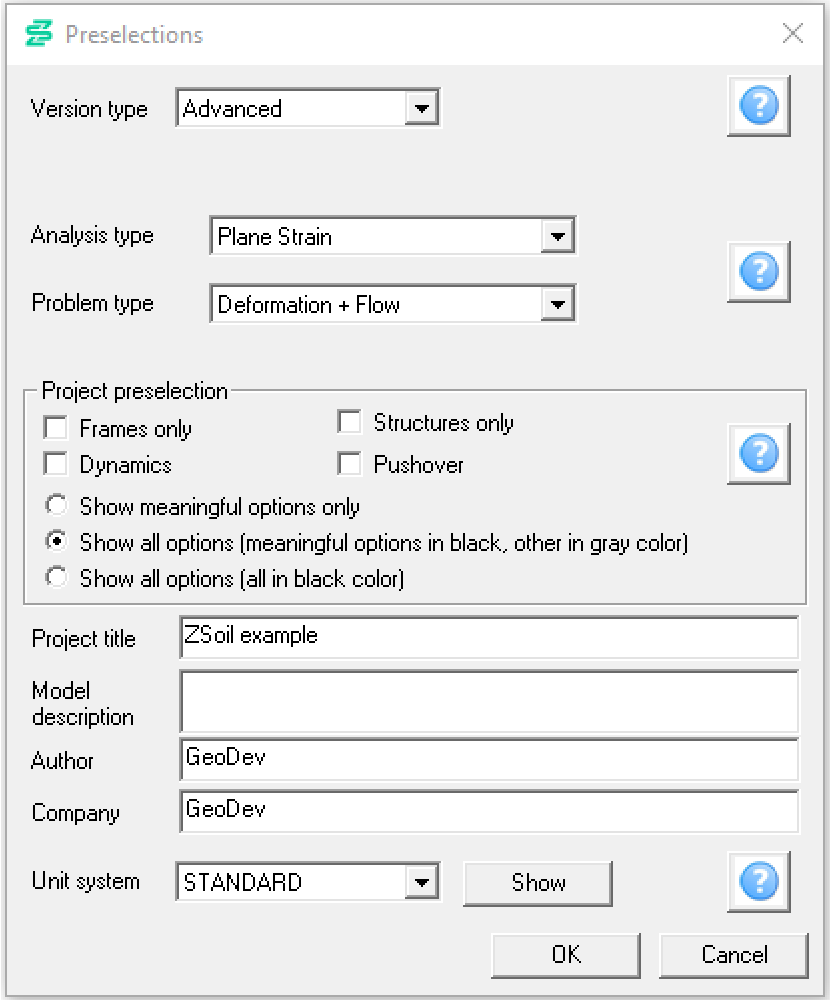
Preselection of the analysis

Set of parameters for the exercise :

- Version type : **Advanced**
- Analysis type : **Plane Strain**
- Problem type : **Deformation + Flow**
- Project preselections : **Show all options (meaningful options in black, other in gray color)**
- Unit system : **Standard**

Save the project with 'File > Save as' in your personal folder. Remember to save regularly your project with 'Ctrl + S'.

Select 'Control > Units' and take note of the unit system in use.



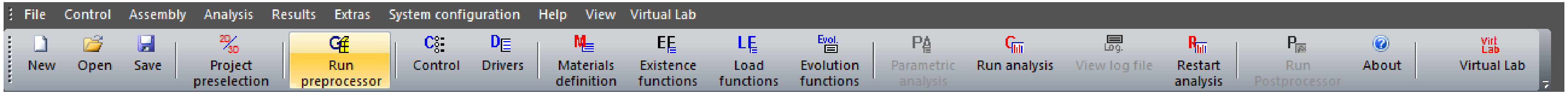
The preprocessing phase

IMPLEMENTATION OF FINITE ELEMENT MODEL WITH ZSOIL

The preprocessing phase

In this phase, you will introduce the input data. (problem geometry, material properties, mesh, boundary conditions, driver definition, ...).

- Select 'Run preprocessor'



The preprocessing phase

MATERIAL

- Select 'Assembly > Materials'. To add a material, click on ‘Add’. Rename the material considered and select the type under ‘Continuum & structure type’ and the constitutive law under ‘Material formulation’ of your material.

- In this study, two constitutive models are considered (Morh-Coulomb or Modified Cam-Clay). Select the correct material formulation according to the project statement. Please note that Modified Cam-Clay is called Cam-Clay in ZSoil.

Number	Material	Material type	Material formulation
1	Clay (soil)	Continuum	Morh-Coulomb or Cam-Clay
2	Concrete	Continuum	Elastic
3	Strut	Beam	Beam

The preprocessing phase

- For each material, enter the following parameters. If nothing else is specified, consider the other parameters with their default values.

Clay (Soil)	Concrete	Strut
Elastic • Young modulus E • Poisson ratio ν	**Elastic** • Young modulus E = 30 GPa • Poisson ratio ν = 0,2	**Elastic** • Young modulus E = 210 GPa • Poisson ratio ν = 0,3
Unit weight • Saturated weight / unit volume γ • Dry weight / unit volume γ_D • Fluid weight / unit volume γ_F • Initial void ratio e_0	**Unit weight** • Saturated weight / unit volume γ=25 kN/m³ • Dry weight / unit volume γ_D=25 kN/m³ • Fluid weight / unit volume γ_F=10 kN/m³ • Initial void ratio e_0=0	**Unit weight** • Unit weight / unit volume γ = 75 kN/m³
Flow • Darcy's coefficient k = 1e-8 m/s • Residual saturation ratio Sr = 0,1 • Air entry suction α = 0,1 1/m • Pore size distribution n = 2	**Flow** • Unselect to have an impermeable layer	**Cross section** • Type - Profiles • Database - CatPro.pro • Section - HEB 200 • Interval between beams a = 1 m

The preprocessing phase

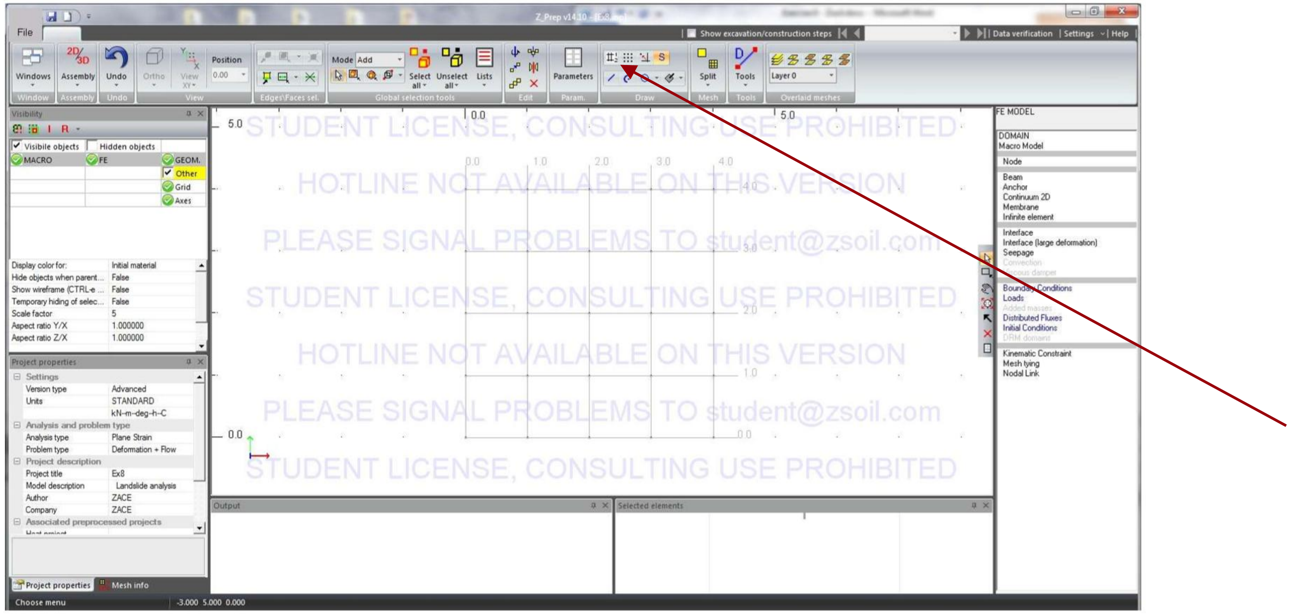
- Parameters table (continued)

