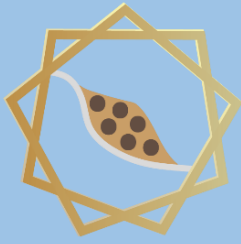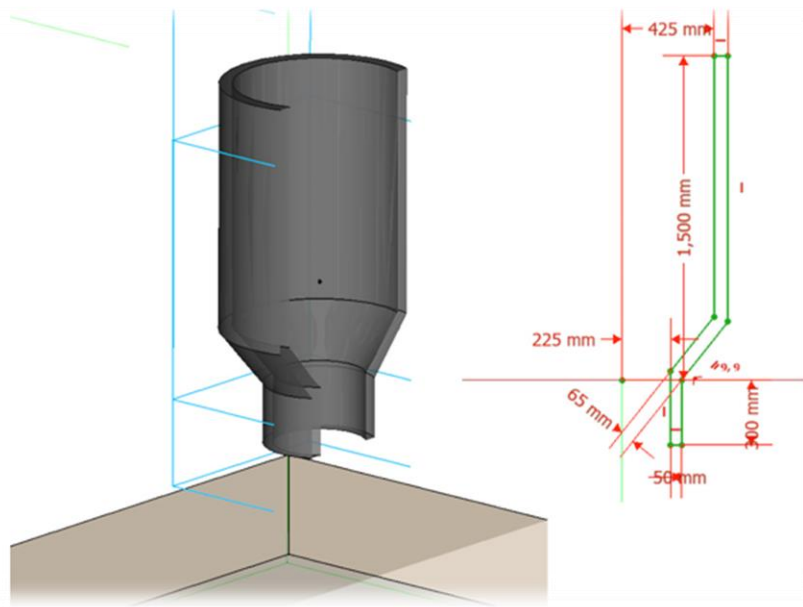# GeoXPM

Particle-based continuum solutions for extreme geoengineering, geomechanics & geophysics applications

GeoXPM
Tutorial Manual

Ha H. Bui

Tien V. Nguyen

*Giang D. Nguyen*

Introduction

GeoXPM is a new generation of computational software package that was originally developed by Prof Ha H. Bui and his research team at Monash University in Australia based on the Smoothed Particle Hydrodynamics (SPH) method. It has been developed specifically to address the ever-evolving demands of solving intricate geotechnical engineering, geomechanics and geophysics challenges, particularly those involving fracturing, large deformation and flow behaviours of geomaterials. One standout feature of GeoXPM is its interactive graphical user-interface, which empowers users to effortlessly generate and modify complex computational models without encountering any obstacles. The calculation procedure within GeoXPM is both robust and fully automated, eliminating the need for extensive training or specialised knowledge. With GeoXPM, academics and professionals alike can unlock the full potential of engineering applications, pushing boundaries and delivering remarkable results that exceed expectations.

The tutorial explores a wide range of challenging and practical scenarios that encompass the intricate aspects of large deformation and post-failure behaviour in geomaterials. The main aim is to equip new users with a comprehensive understanding of GeoXPM. However, it is crucial to recognise that this tutorial should not be solely relied upon as the basis for practical projects. Instead, it serves as an invaluable resource to familiarise users with the functionalities and capabilities of the software.

Users are expected to possess a fundamental understanding of the SPH method and constitutive modelling of geomaterials. For those who are new to the software, it is highly recommended that they adhere to the default setting parameters and follow the step-by-step procedure meticulously to ensure accurate results. It is important to note that variations in hardware and software configurations may yield slight differences in the obtained results.

This Tutorial Manual does not provide the theoretical background information on the SPH method or elaborate on the specific details of the constitutive models utilised within the software. Therefore, users are highly advised to refer to the provided references for comprehensive information regarding the SPH method and constitutive models implemented in GeoXPM. These references offer in-depth insights into the underlying principles and methodologies employed, allowing users to gain a thorough understanding of the theoretical aspects involved.

Example:

