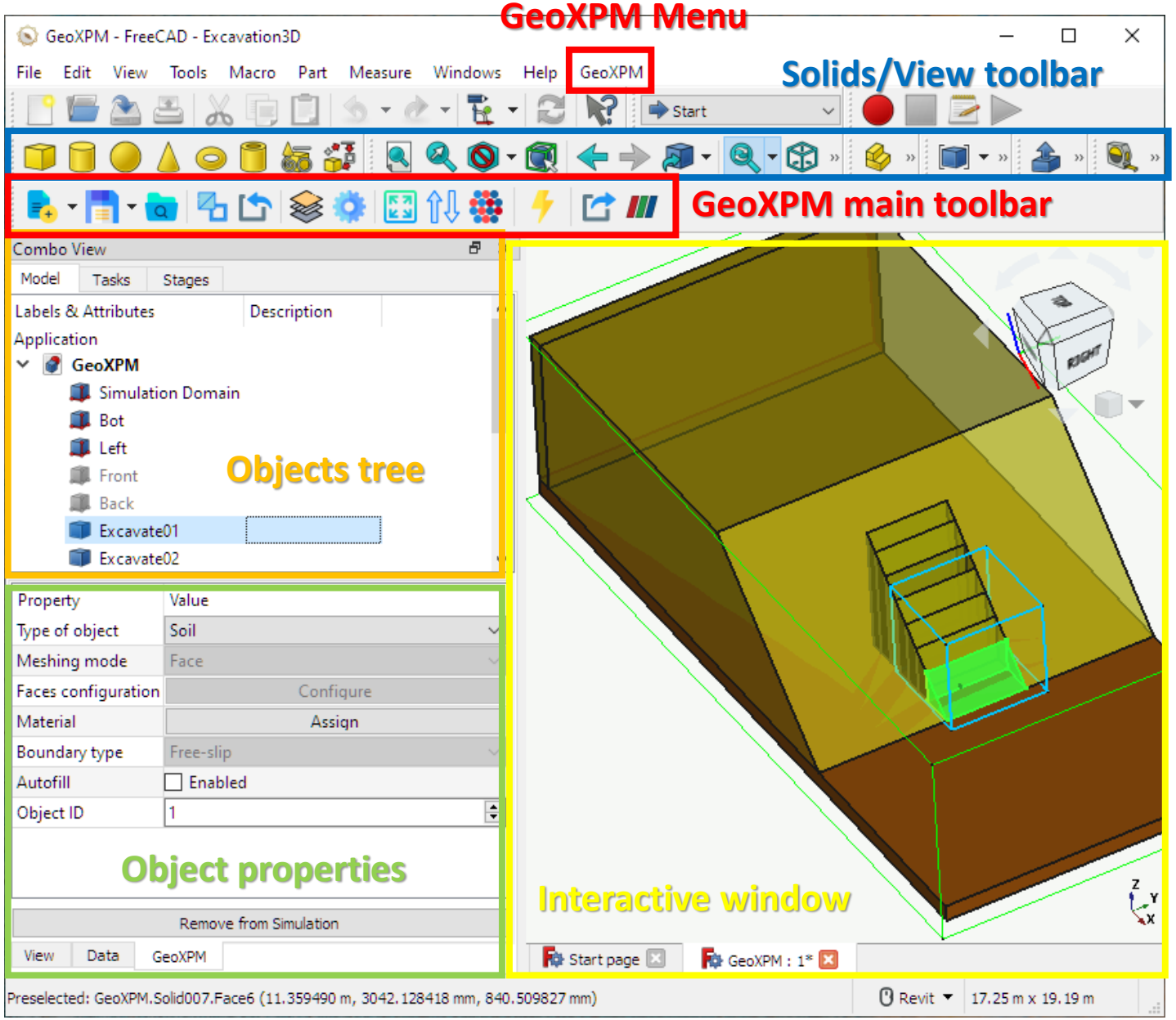
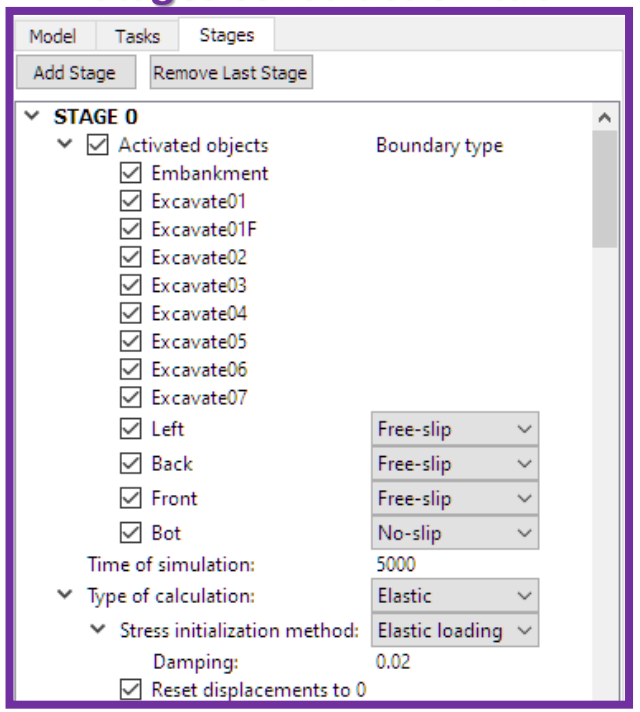
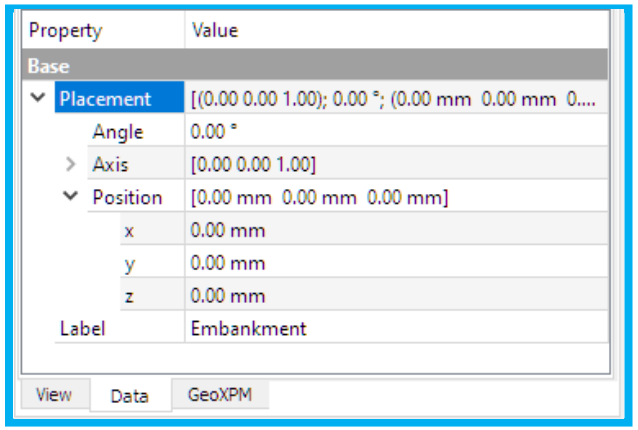
GeoXPM INTERFACE - GENERAL