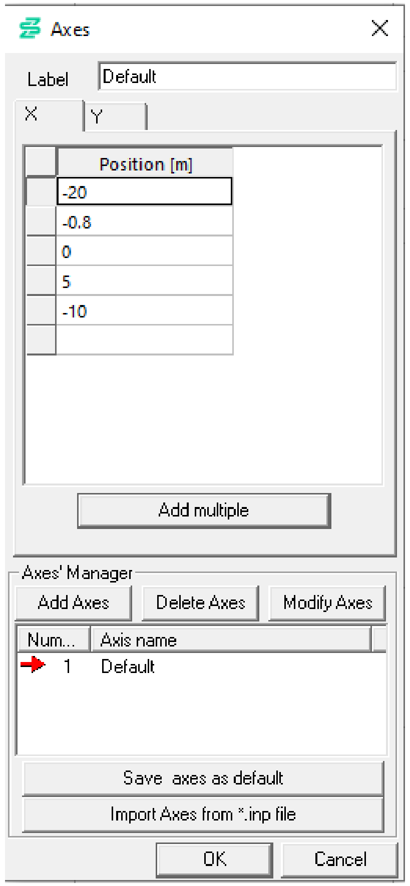
Clay (Soil)	Concrete	Strut
Non linear For Morh-Coulomb: • Cohesion c • Friction angle ϕ • Dilatancy angle $\psi=\phi/3$ For Modified Cam-Clay: • Slope of critical state line M_c • Slope of 1st consolidation λ • Slode of 2nd consolidation κ		**Main** • As default
Initial K_0 state • $K_0(x') = 1 - \sin(\phi')$ • $K_0(z) = 1 - \sin(\phi')$		

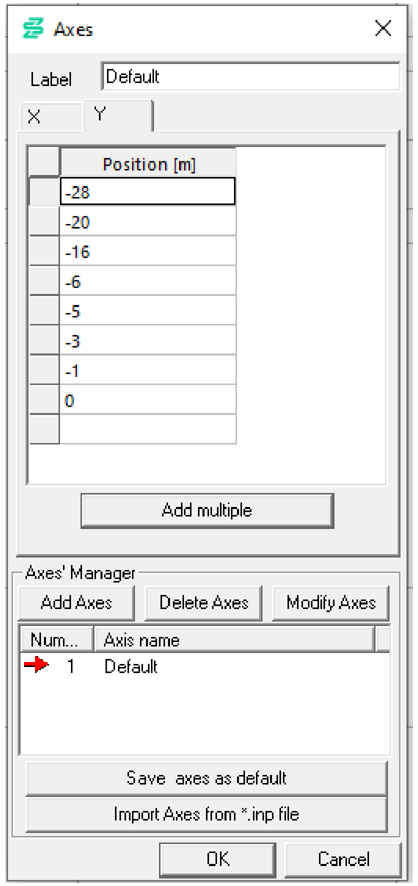
The preprocessing phase

GEOMETRY

Set the construction lines ('Draw > Construction Axes'). Delete the actual positions ('Delete all'), then add the following positions writing the values in the position box and clicking ‘Add.







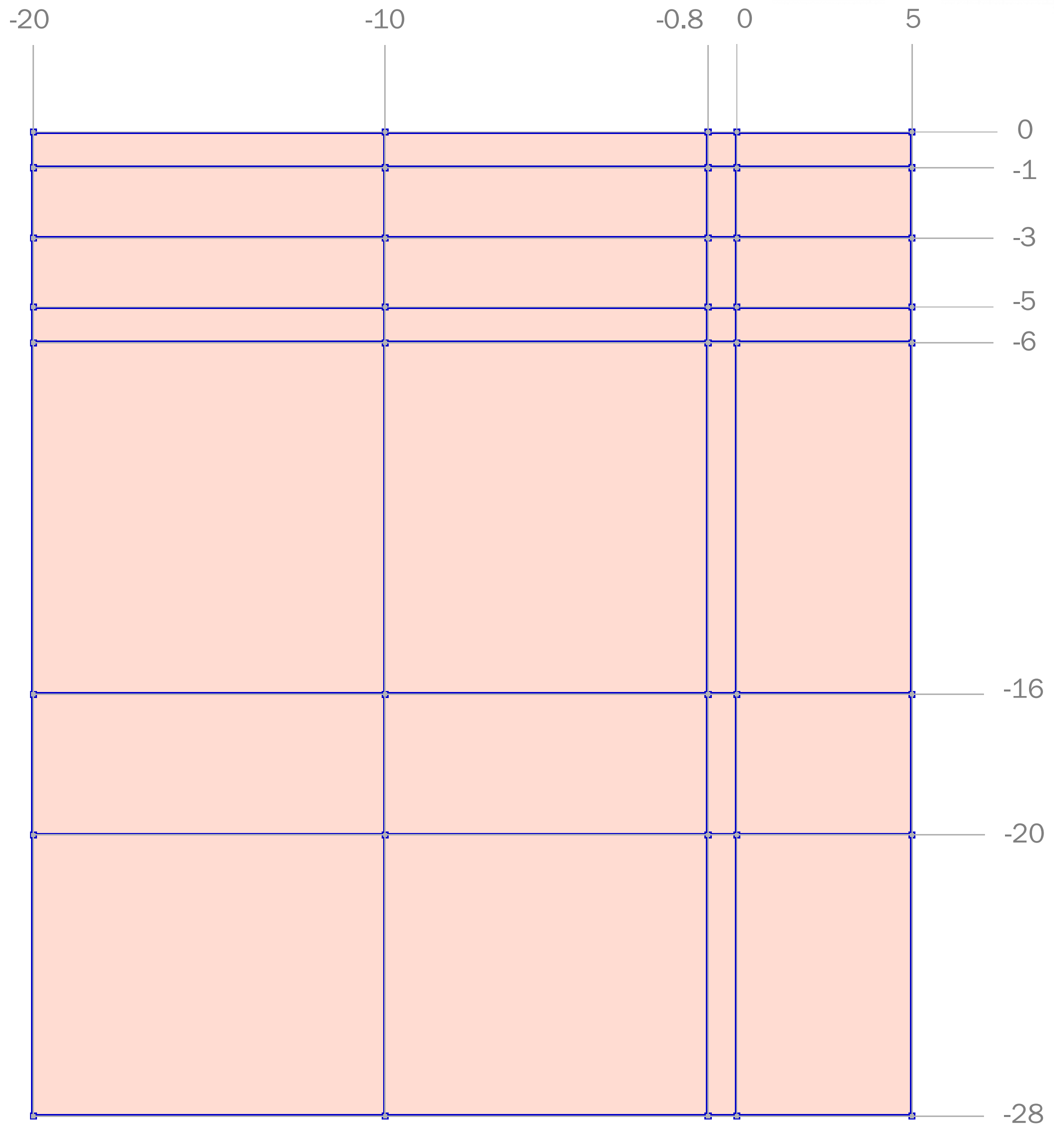
The preprocessing phase

Select ‘Macro Model > Point > Create ... > Point’. Select each intersection grid point and define a point. Press ‘Esc’ when finished.

