

# GeoXPM INTERFACE - GENERAL

**GeoXPM Menu**

File Edit View Tools Macro Part Measure Windows Help **GeoXPM**

**Solids/View toolbar**

**GeoXPM main toolbar**

**Objects tree**

**Object properties**

**Interactive window**

The screenshot shows the GeoXPM interface within FreeCAD. At the top is the 'GeoXPM Menu' with options like File, Edit, View, Tools, Macro, Part, Measure, Windows, Help, and GeoXPM. Below the menu are two toolbars: 'Solids/View toolbar' and 'GeoXPM main toolbar'. The 'Objects tree' on the left lists simulation components: Simulation Domain, Bot, Left, Front, Back, Excavate01, and Excavate02. The 'Object properties' panel at the bottom left shows details for 'Excavate01', including 'Type of object: Soil', 'Meshing mode: Face', 'Boundary type: Free-slip', and 'Object ID: 1'. The central 'Interactive window' displays a 3D model of an excavation site with a staircase and a 'RIGHT' label on a wall. The status bar at the bottom indicates the preselected object: 'GeoXPM.Solid007.Face6 (11.359490 m, 3042.128418 mm, 840.509827 mm)'.

**Stages construction tab**

Model Tasks Stages

Add Stage Remove Last Stage

**STAGE 0**

Activated objects

- Embankment
- Excavate01
- Excavate01F
- Excavate02
- Excavate03
- Excavate04
- Excavate05
- Excavate06
- Excavate07
- Left
- Back
- Front
- Bot

Boundary type

- Free-slip
- Free-slip
- Free-slip
- No-slip

Time of simulation: 5000

Type of calculation: Elastic

Stress initialization method: Elastic loading

Damping: 0.02

Reset displacements to 0

This panel shows the configuration for 'STAGE 0'. It includes a list of 'Activated objects' with checkboxes for various simulation components. A 'Boundary type' column shows settings for different faces (Left, Back, Front, Bot). Other parameters include 'Time of simulation' (5000), 'Type of calculation' (Elastic), 'Stress initialization method' (Elastic loading), 'Damping' (0.02), and a checked option for 'Reset displacements to 0'.

**Object data tab**

Property	Value
<b>Base</b>	
Placement	[(0.00 0.00 1.00); 0.00 °; (0.00 mm 0.00 mm 0....
Angle	0.00 °
Axis	[0.00 0.00 1.00]
Position	[0.00 mm 0.00 mm 0.00 mm]
x	0.00 mm
y	0.00 mm
z	0.00 mm
Label	Embankment

This panel displays a table of object data for the selected 'Embankment' object. The table lists properties such as 'Placement', 'Angle', 'Axis', 'Position', 'x', 'y', 'z', and 'Label' with their corresponding values.

# GeoXPM INTERFACE – TOOLBAR, MENU AND DIALOGS



GeoXPM Toolbar

Open results in Paraview if available

Export output to Paraview compatible format

Run simulation

Generate particles

Meshing order

Autofit simulation domain

Execution parameters

Materials (constitutive models)

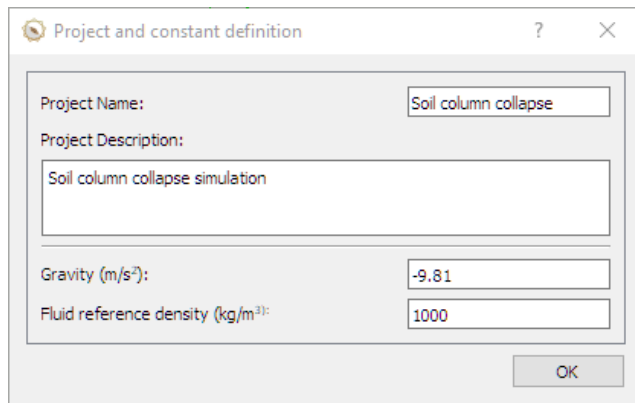
Import external 3D object (terrain)

Add a fillbox (material inlet)