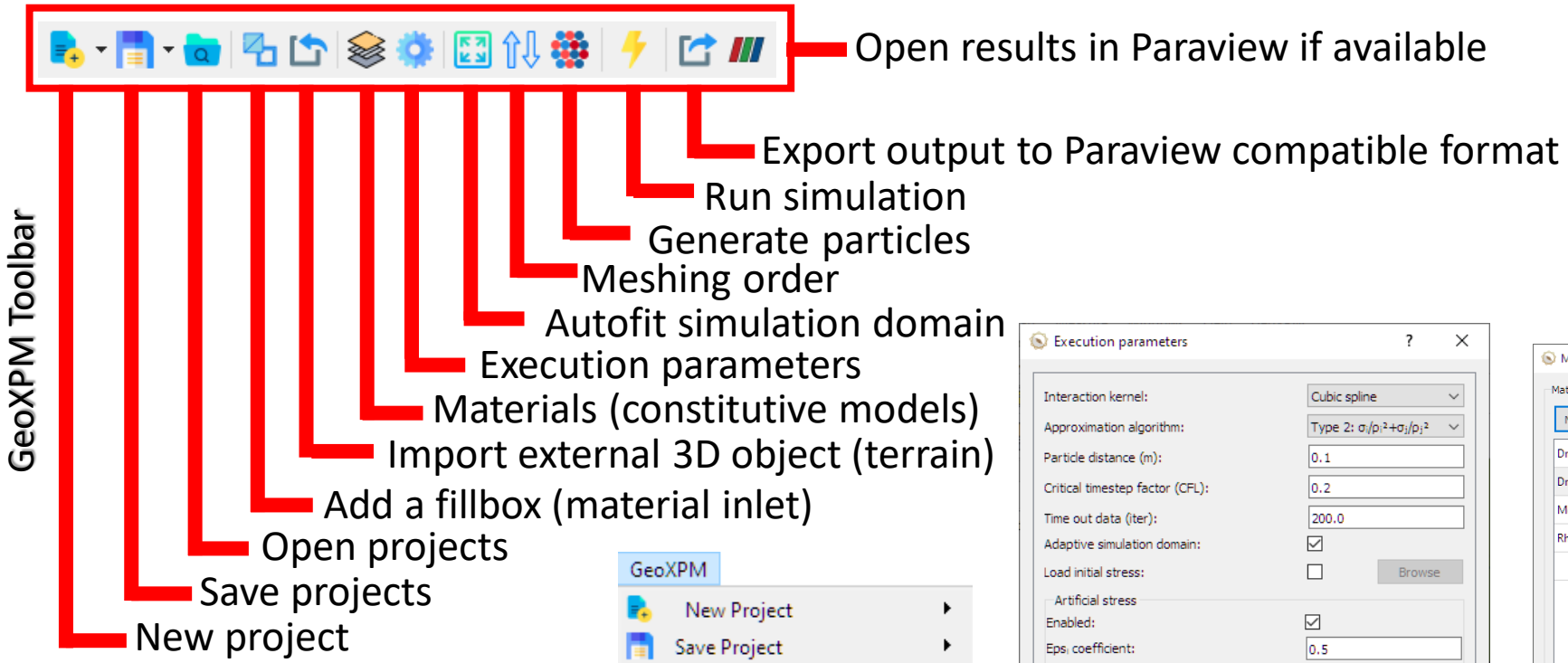
Stages construction tab

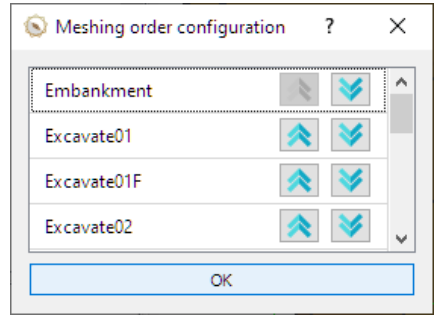




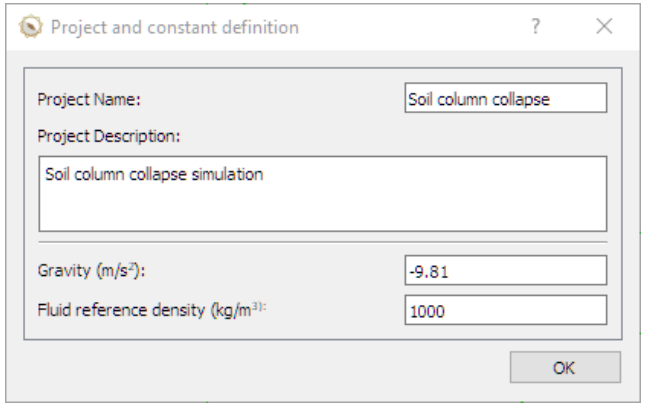
Object data tab

GeoXPM INTERFACE – TOOLBAR, MENU AND DIALOGS

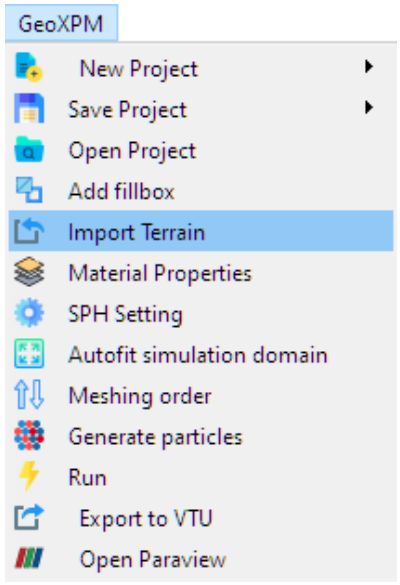




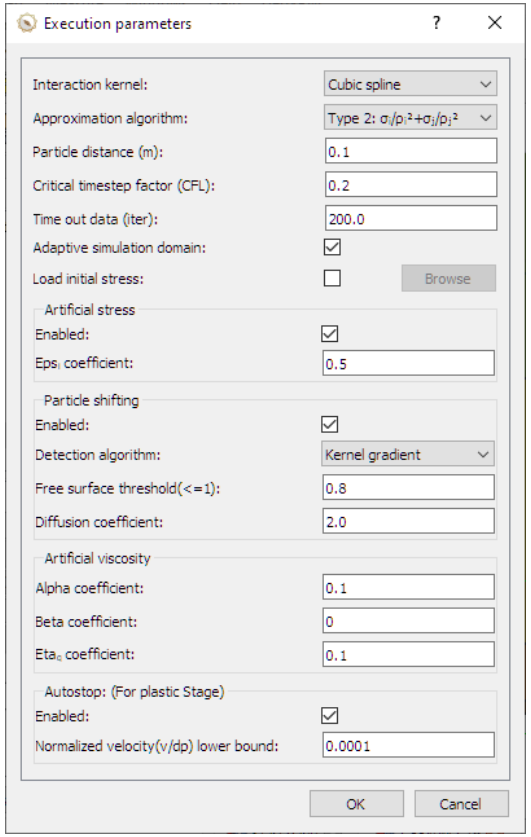
Meshing order dialog



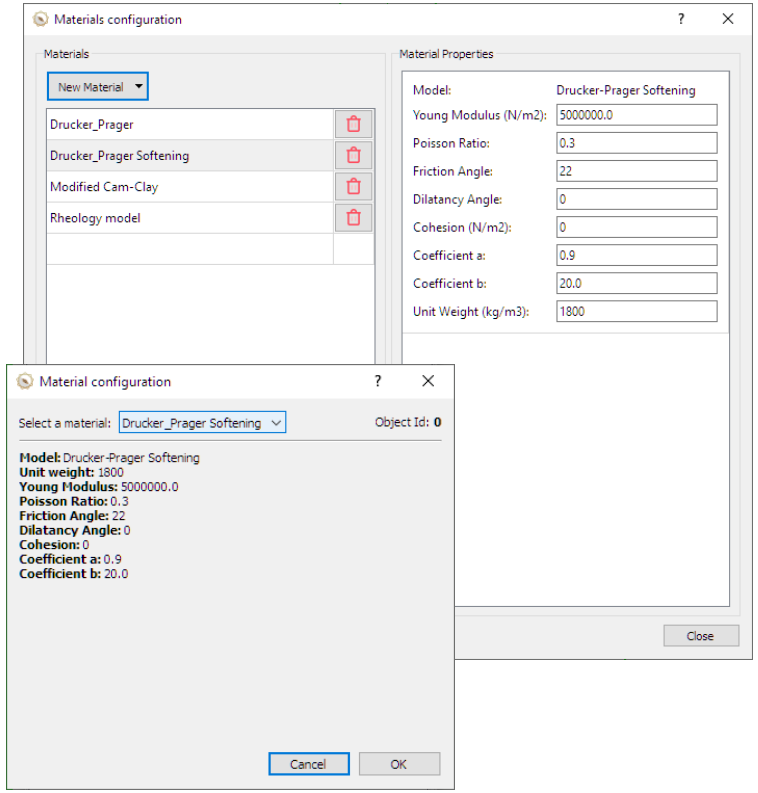
New project dialog



GeoXPM Menu

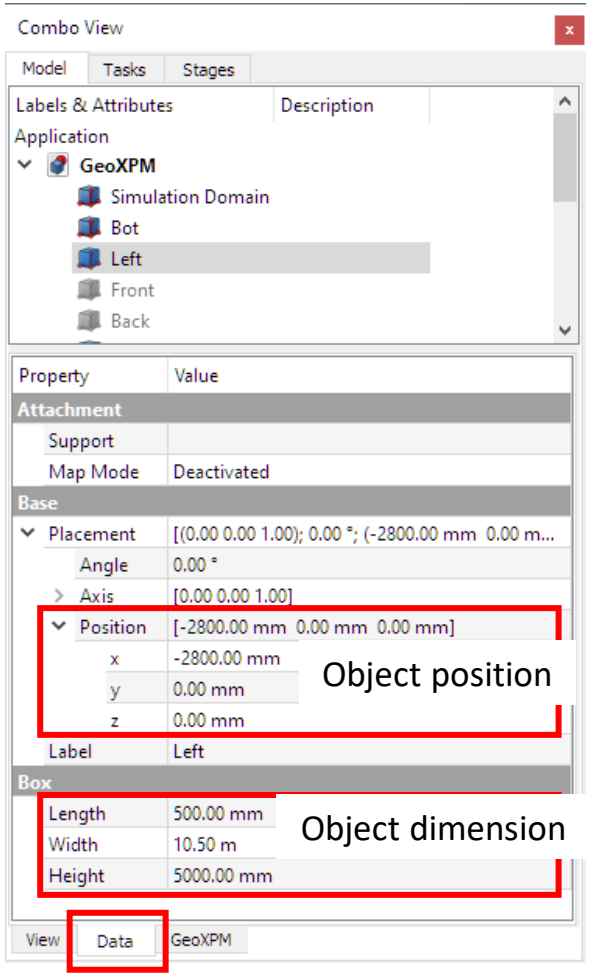


Execution parameters dialog