Select ‘Macro Model > Objects > Create ... > Line ... > By 2 points’. Unselect continue. Draw lines between each points.

Subdomains have to be created inside the drawn contours. On the right menu select ‘Macro Model > Subdomain > 2D continuum inside contour' and click inside the contours.

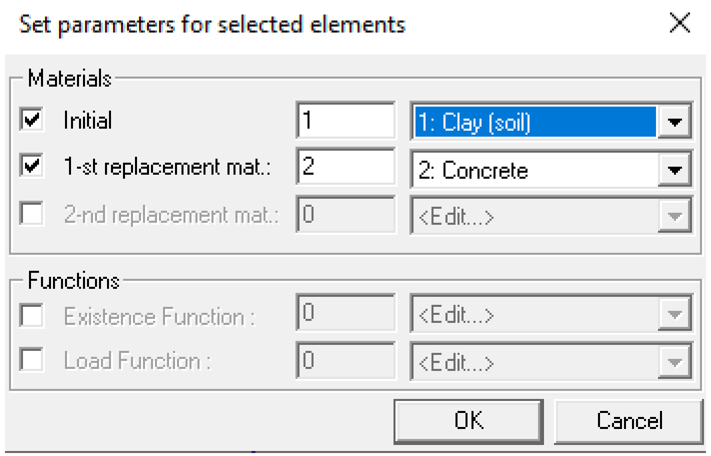
Subdomains will be created considering the material number 1. Ensure that this is the clay (soil). If not, go to ‘Select all > Select all’ and then on ‘Macro Model > Subdomain > Parameters’. On material initial, select the clay (soil).

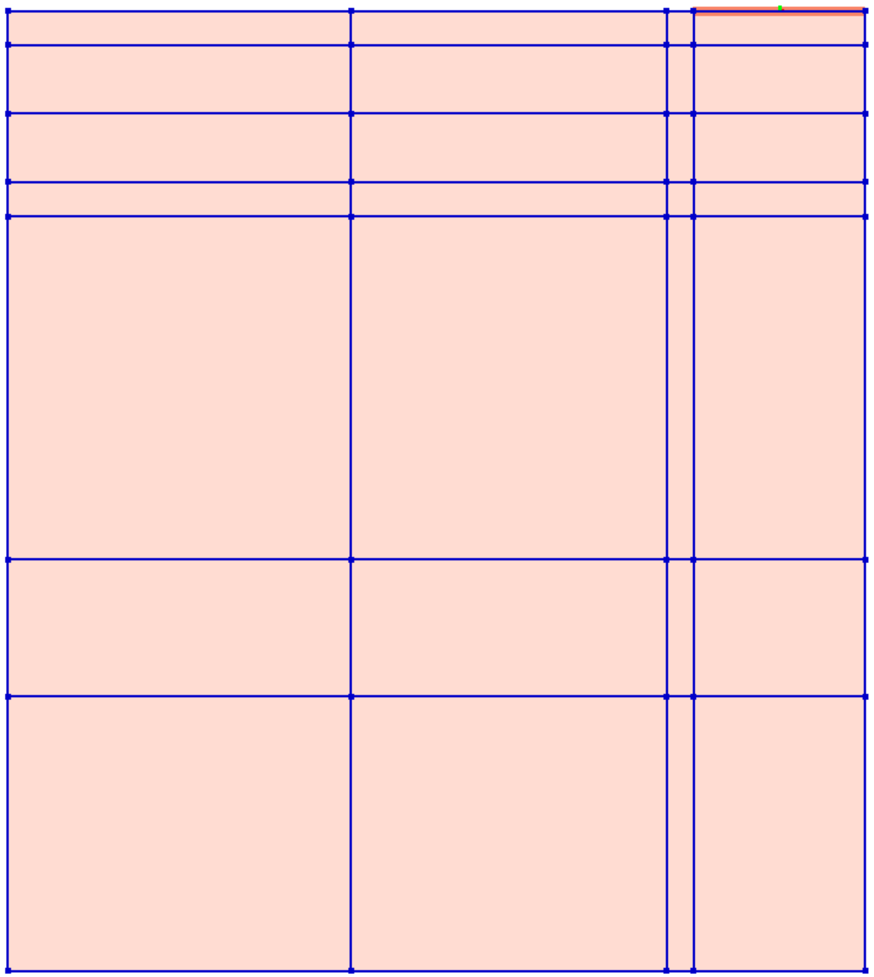


The preprocessing phase

Select the line with the coordinate x∈ $[0\text{m}; 5\text{m}]$, y = 0m. Go to 'Macro Model > Subdomain > Beam on object(s)'. Define the initial material as the strut.

Select the element corresponding to the diaphragm wall. At initial conditions, these elements were made of soil. Then the diaphragm wall is installed. Go to 'Macro Model > Subdomain > Parameters'. Define the initial material as the clay (soil) and the 1-st initial replacement material as concrete.





The preprocessing phase

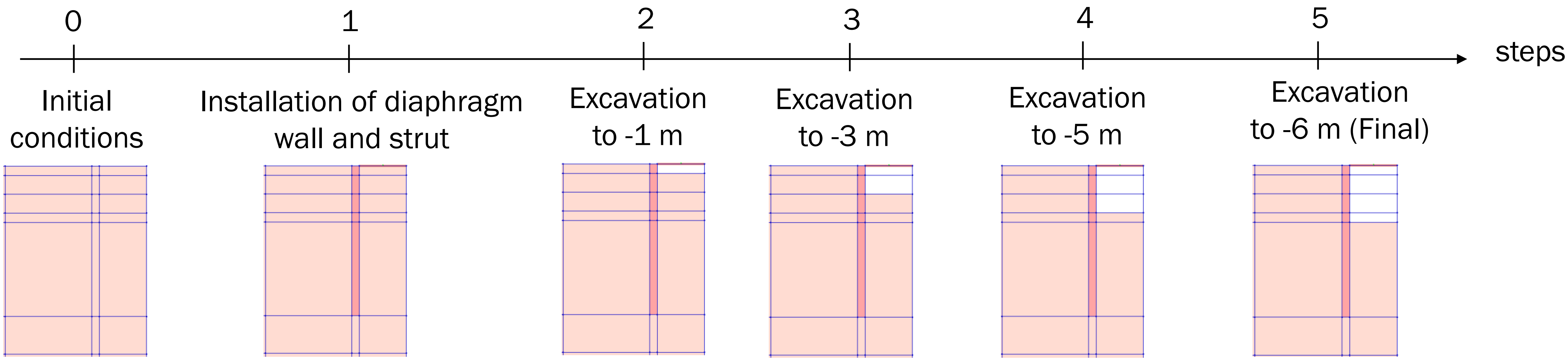
STAGING

We will now define the different steps of the project. In this study, two staging are considered:

- Configuration 1 : Strut installation at the end of the excavation
- Configuration 2 : Strut installation at the beginning of the excavation

Select the correct configuration according to the project statement.