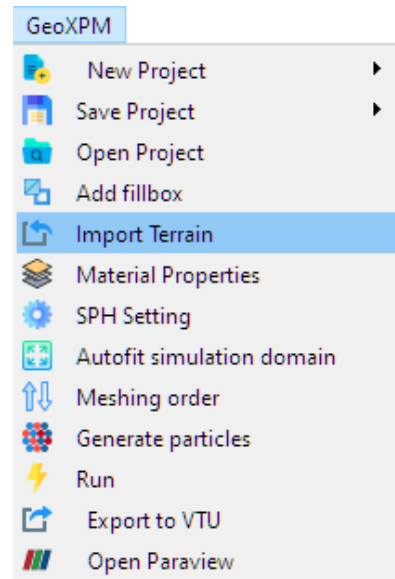
Open projects

Save projects

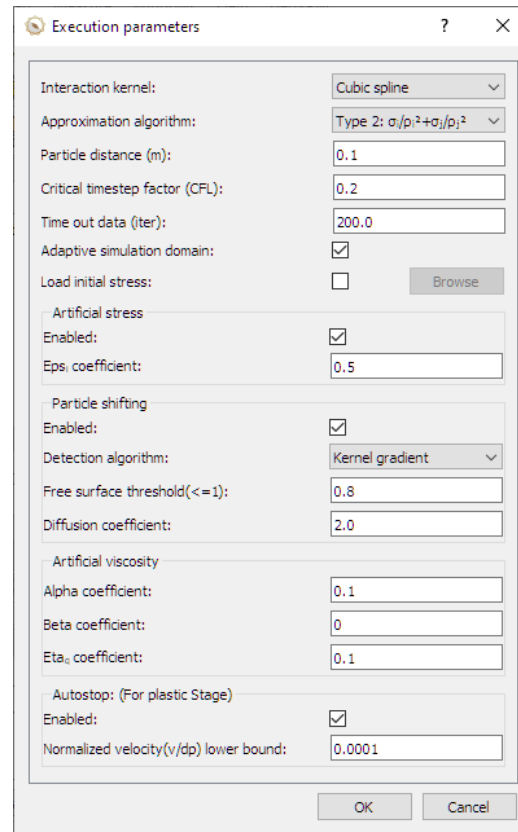
New project



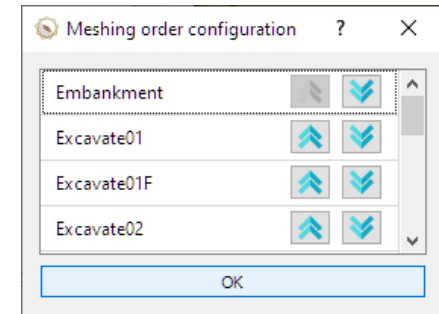
New project dialog



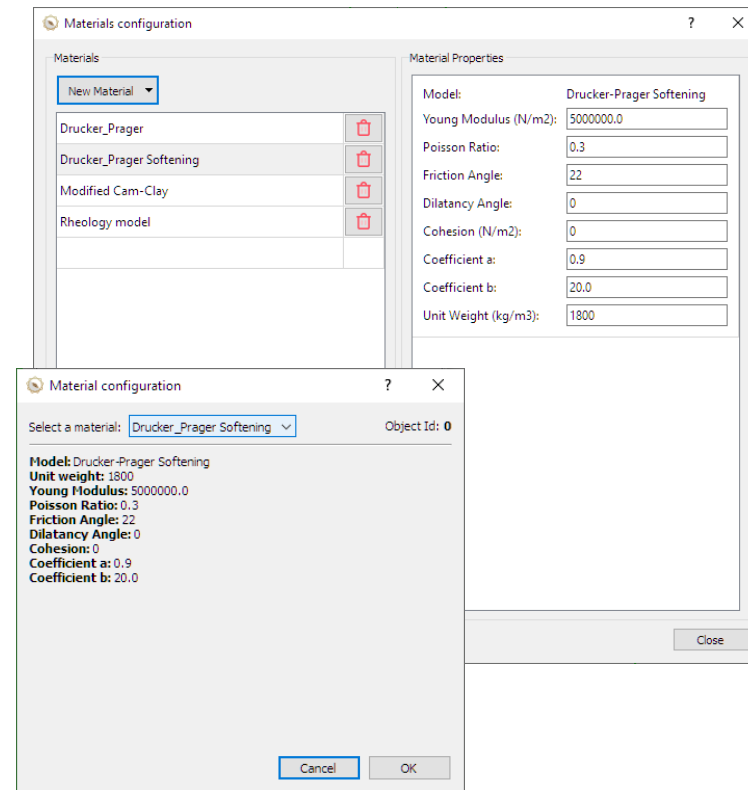
GeoXPM Menu



Execution parameters dialog

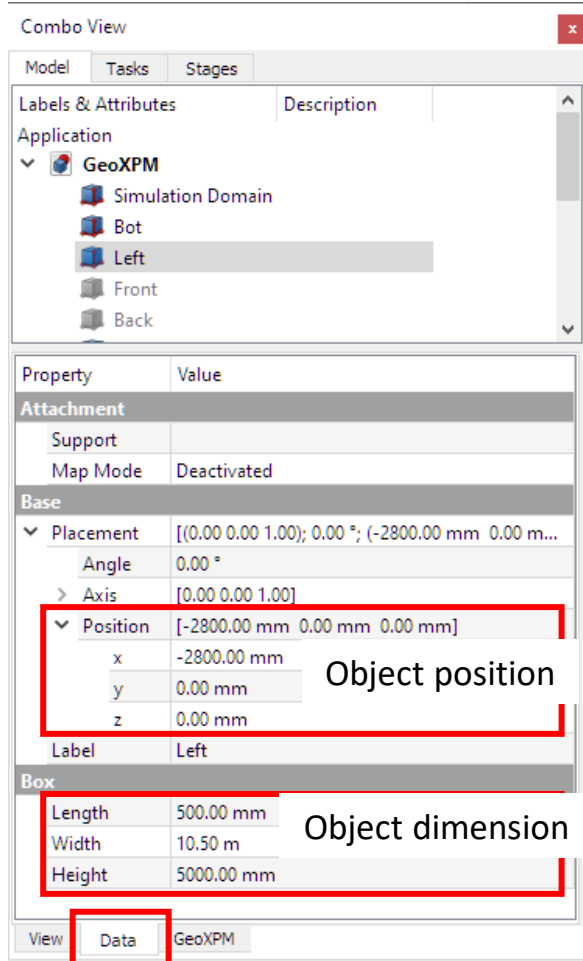


Meshing order dialog

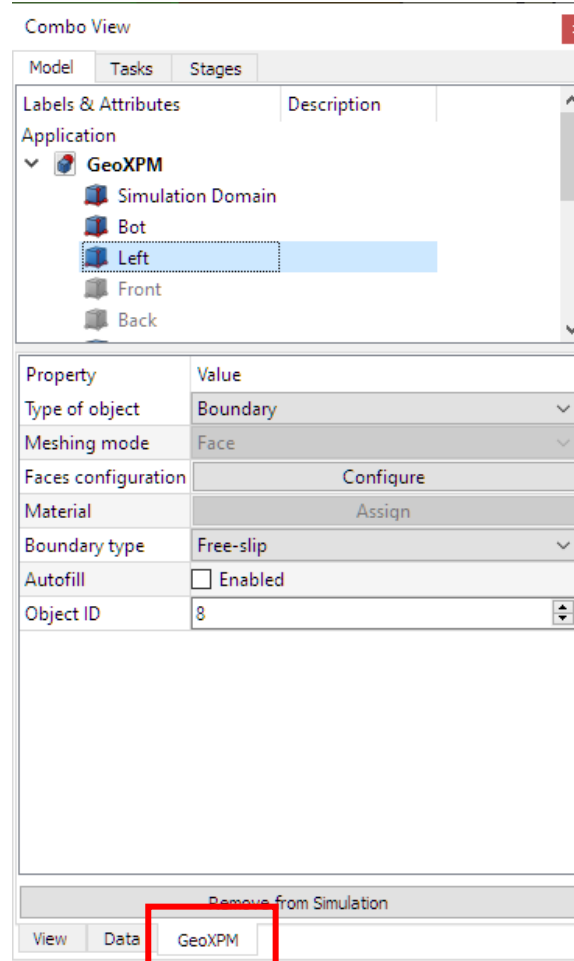


Material dialogs

# GeoXPM INTERFACE – OBJECT PROPERTIES AND STAGES



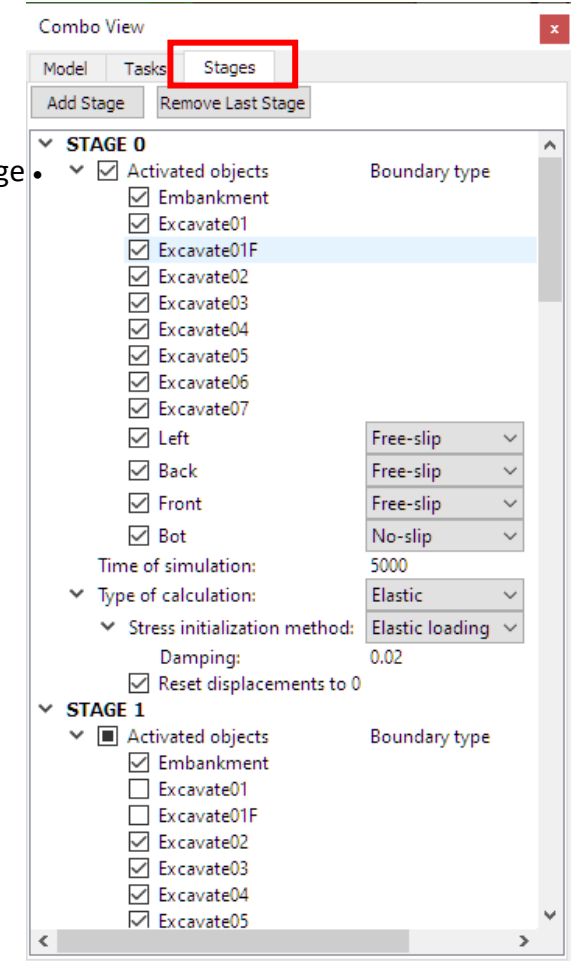
Geometry Data tab



SPH simulation object tab

Ticked objects will be included in stage

- Object type: Soil, boundary.
- Meshing modes: Full, solid, face, wire.
- Faces mode configuration.
- Assign material (model) to a Soil object
- Boundary type: non-slip, free-sip
- Fill the imported object with particles.