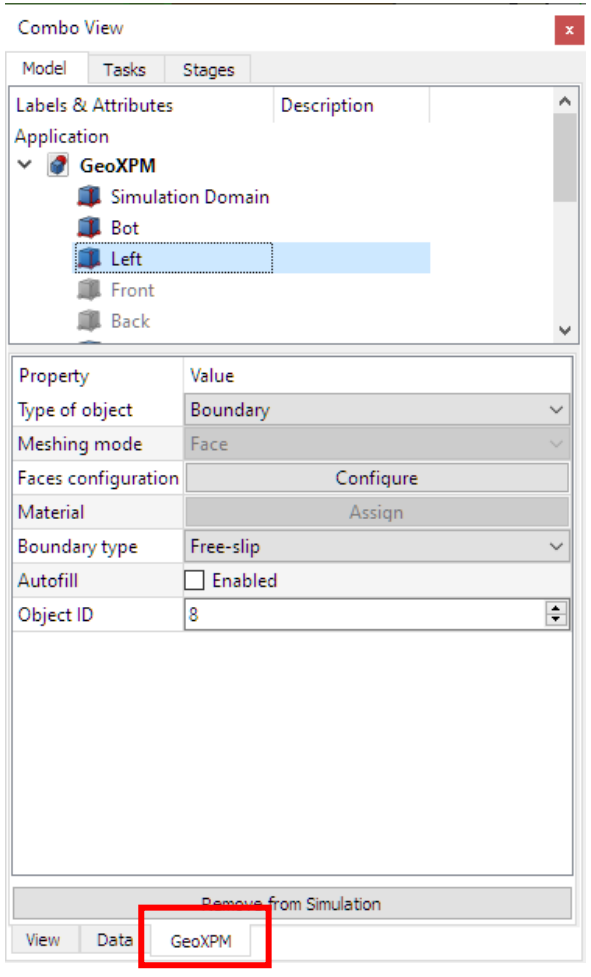


Material dialogs

GeoXPM INTERFACE – OBJECT PROPERTIES AND STAGES



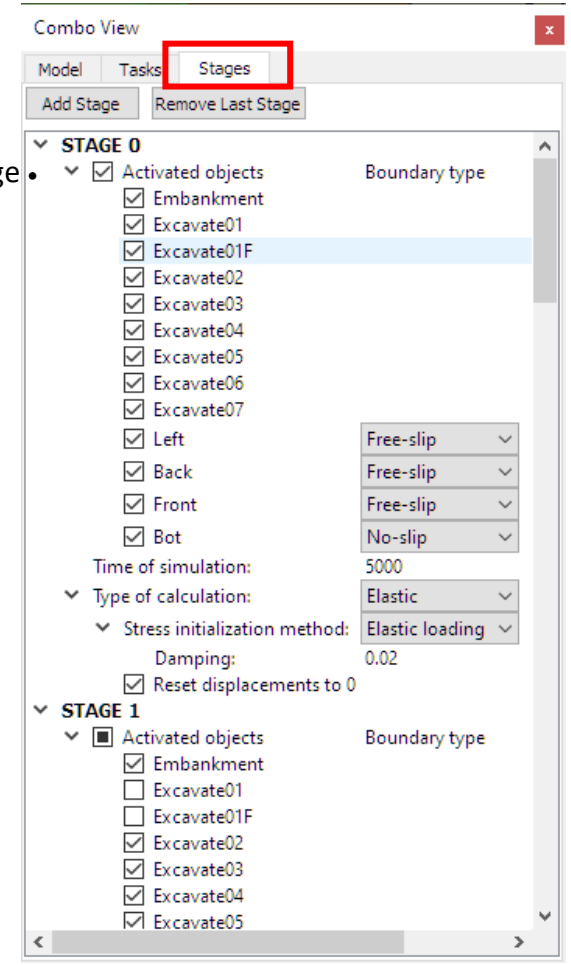
Geometry Data tab



SPH simulation object tab

- 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

Ticked objects will be included in stage



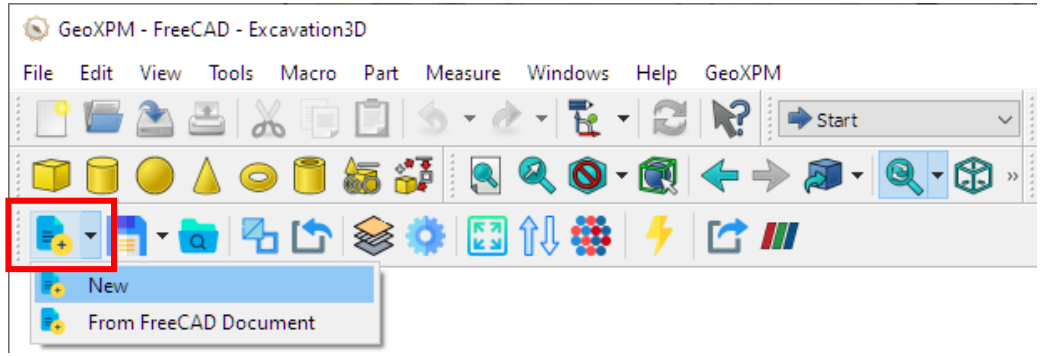
Stages construction tab

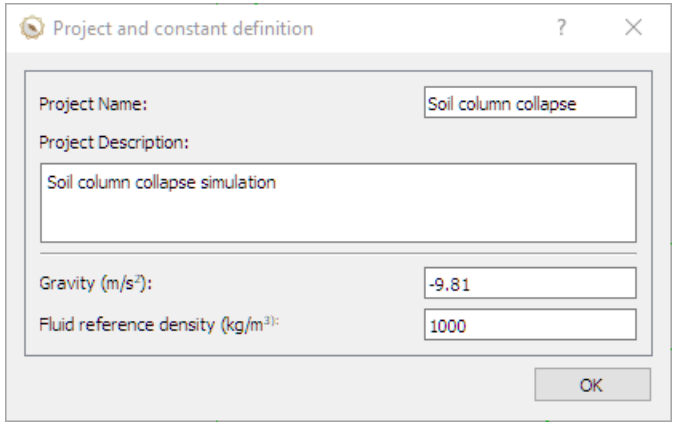
WORKFLOW

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.