For exemple, the sequence is as follows for the configuration 2 :



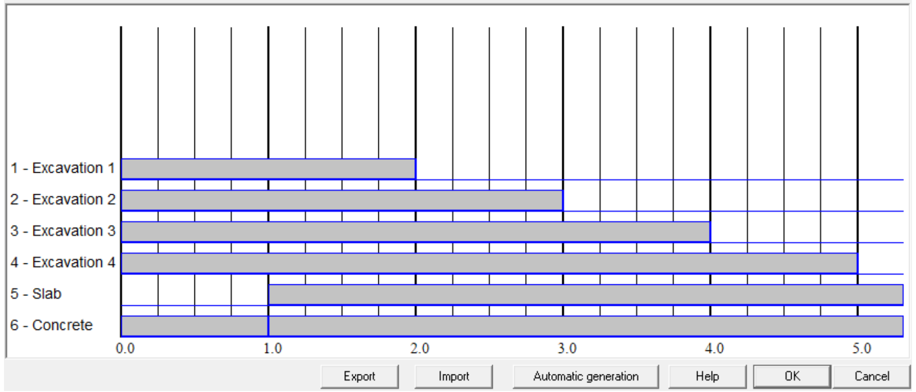
The preprocessing phase

STAGING

The steps are defined through existence functions. Select 'Assembly > Existence function'. Define an existing function for each steps according to the studied configuration. Please note that :

- Active period 1 refers to the initial material
- Active period 2 refers to the 1-st replacement material
- The active period are defined through t_0 (first step the material should appear) and t_1 (the last step the material should disappear)
- If a material appears and then is always present, enter INF for t_1
- If a material is always existing, there is no need to define an existing function

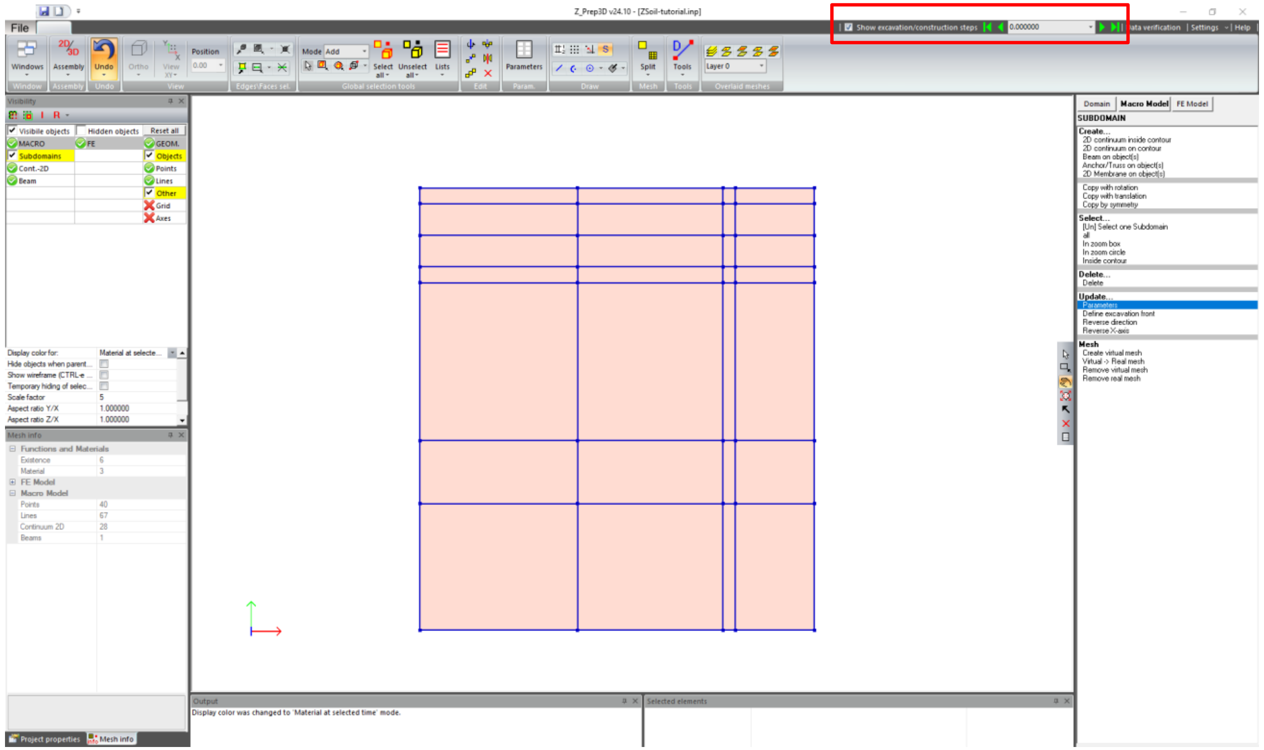
For configuration 2, the existence function should be as follows :



The preprocessing phase

Select each elements and associated the corresponding existence function through ‘Macro Model > Subdomain > Parameters > Existence function’.

Visualize the staging by clicking on show excavation/construction steps. Please note that if an element appears in step i, it will be visualised in step i+1,