- Object(s) ID



Stages construction tab

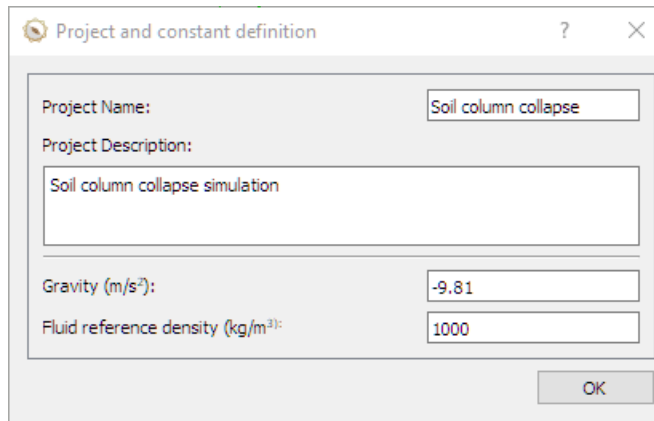
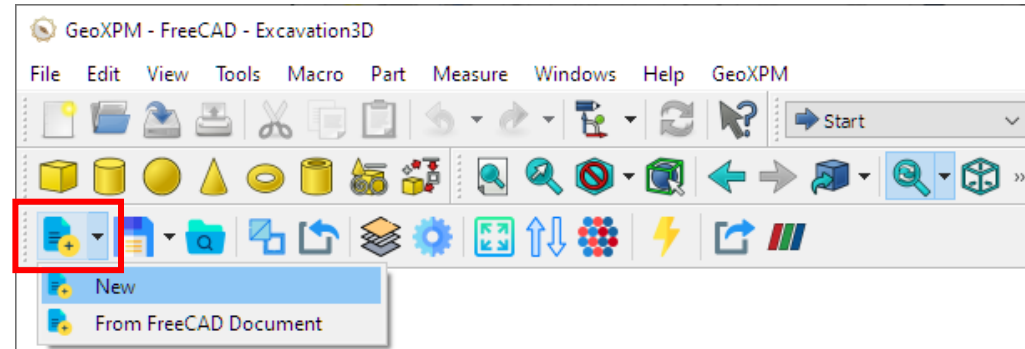
## WORKFLOW

---

1. CREATE A PROJECT.
2. CREATE INITIAL GEOMETRY: 3 available approaches.
  - Primitive predefined objects: Cube, cylinder, sphere, cone ...
  - Sketch: Drawing complex parametric objects.
  - External Import (solids/meshes).
3. INPUT **EXECUTION PARAMETERS**.
4. **CREATE MATERIAL MODELS**.
5. **INCLUDE** OBJECTS IN SIMULATIONS AND **ASSIGN MATERIAL** PROPERTIES.
6. DEFINE **CONSTRUCTION STAGES**: elastic/plastic loading, activated objects, number of time steps.
7. DEFINE **MESHING ORDER**.
8. MESH OBJECTS – GENERATE PARTICLES FOR CALCULATIONS.
9. **CALCULATE**
10. **VISUALIZE RESULTS**: data extraction, data mining