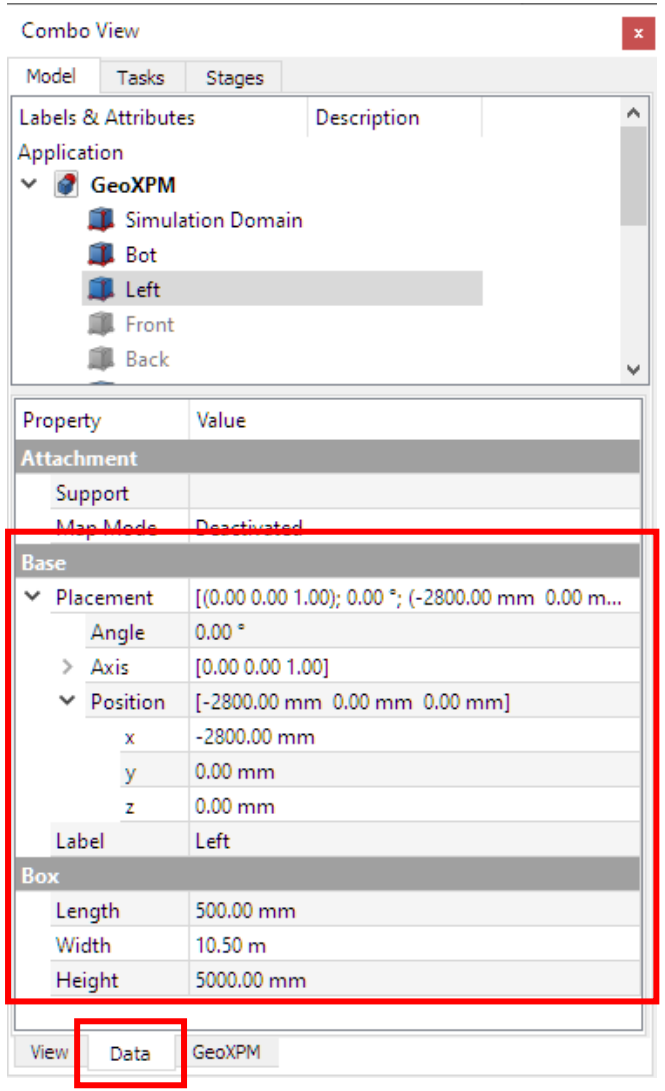


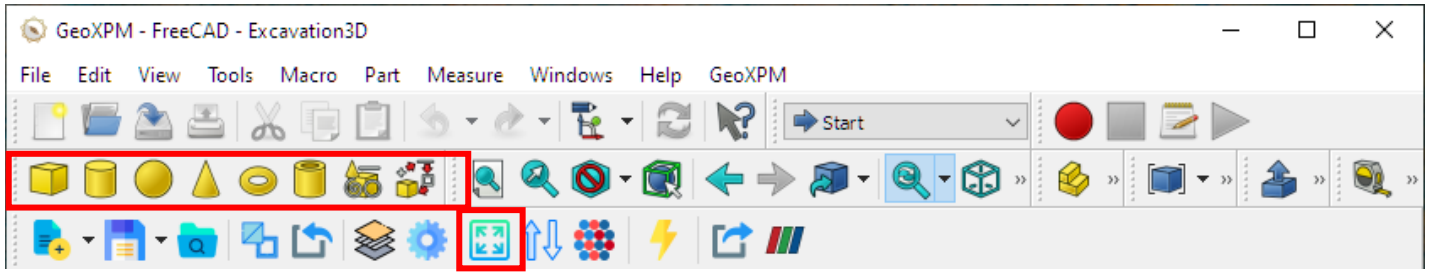


New project dialog

WORKFLOW – 2. CREATE INITIAL GEOMETRY

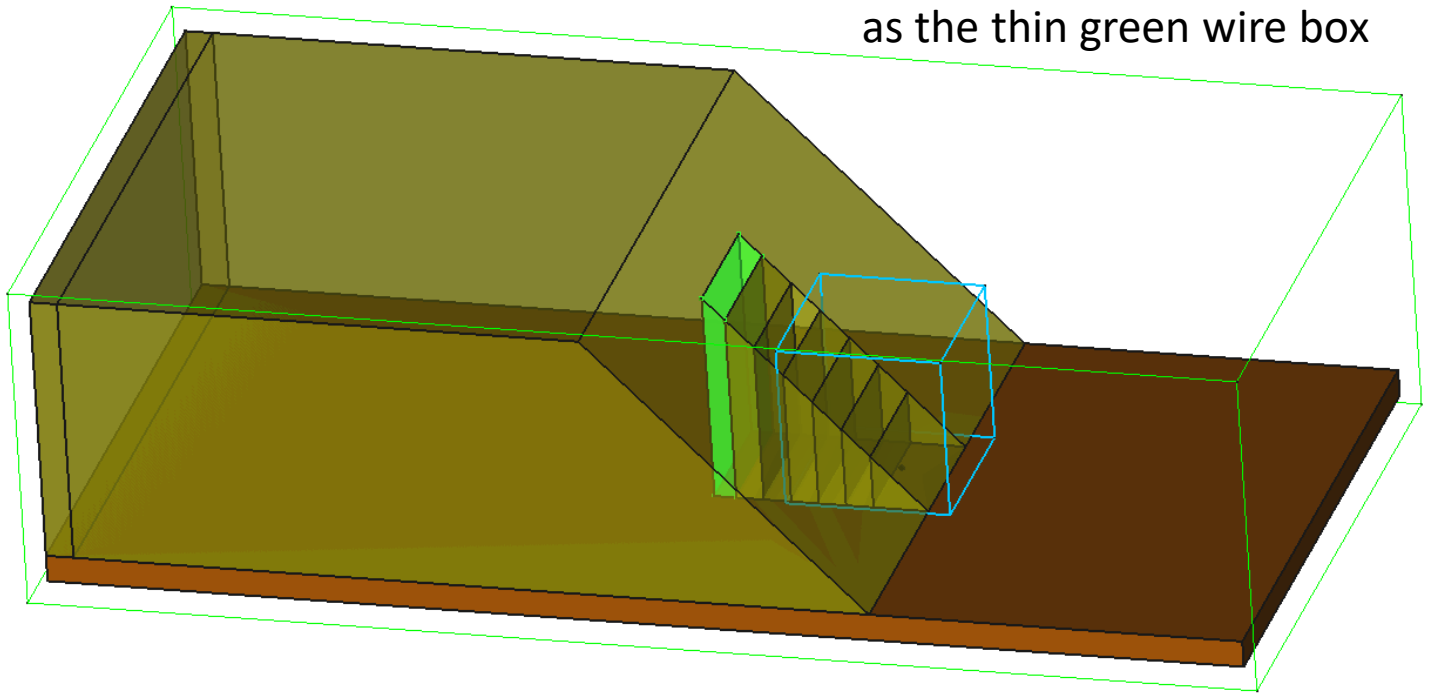
1. Use “Solids” toolbar and “Data” tab to draw soil/boundary geometries
2. Use “Autofit” tool to fit the simulation domain (objects outside of this domain will not be considered in simulations)