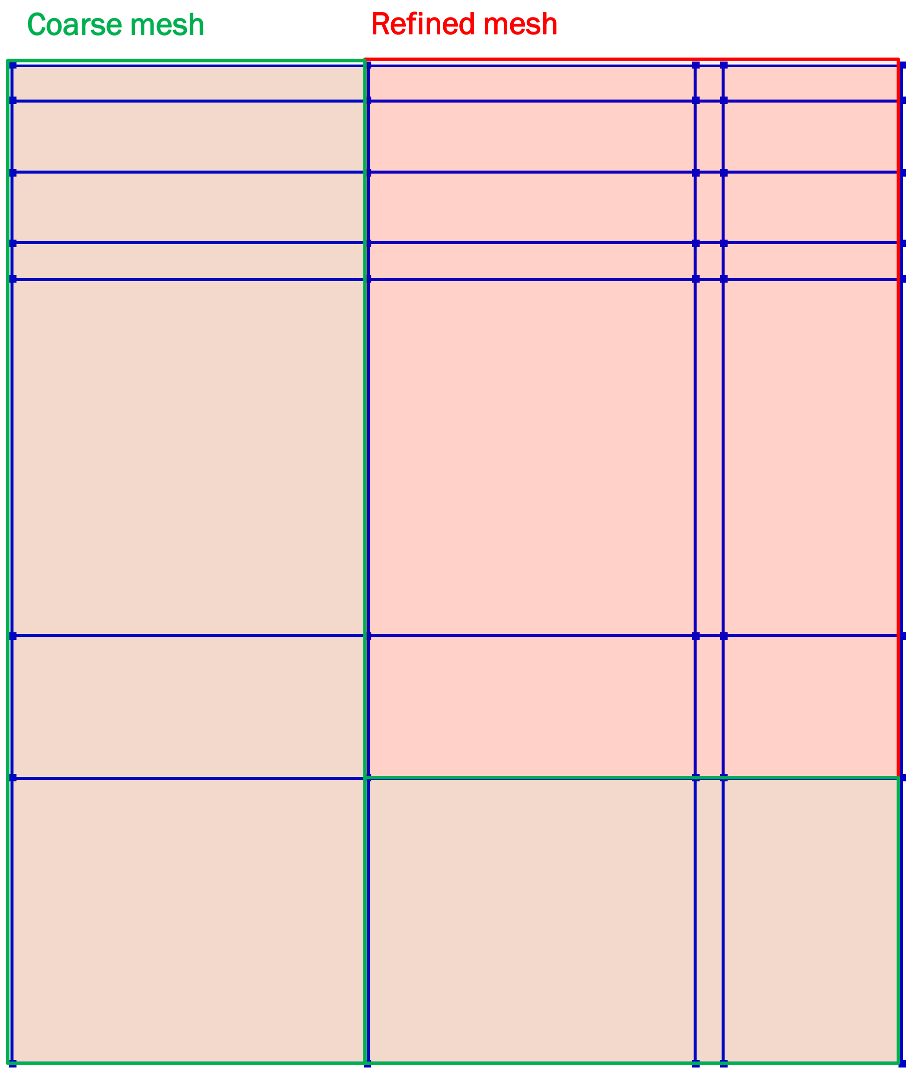
The preprocessing phase

MESH

To create the mesh, select on the right menu 'Macro Model> Subdomain > Create virtual mesh' then click inside a subdomain.

You need to select each subdomain independently.

We would like a refined mesh close to the excavation and a coarser mesh on the boundary.



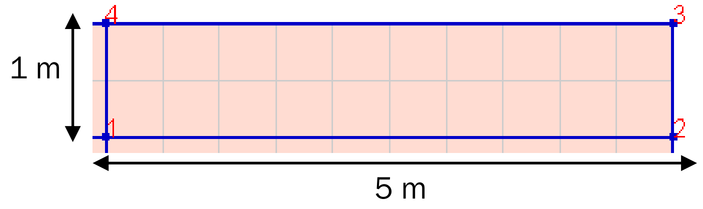
Example of refined /coarse virtual mesh

The preprocessing phase

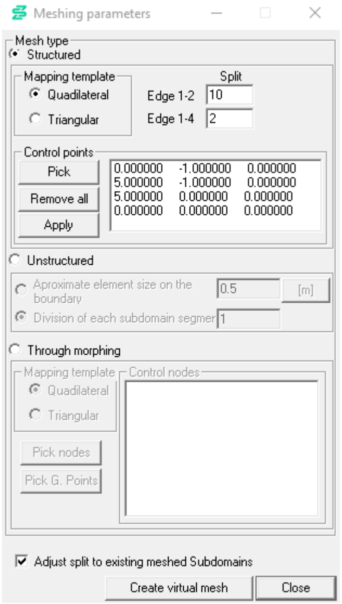
<u>Mesh:</u>

In the meshing parameters window select structured mesh type, with a quadrilateral mapping template. 4 red numbers automatically appear on the 4 subdomain corners and their coordinates appear in the 'Control points' box (Note that you could obtain a different order than the one in the figure below). These points are used to split the subdomain edges. The number in the split boxes corresponds to the number of division of an edge.

Click on 'Create virtual mesh' at the bottom of the window



Example of refined mesh (element size : 0,50 m)

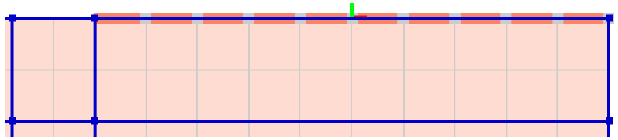


The preprocessing phase

Mesh:

For a beam element (the strut), simply select it and select on the right menu 'Macro Model> Subdomain > Create virtual mesh'. Keep the default parameters as they are.

Once all the subdomains have a virtual mesh, click on select all. On the right menu, select 'Macro Model> Subdomain > Virtual -> Real mesh'

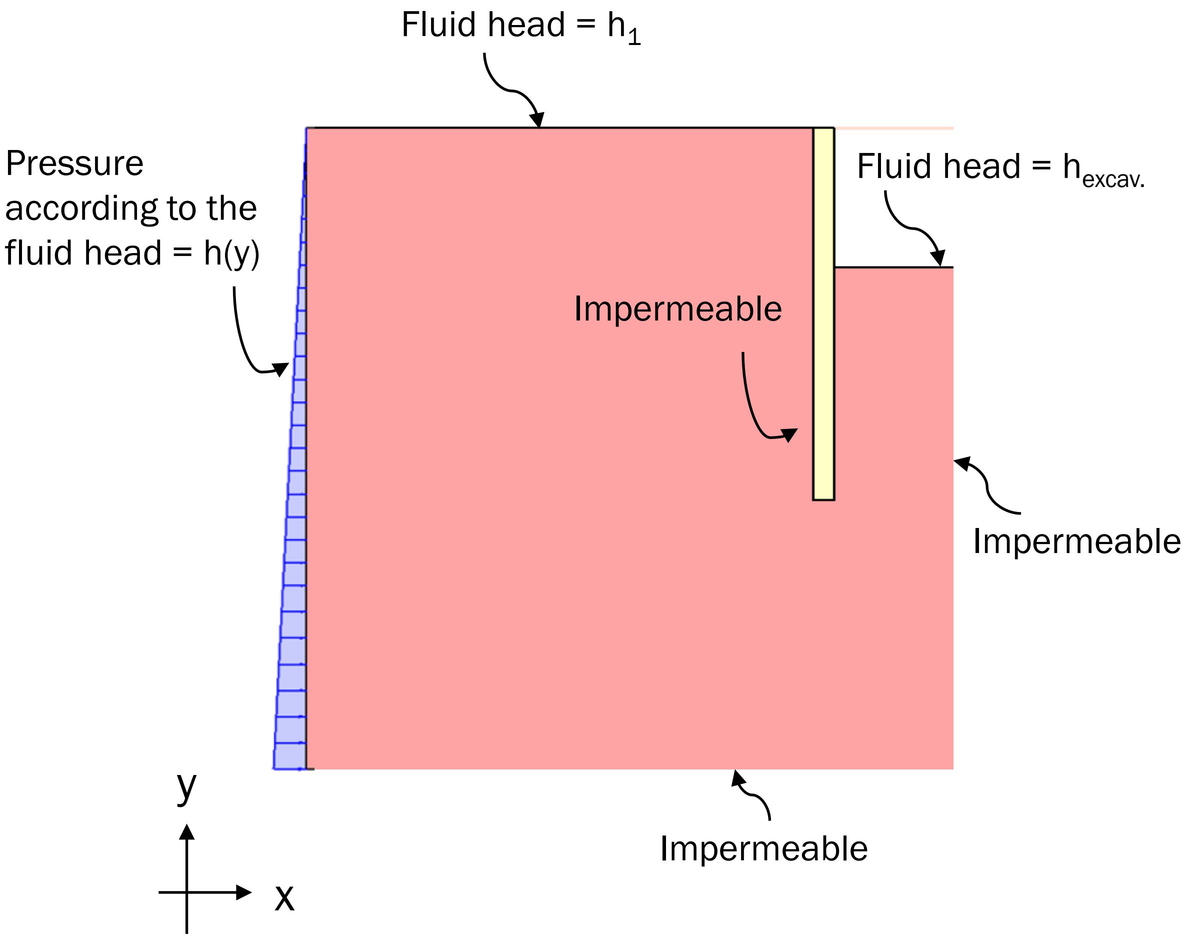


The preprocessing phase

FLOW CONDITIONS

Since we consider the symmetry, the boundary conditions are as in the Figure.