# WORKFLOW – 1. CREATE A PROJECT

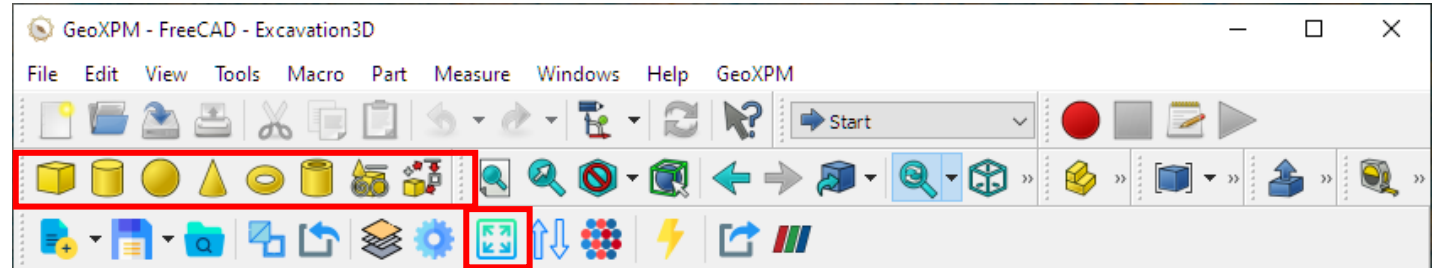
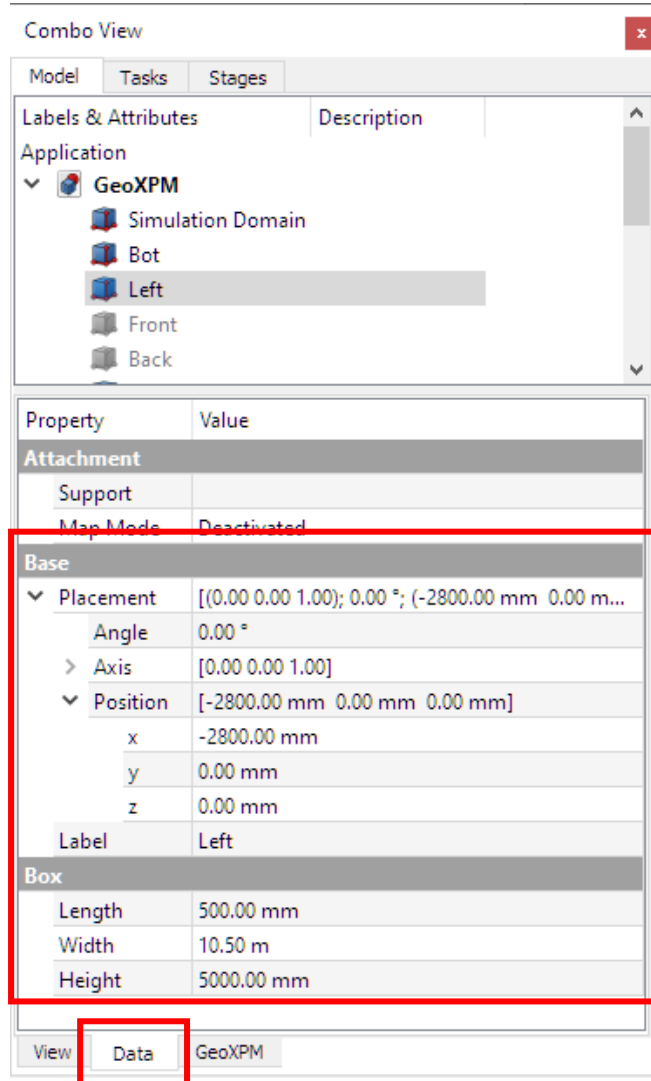
1. Use “New” tool to create a new project.
2. Enter the project’s name, descriptions and basic constants.



New project dialog

## WORKFLOW – 2. CREATE INITIAL GEOMETRY

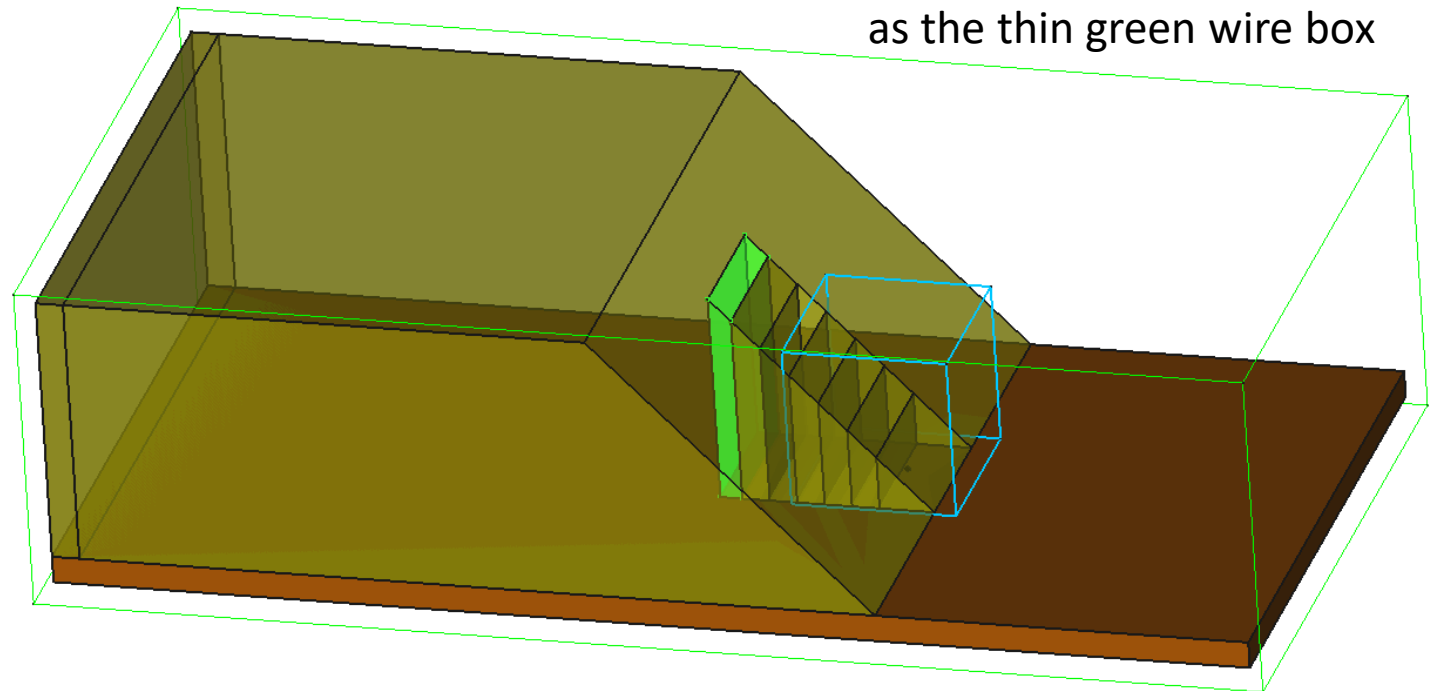
1. Use “Solids” toolbar and “Data” tab to draw soil/boundary geometries
2. Use “Autofit” tool to fit the simulation domain (objects outside of this domain will not be considered in simulations)