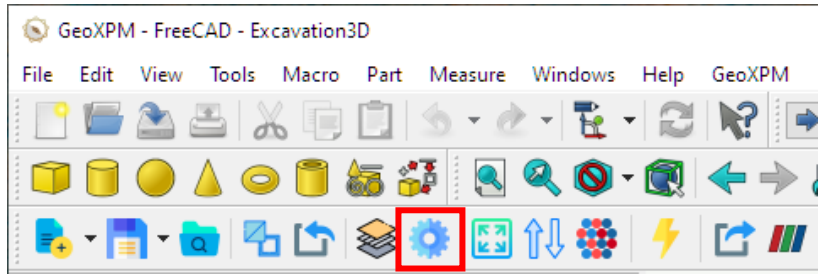
Solids toolbar “Autofit” tool

Simulation domain is shown as the thin green wire box

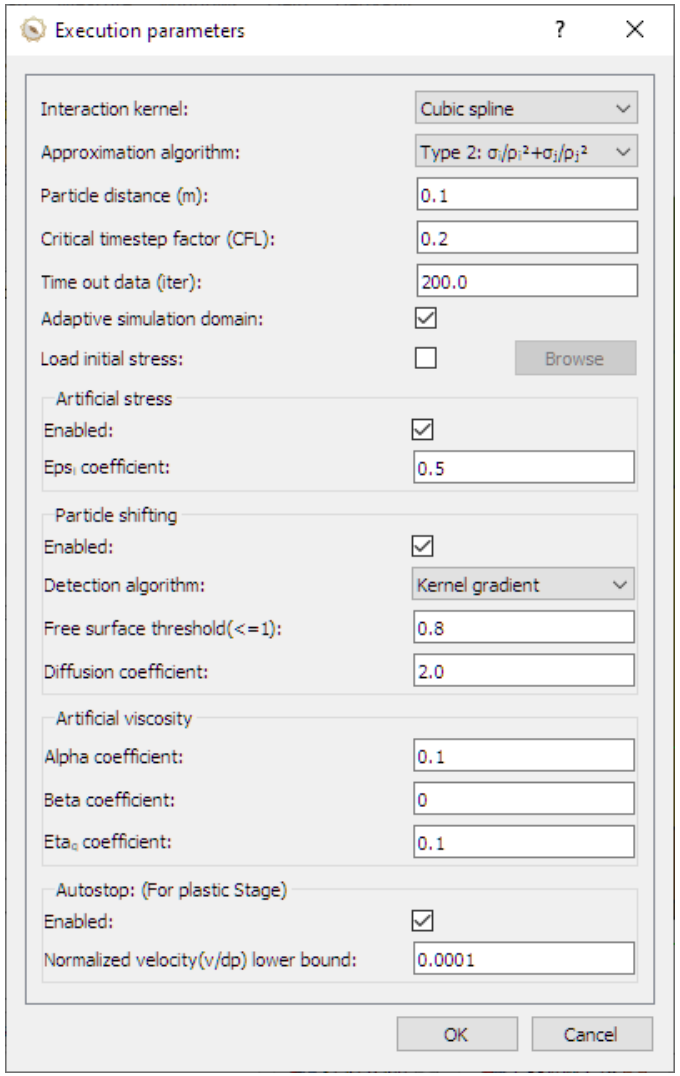


WORKFLOW – 3. INPUT EXECUTION PARAMETERS

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



“Execution parameters” tool