Please note that during the excavation, the fluid head is evolving according to the excavation depth.



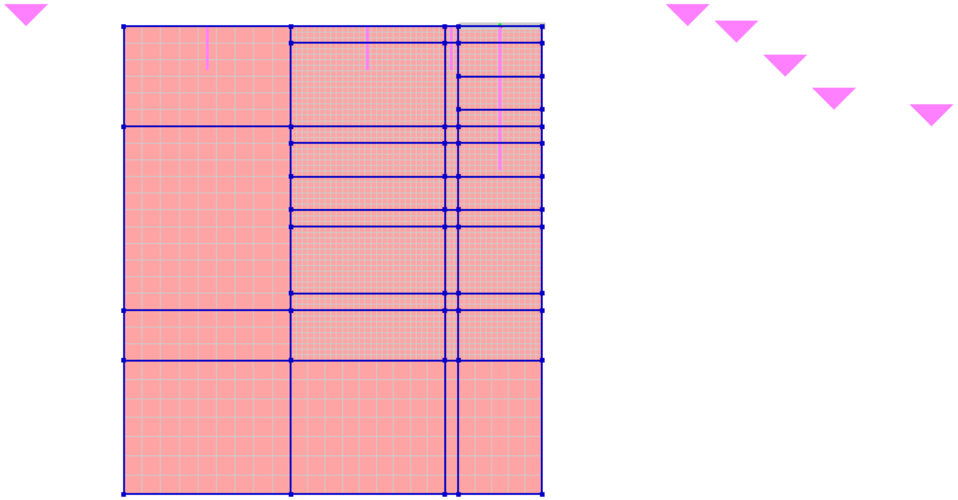
The preprocessing phase

To define the fluid head, select the edges with the same fluid head on the Macro Model with :

Select 'Macro Model> Pressure BC > Fluid head on selected edges' and enter :

- Corresponding fluid head [m]
- Existence function defined the same way as previously

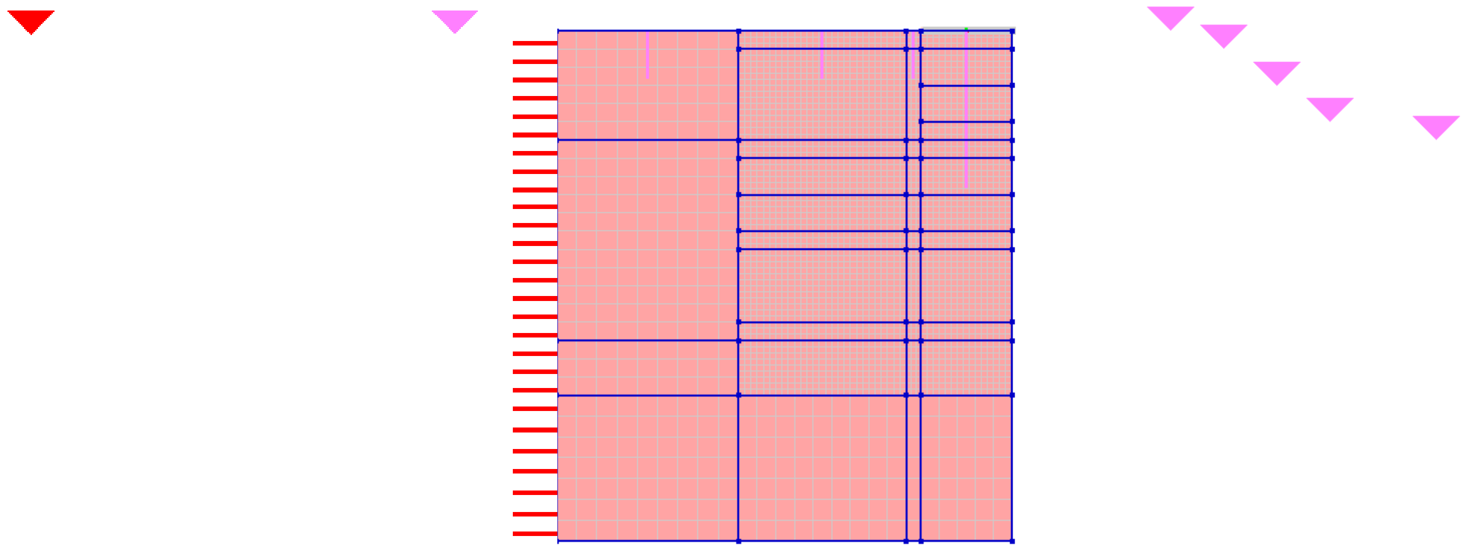
The pink triangles should be placed at the corrected depths. Check that the conditions are well implemented by visualising the different steps.



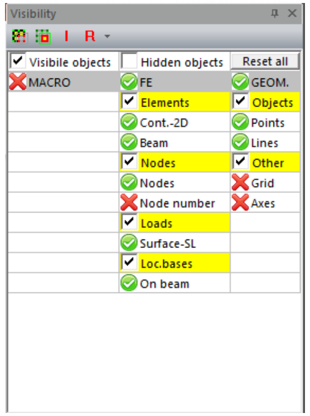
The preprocessing phase

To define the pressure according to fluid head, select the boundary edges on the left (x = -20m, y ∈ $[-28\mathrm{m}; 0\mathrm{m}]$. Select 'Macro Model> Surface load > Pressure via Fluid Head > On selected edges (SH)' and enter the corresponding fluid head at the surface

The red triangle should be placed at the corrected depth.