Solids toolbar

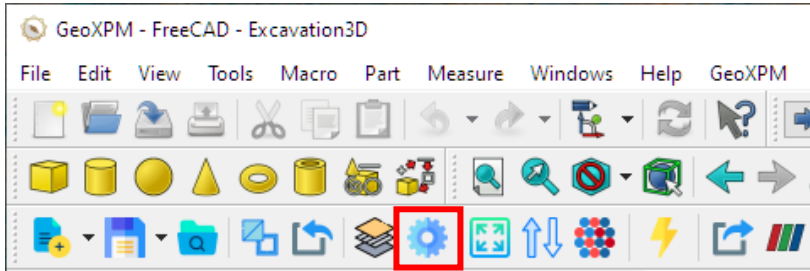
“Autofit” tool

Simulation domain is shown as the thin green wire box

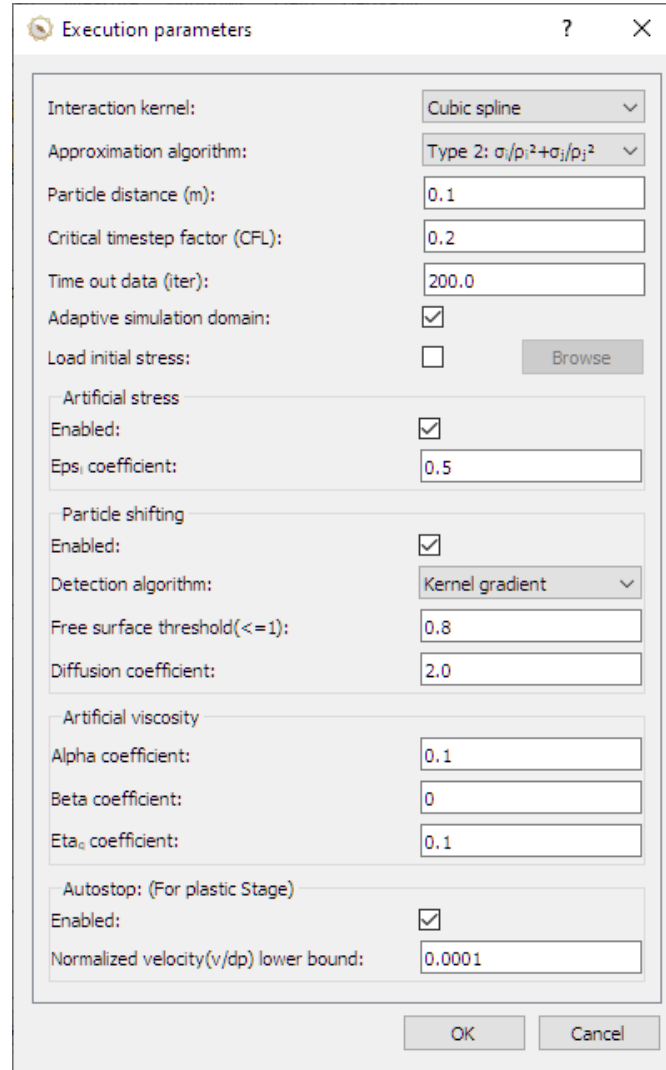


# WORKFLOW – 3. INPUT EXECUTION PARAMETERS

1. Use “Execution parameters” tool to input required parameters



“Execution parameters” tool

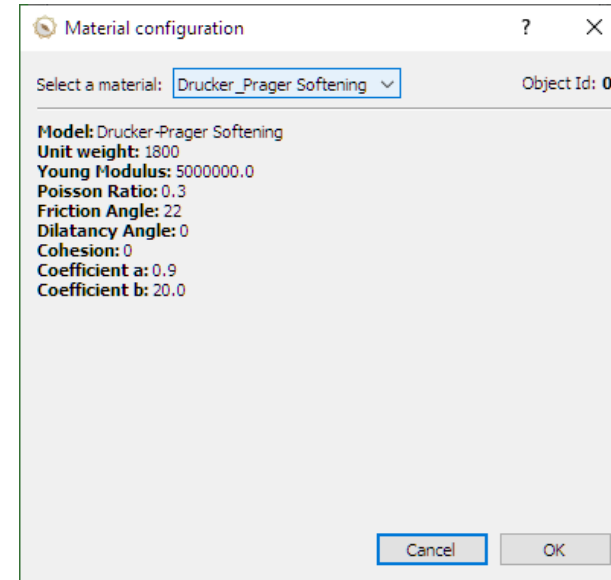
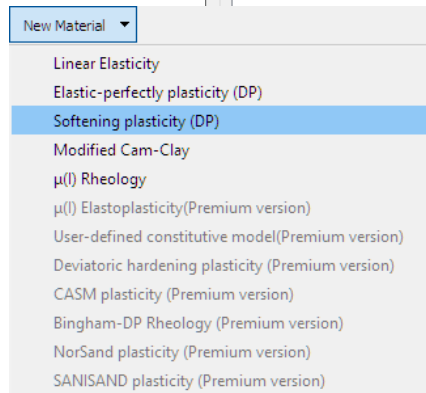
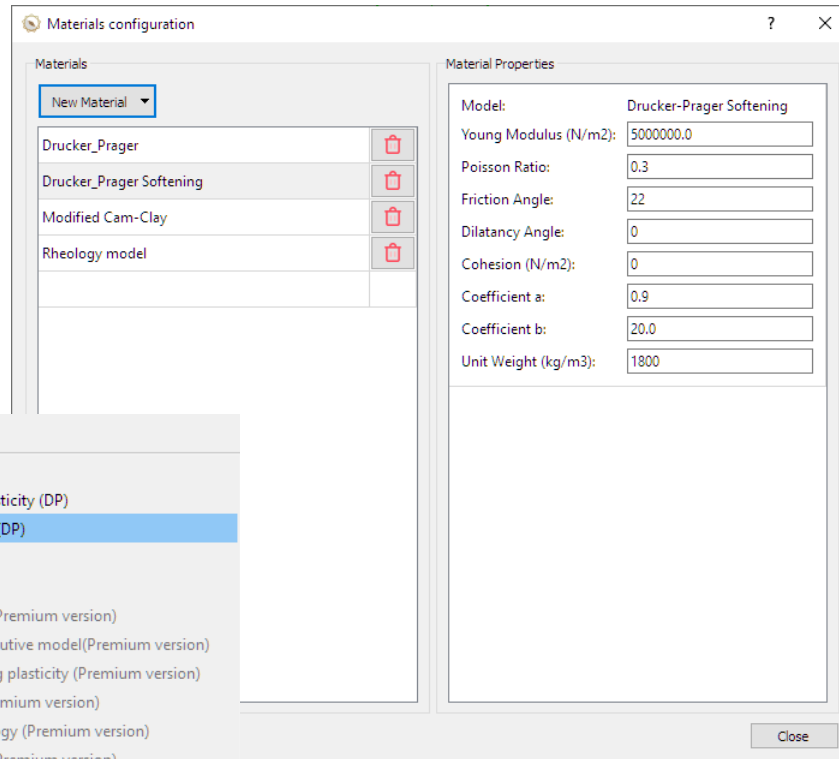
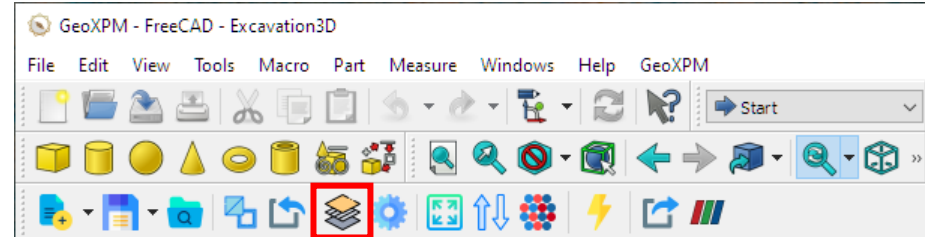