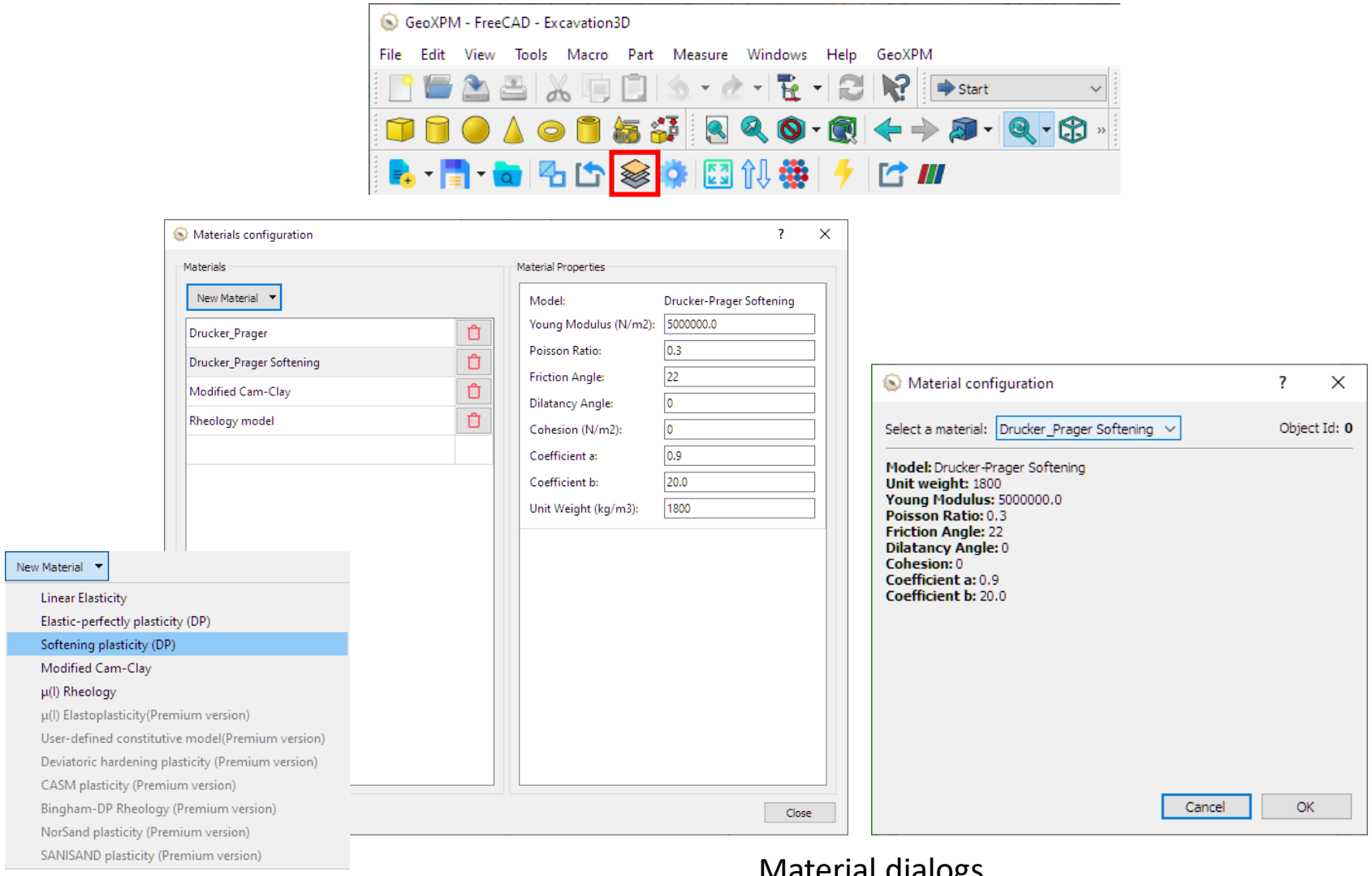


Execution parameters dialog

- 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

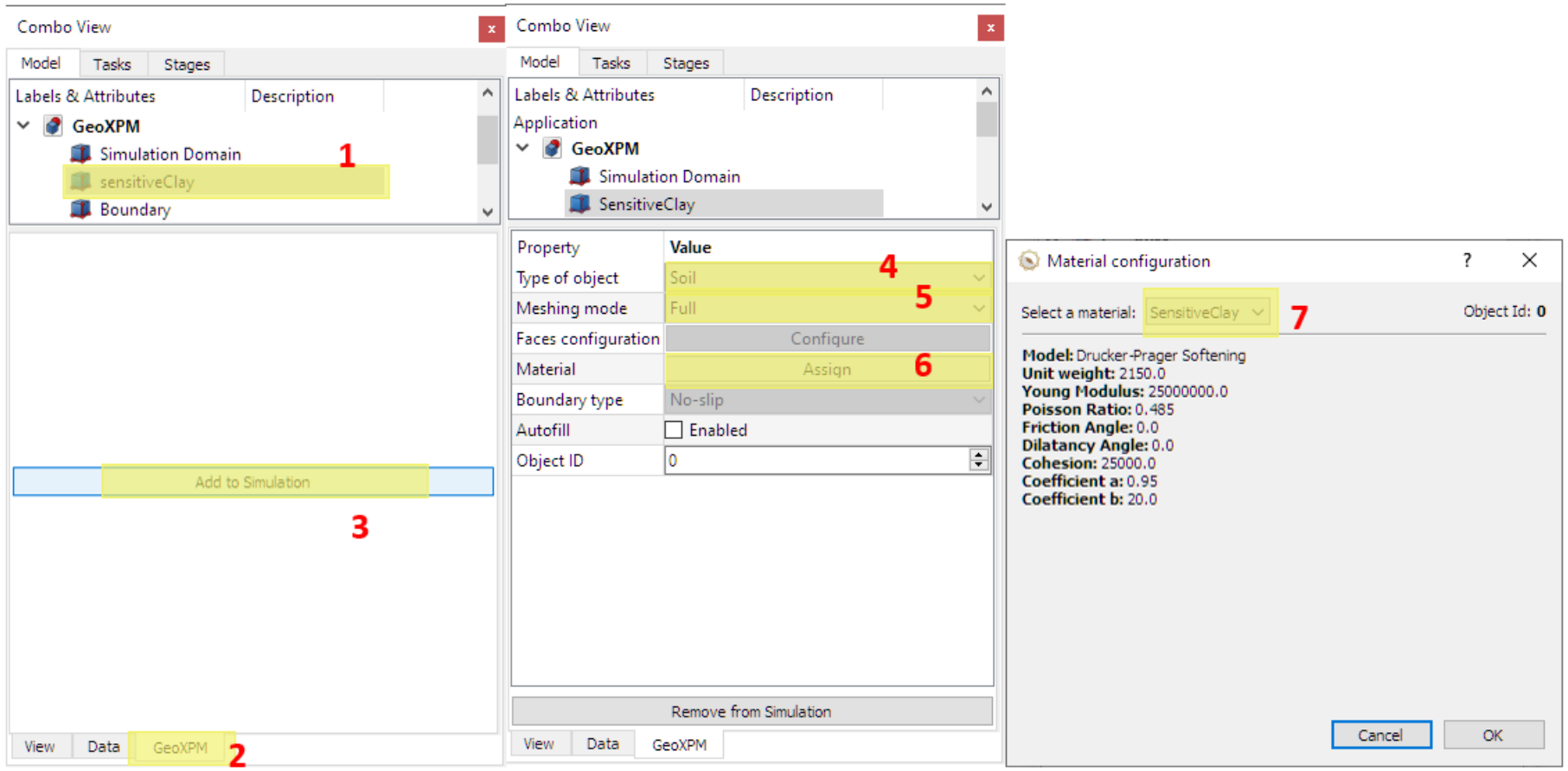
1. Click the Material Property button, and define the material properties using your desired constitutive model.