The preprocessing phase

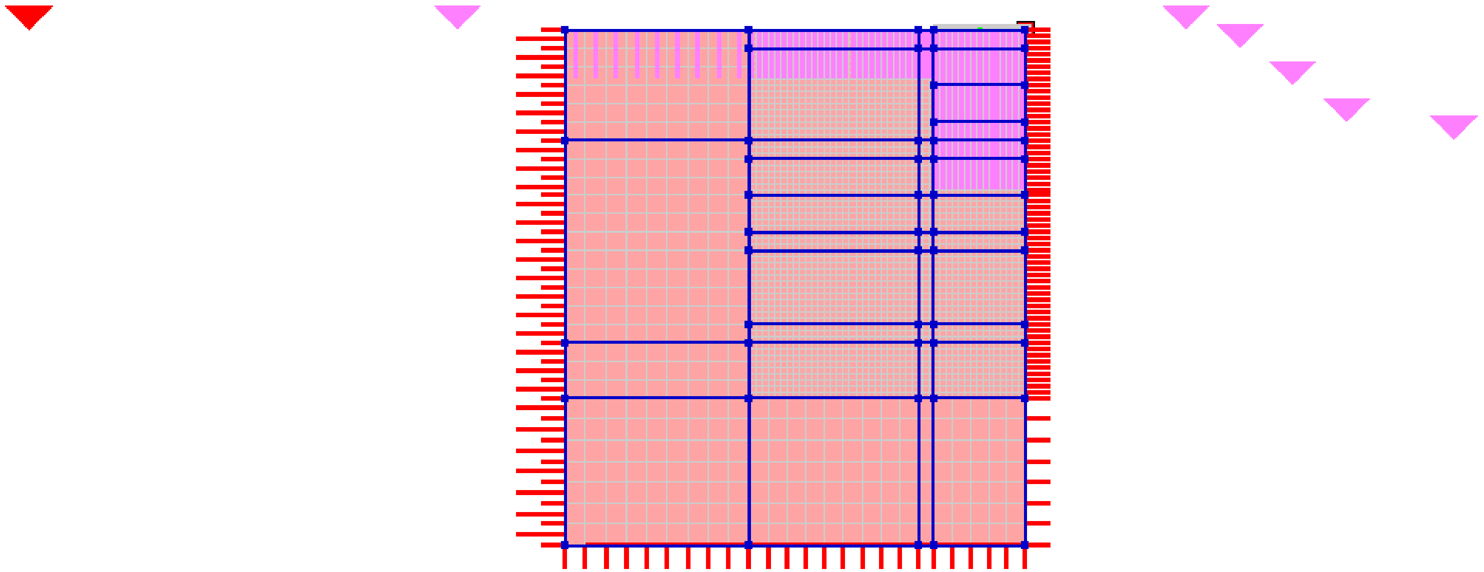


Check that the flow conditions appear in the FE model (select FE in the left pane).

The edges where no boundary conditions have been applied are impermeable.

BOUNDARY CONDITIONS

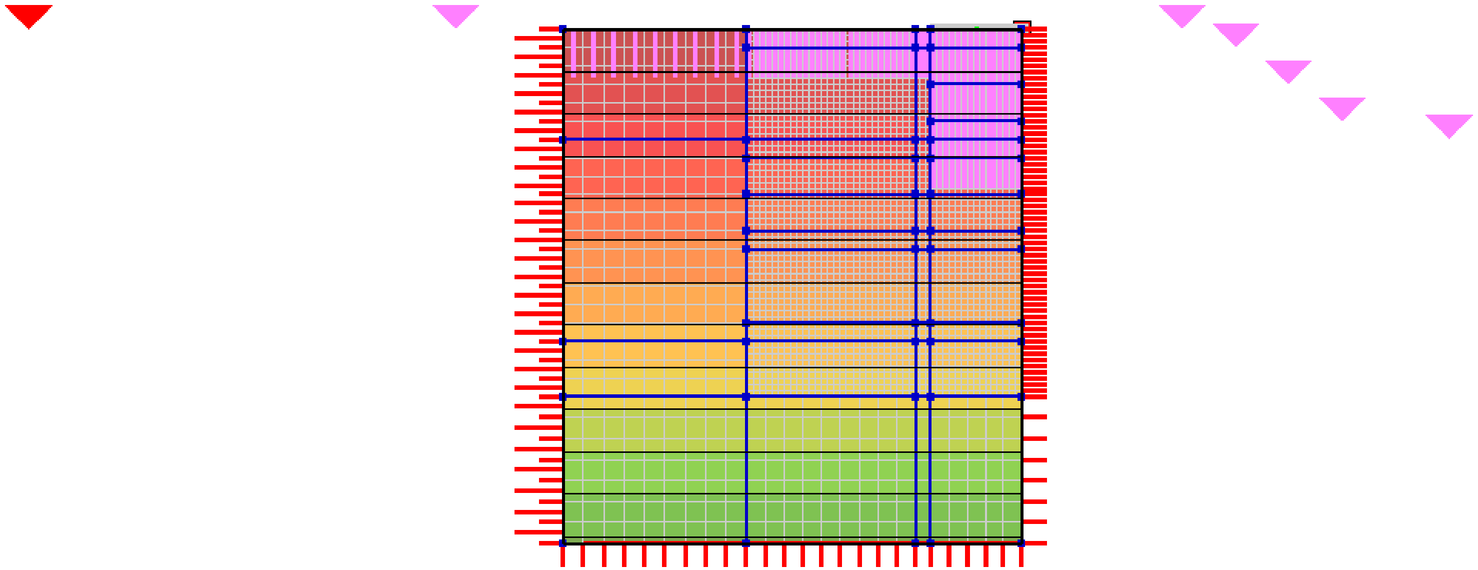
Select all. Select on the right menu ‘FE Model > Boundary conditions > Solid BC > On box’.



The preprocessing phase

INITIAL CONDITIONS

Select all. Select on the right menu 'FE Model > Initial conditions > Initial stresses > On bounding box'. Click on simplified definition and enter the saturated density (Gamma) and the initial K_0 (K_0). The vertical stress (Sig YY) is zero since we don't have any external loading.



Your model is ready !

The calculation step

IMPLEMENTATION OF FINITE ELEMENT MODEL WITH ZSOIL

The calculation step

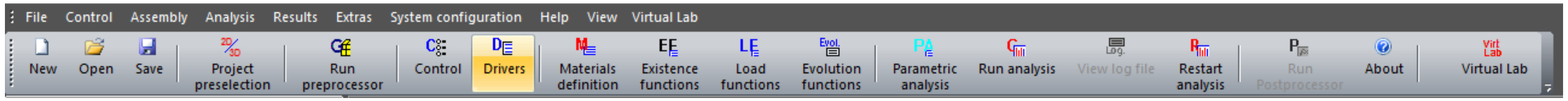
The sequence of the calculations to be performed is now specified selecting 'Drivers'.

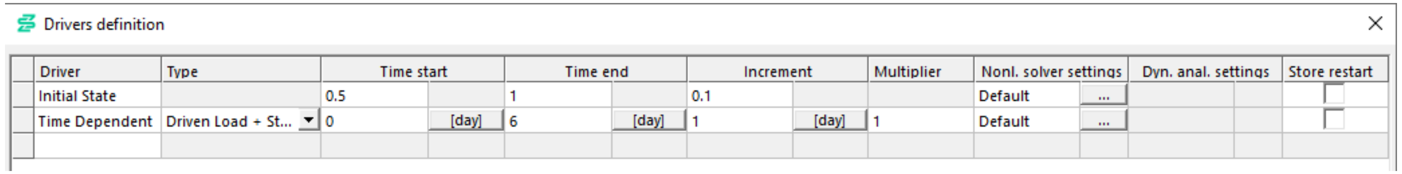
An initial state driver is used to define in situ stress state induced by gravity as well as other loads and fluxes acting at time instance t = 0. The default values for this driver can be applied.