Execution parameters dialog

- Kernel functions: Cubic spline, Gauss, Wendland C2
- Approximation algorithm:
  - Type 1: guarantees vanishing gradient for a constant field function.
  - Type 2: pairwise-symmetry algorithm
- If enabled, The simulation domain expands with the soil domain and is limited by the “Simulation domain” object
- Artificial stress for tensile instability treatment
- Shifting criteria based on: Kernel/ Kernel gradient
- Threshold for free-surface detection
- Artificial viscosity for damping system’s fluctuations.
- Recommended Alpha = 0.1 for 2D and 1.0 for 3D
- Autostop condition for plastic stages. If the fastest particle has a normalized velocity lower than the set lower bound. The stage is completed.

# WORKFLOW – 4. CREATE MATERIAL

1. Click the Material Property button, and define the material properties using your desired constitutive model.



Material dialogs



## WORKFLOW – 5. INCLUD OBJECTS IN SIMULATIONS AND ASSIGN MATERIAL PROPERTIES

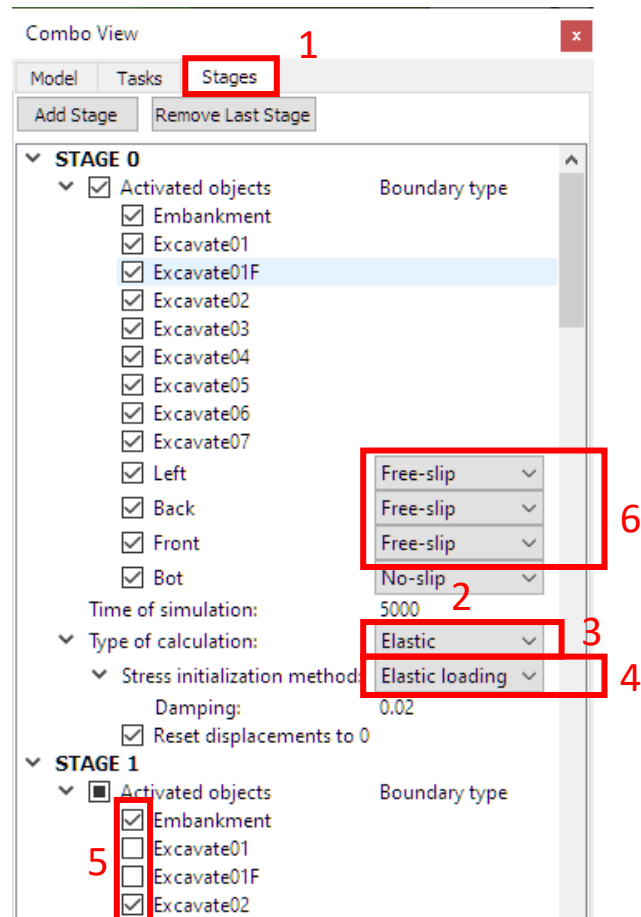
1. Choose the object you want to add to the simulation in the “Object Tree” and press “Add to Simulation” in “GeoXPM” tab.
2. Select “Type of object”, and the “Meshing mode” in the “GeoXPM” tab.
3. Choose/Assign appropriate parameters depending on the selected type of object.

The image displays a software interface with two main windows and a dialog box. The left window, titled "Combo View", shows a tree structure under "GeoXPM" with "sensitiveClay" selected (1). Below the tree is an "Add to Simulation" button (3). The right window, also titled "Combo View", shows the same tree structure with "SensitiveClay" selected. Below it is a table of properties (4) with "Soil" selected for "Type of object" (4) and "Full" for "Meshing mode" (5). The "Material" property has an "Assign" button (6). The "Object ID" is 0. At the bottom of this window is a "Remove from Simulation" button. The rightmost window is a "Material configuration" dialog box (7) with "SensitiveClay" selected in the "Select a material" dropdown. It lists material properties: Model: Drucker-Prager Softening, Unit weight: 2150.0, Young Modulus: 25000000.0, Poisson Ratio: 0.485, Friction Angle: 0.0, Dilatancy Angle: 0.0, Cohesion: 25000.0, Coefficient a: 0.95, Coefficient b: 20.0. The dialog has "Cancel" and "OK" buttons.