Material dialogs

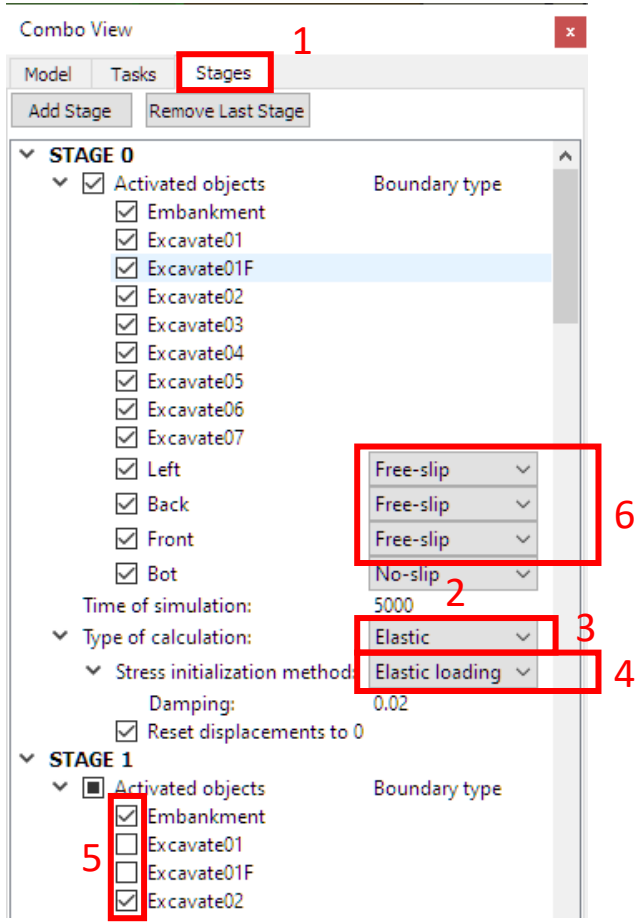
WORKFLOW – 5. INCLUD OBJECTS IN SIMULATIONS AND ASSIGN MATERIAL PROPERTIES

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.



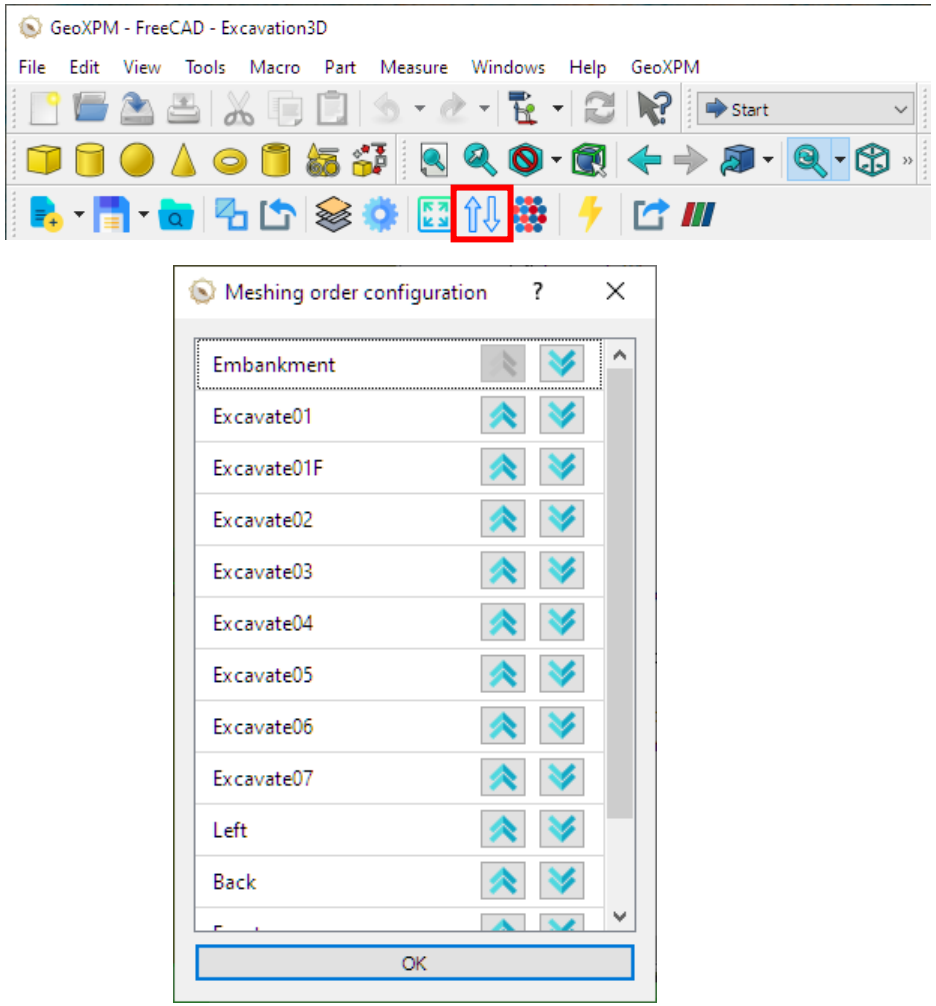
WORKFLOW – 6. DEFINE CONSTRUCTION STAGES

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

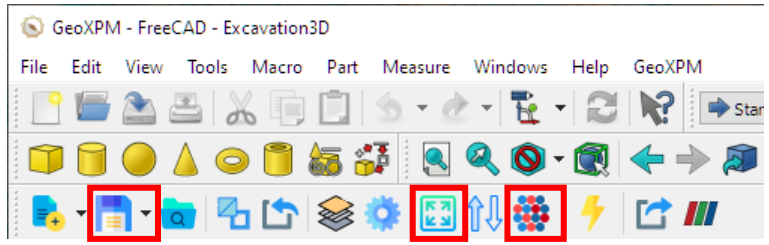
1. Click the meshing order button, and change the meshing sequence for the included objects in step 5. Note that the lower object will overwrite the upper one if there are overlaps.