The Time-dependent driver is then used to calculate the evolution of the stress-state, deformation, and flow. Select 'Driven Load + Steady state flow'. Initial and final time for the driver have to be inserted along with the time increment. We will consider :

- Time start : 0
- Time end : last step
- Time increment : 1

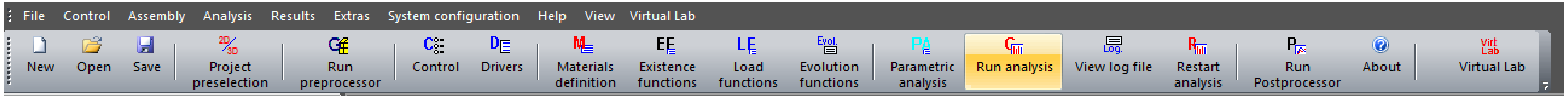
For example (configuration 2) :

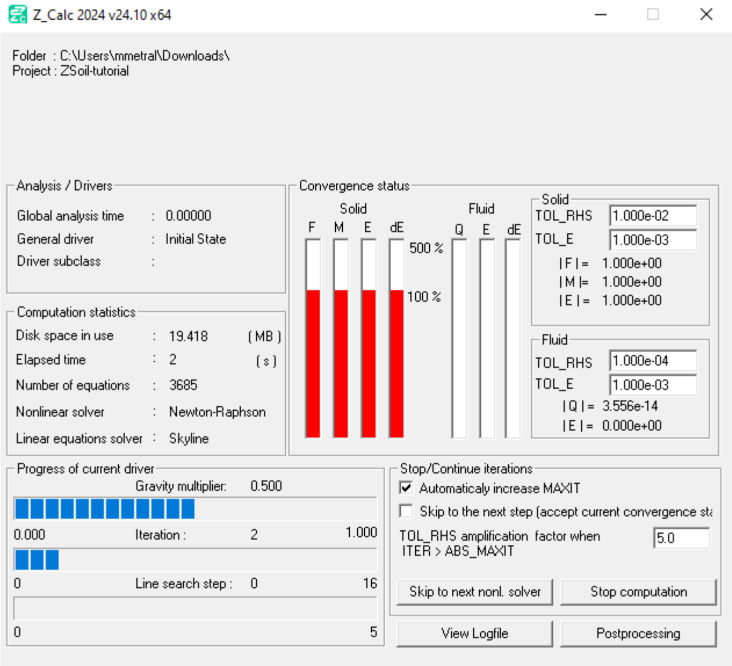




The calculation step

When data input phase is completed, launch the calculation selecting ‘Analysis > Run Analysis’ from the main window menu. The calculation module window appears, and the calculation progress can be followed.



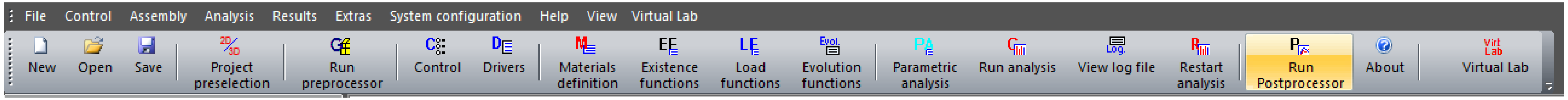


The postprocessing phase

IMPLEMENTATION OF FINITE ELEMENT MODEL WITH ZSOIL

The postprocessing phase

When calculation is finished, launch the postprocessor from the main window menu: ‘Results > Postprocessing’.



From the menu in the high right corner ‘current time’, it is possible to select a time step; then select in ‘graph options’ tab ‘Results > Maps’. The result to be visualized can be selected in the ‘graph tab’ with ‘Settings’. For example, in the window we can select ‘Nodal quantities’ to visualize the displacement, or ‘Continuum’ to visualize stresses.

Graphical settings can be adjusted here. Once the variable has been selected, it is possible to switch among the calculation steps using the arrows ‘<’ and ‘>’ in the top right corner. This is very useful to see the evolution of the variables with time in the overall domain.

To copy the figures from ZSoil to Word: click on ‘Ctrl+C’ in the ZSoil window and ‘Ctrl+V’ in a Word document.

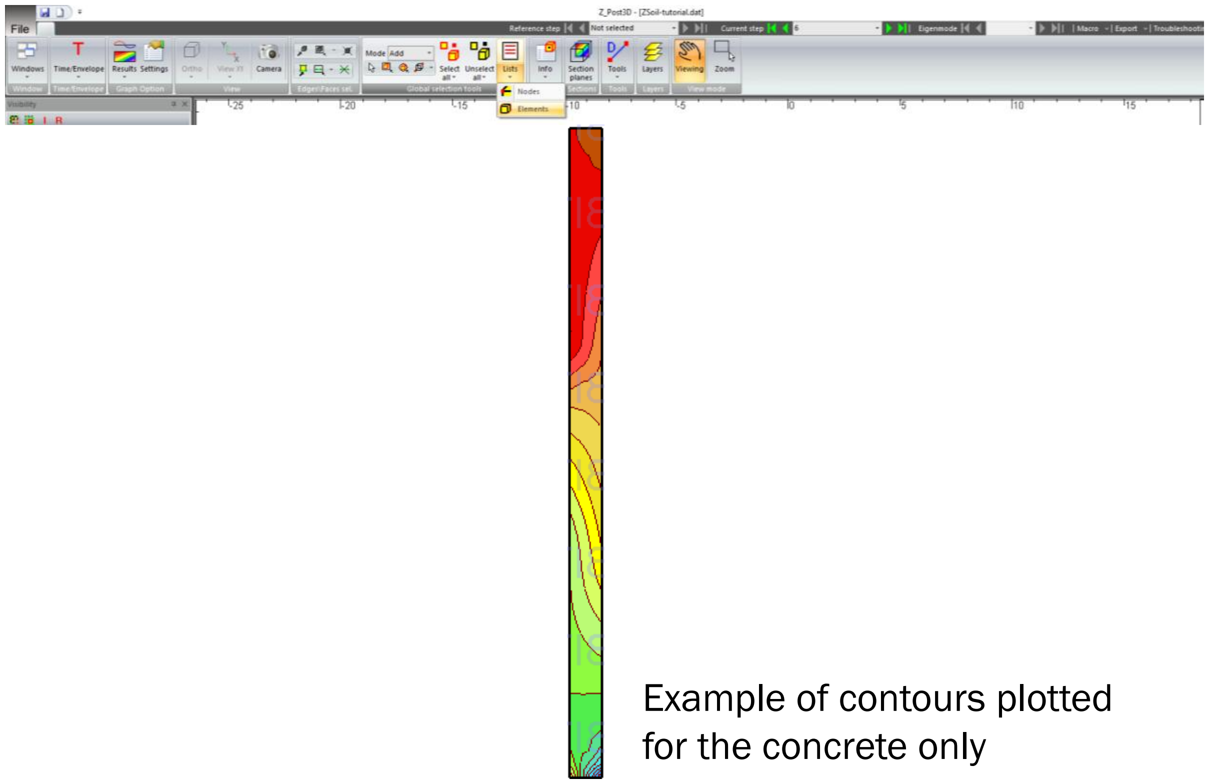
The postprocessing phase

Different types of results can be visualized and selected in the ‘Results’ section. This is a non-exhaustive list of possible representations:

- Deformed mesh
- Fluid velocities
- Maps (stress, displacement, total head, pore pressure, fluid velocities, …)
- Element and nodal time history
- Results in Beams

The postprocessing phase

Please note that the results are plotted for the entire model (all materials are included). If you want to focus on a specific material, select ‘Lists > Elements > Section rule > Material’ and select the material of interest. On the bottom left, select ‘Reset MIN/MAX to visible results’



Example of contours plotted for the concrete only