Property	Value
Type of object	Soil
Meshing mode	Full
Faces configuration	Configure
Material	Assign
Boundary type	No-slip
Autofill	<input type="checkbox"/> Enabled
Object ID	0

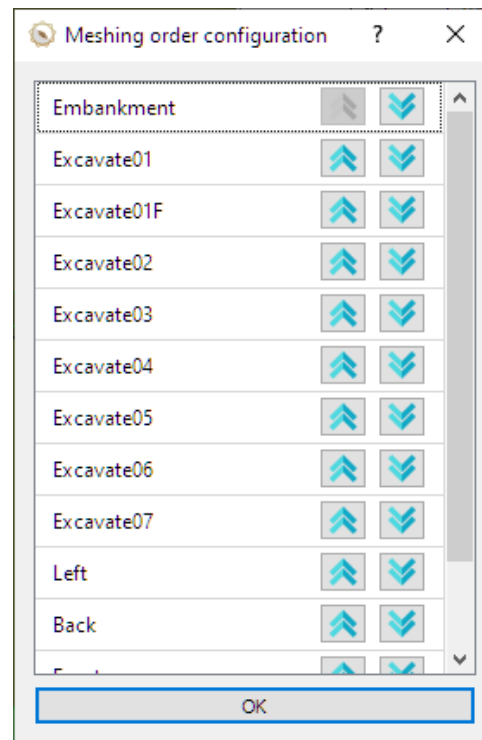
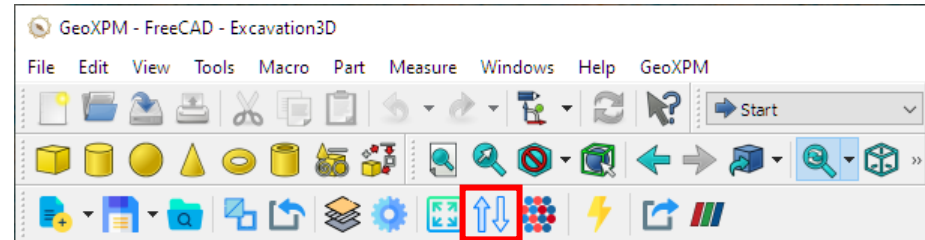
## WORKFLOW – 6. DEFINE CONSTRUCTION STAGES

1. Go to “Stage” tab and Add or remove the required stages.
2. Input the number of time steps to run for each stage by double-clicking the number next to “Time of simulation”.
3. Define the type of calculation for the stage, either Elastic or Plastic.
4. If the first stage (Stage 0) is Elastic, 2 methods of stress initialization will be available: K0 or Elastic loading.
5. Tick or untick an object to activate or deactivate that object in each stage.
6. Boundary type can also be changed for initial Free-slip boundaries if necessary.



## WORKFLOW – 7. DEFINE MESHING ORDER

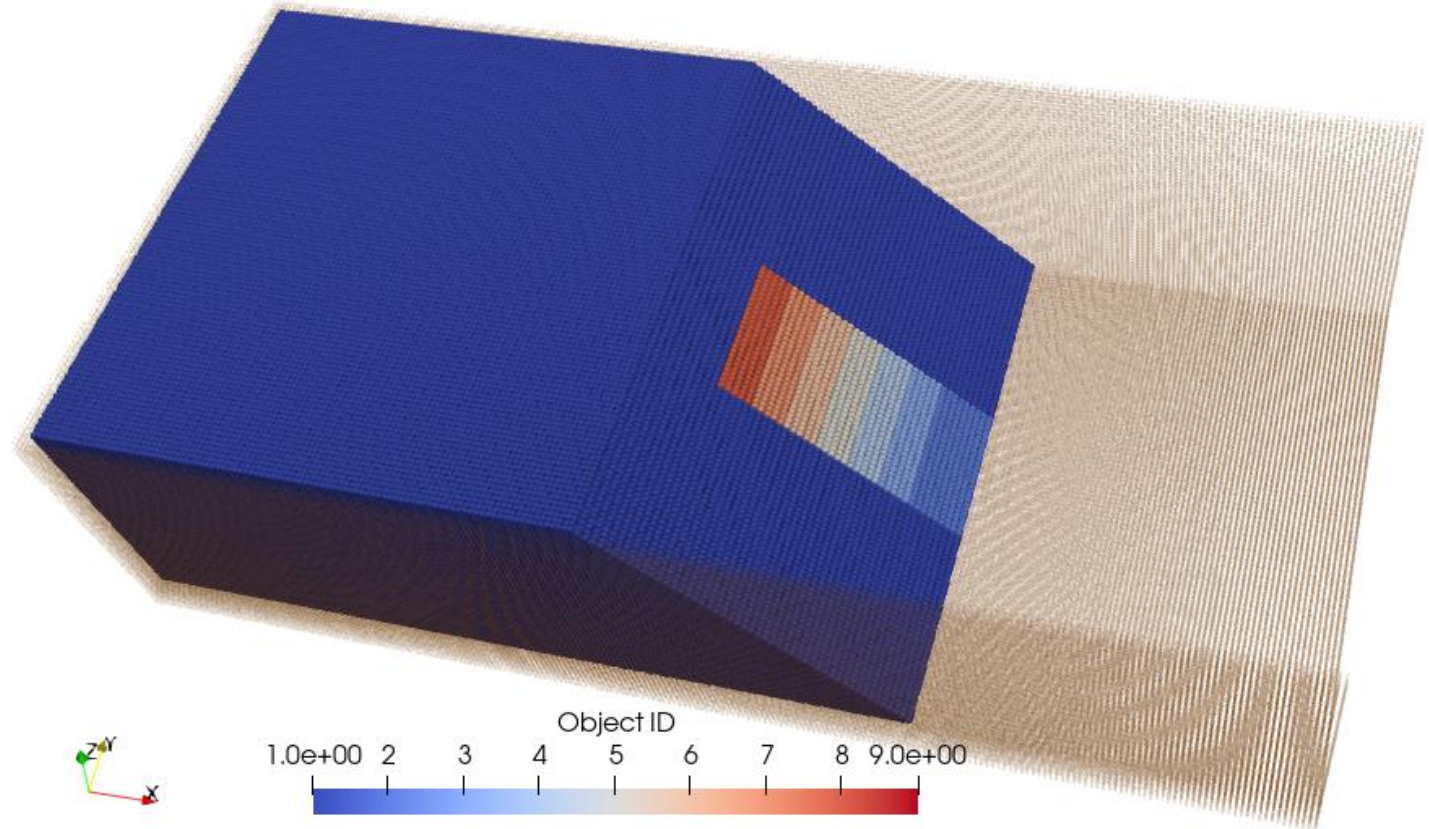
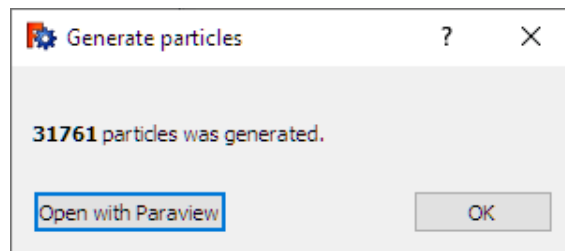
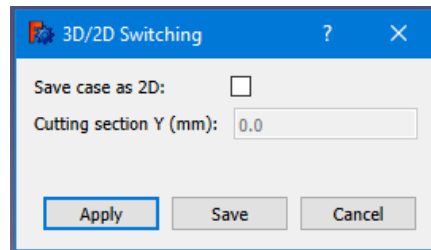
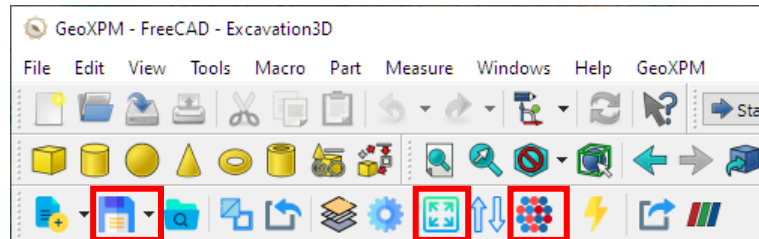
1. Click the meshing order button, and change the meshing sequence for the included objects in step 5. Note that the lower object will overwrite the upper one if there are overlaps.