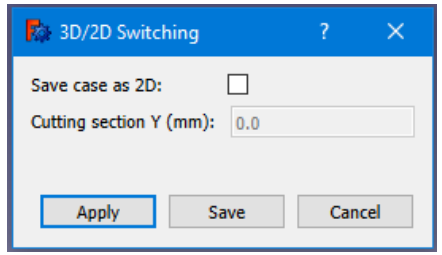


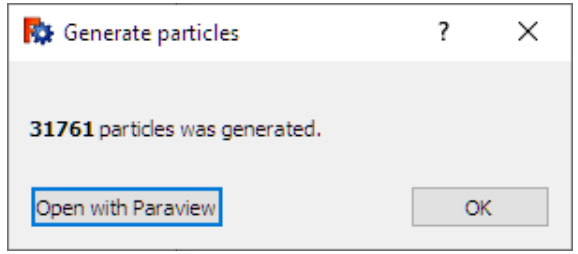
Meshing order dialog

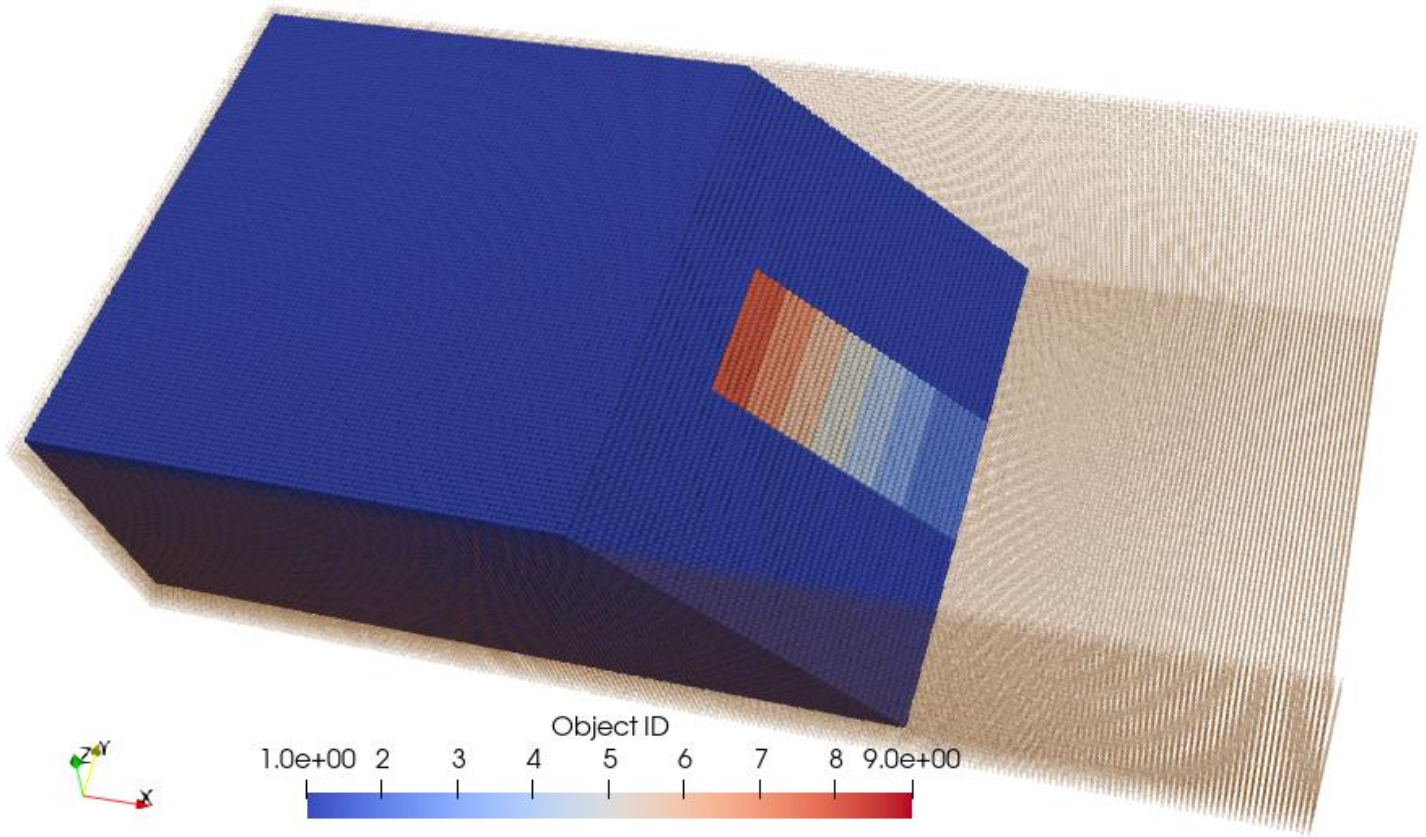
WORKFLOW – 8. GENERATE PARTICLES FOR CALCULATION

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.



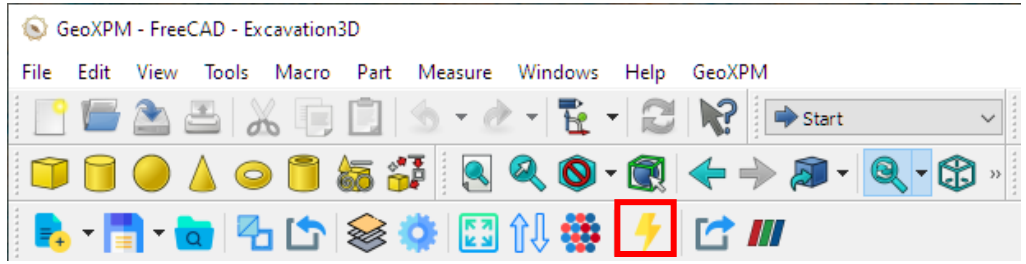


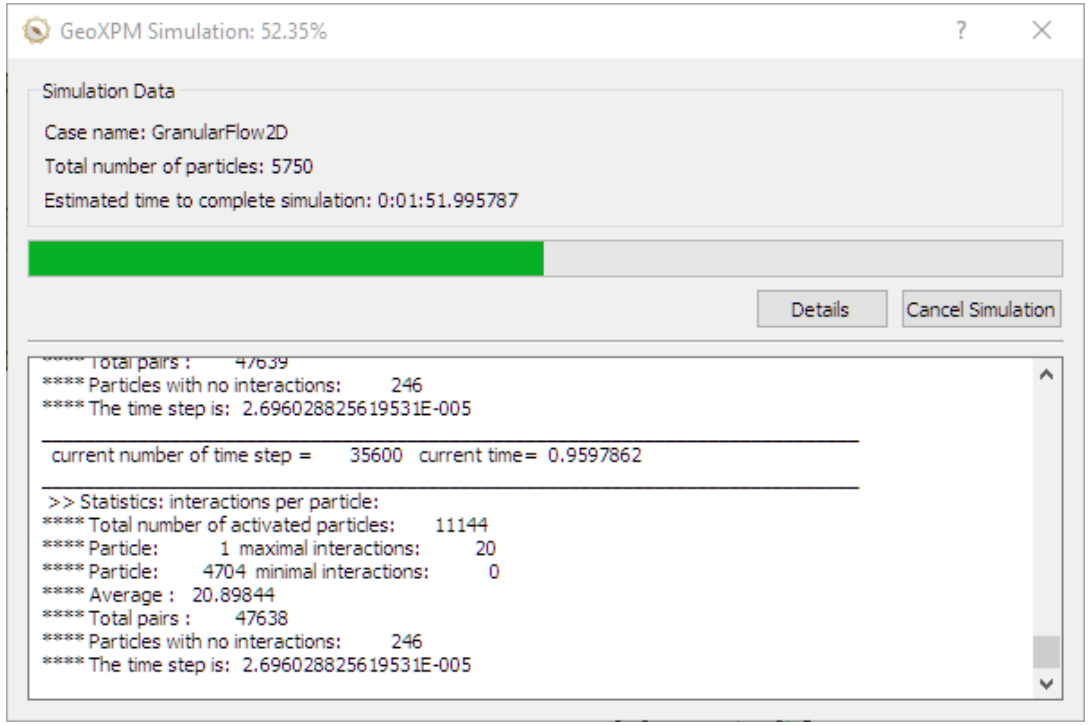




WORKFLOW – 9. CALCULATE

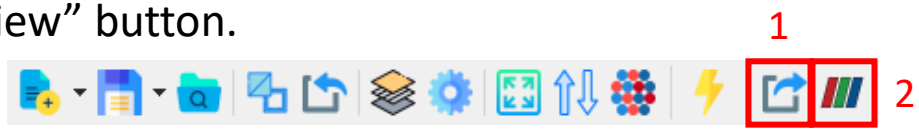
1. Click the Calculation button to start the calculations.

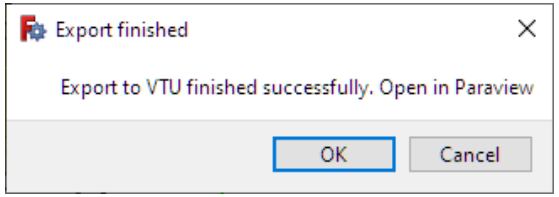




WORKFLOW – 10. VISUALIZE RESULTS

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.