Meshing order dialog

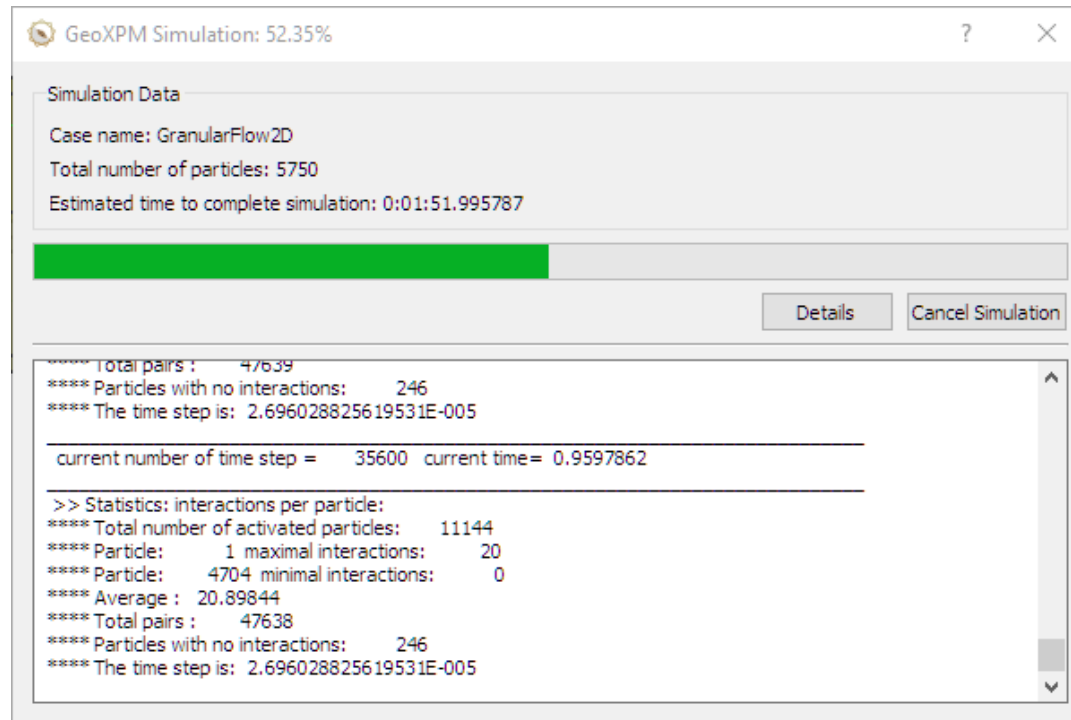
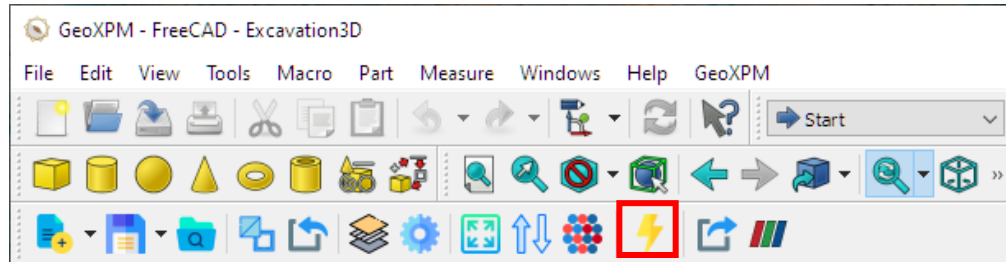
## WORKFLOW – 8. GENERATE PARTICLES FOR CALCULATION

1. Before meshing, click the “Save” button to save the project as 2D or 3D based on your need.
2. Click the “Autofit simulation domain” if needed to fit all simulated objects in the Simulation domain.
3. Click the “Generate particles” button to mesh the computational domain.
4. After meshing, click “Open with Paraview” to view the particle configuration.



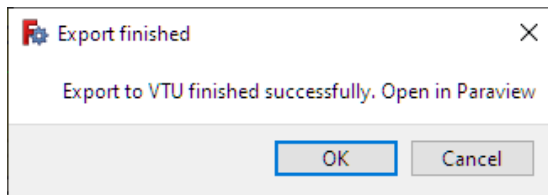
# WORKFLOW – 9. CALCULATE

1. Click the Calculation button to start the calculations.



# WORKFLOW – 10. VISUALIZE RESULTS

1. You can click on the tool “Export output data to VTU format” to start exporting data to Paraview any time during the simulation or after completion.
2. When the export is done, click “OK” to open Paraview with already imported results. All the data can be customized and visualized from here. To reopen the exported data without re-exporting, use the “Open outputs in Paraview” button.



Export finished dialog

Note that if you export a second time, without re-meshing, a dialog will appear asking if you want to overwrite previous exports. Click “No” if you want to continue exporting new data only.