3D Excavation
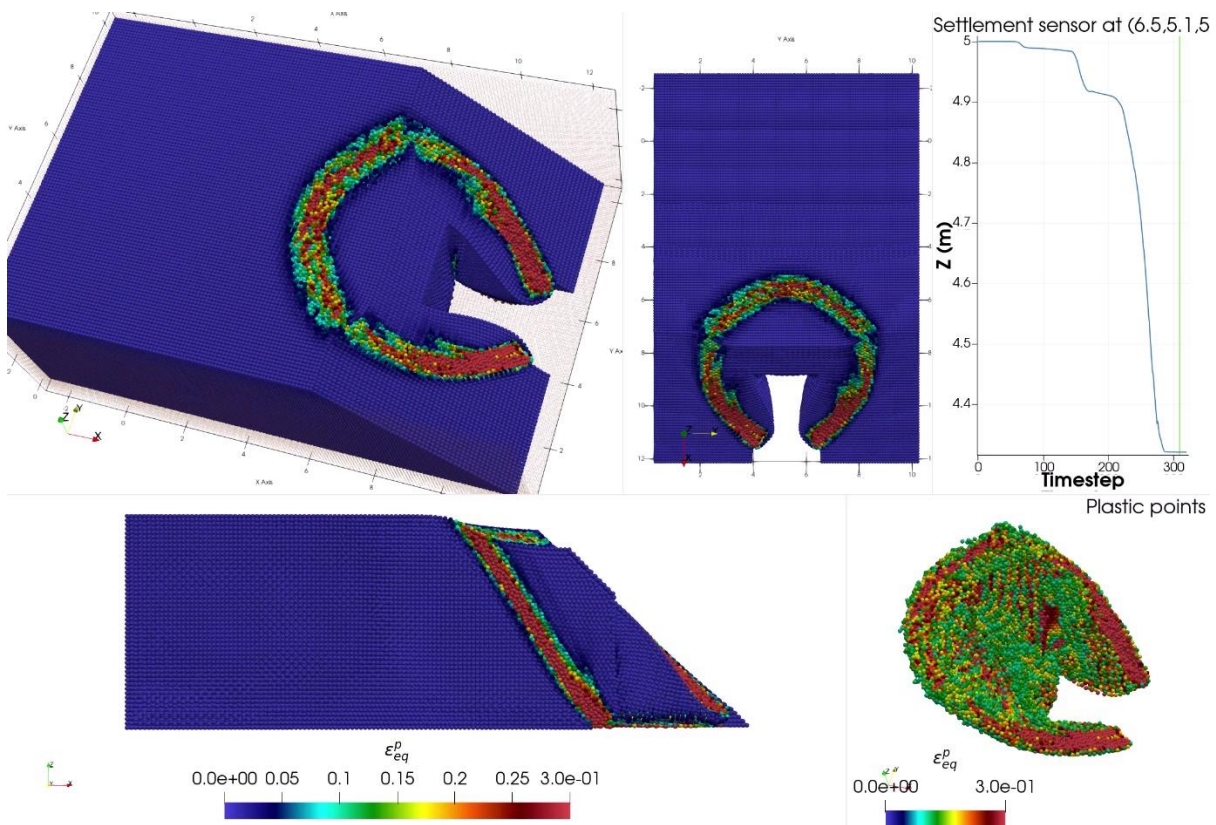
In this project, you will learn about using:

- **Stages,** "**FillBox**"
- **Autofill** tools
- **Drawing and manipulating 3D objects**.

We will follow the below standard workflow:

1. Create a new project.
2. Create initial geometry.
3. Input SPH parameters.
4. Create material models.
5. Include objects in simulations and assign material properties.
6. Define construction stages.
7. Define meshing sequences.
8. Mesh objects – generate particles for calculations.
9. Calculate.
10. Post-processing - Visualisation: extract, data mining.

➢ Step 1. Create a new project

a. Click the "New Project" button in the dialogue below to create a new project.
b. Enter the project name and project description.

➢ Step 2. Create initial geometry

a. Creating the four boundaries wall (Front, Back, Left, Bottom). The first boundary is the Front:

  i. Click on the "Cube" primitive object ( )to insert a basic cube.
  ii. On the Model tab. Rename the cube to "Front".
  iii. Input the dimension and the position of the "Front" wall into the "Data" tab.
  iv. Repeat the process for The Left, Bottom and Back walls.

Front	
Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
⌄ Placement	[(0.00 0.00 1.00); 0.00 °; (-2800.00 m...
Angle	0.00 °
› Axis	[0.00 0.00 1.00]
⌄ Position	[-2800.00 mm 0.00 mm 0.00 mm]
x	-2800.00 mm
y	0.00 mm
z	0.00 mm
Label	Front
Box	
Length	21.80 m
Width	500.00 mm
Height	5000.00 mm

Bottom	
Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
⌄ Placement	[(0.00 0.00 1.00); 0.00 °; (-2800.00 m...
Angle	0.00 °
› Axis	[0.00 0.00 1.00]
⌄ Position	[-2800.00 mm 0.00 mm -500.00 m...
x	-2800.00 mm
y	0.00 mm
z	-500.00 mm
Label	Bottom
Box	
Length	21.80 m
Width	10.50 m
Height	500.00 mm

Property	Value		Property	Value
Attachment			**Attachment**	
Support			Support	
Map Mode	Deactivated		Map Mode	Deactivated
Base			**Base**	
∨ Placement	[(0.00 0.00 1.00); 0.00 °; (-2800.00 m...		∨ Placement	[(0.00 0.00 1.00); 0.00 °; (-2800.00 m...
Angle	0.00 °		Angle	0.00 °
> Axis	[0.00 0.00 1.00]		> Axis	[0.00 0.00 1.00]
∨ Position	[-2800.00 mm 10.00 m 0.00 mm]		∨ Position	[-2800.00 mm 0.00 mm 0.00 mm]
x	-2800.00 mm		x	-2800.00 mm
y	10.00 m		y	0.00 mm
z	0.00 mm		z	0.00 mm
Label	Back		Label	Left
Box			**Box**	
Length	21.80 m		Length	500.00 mm
Width	500.00 mm		Width	10.50 m
Height	5000.00 mm		Height	5000.00 mm
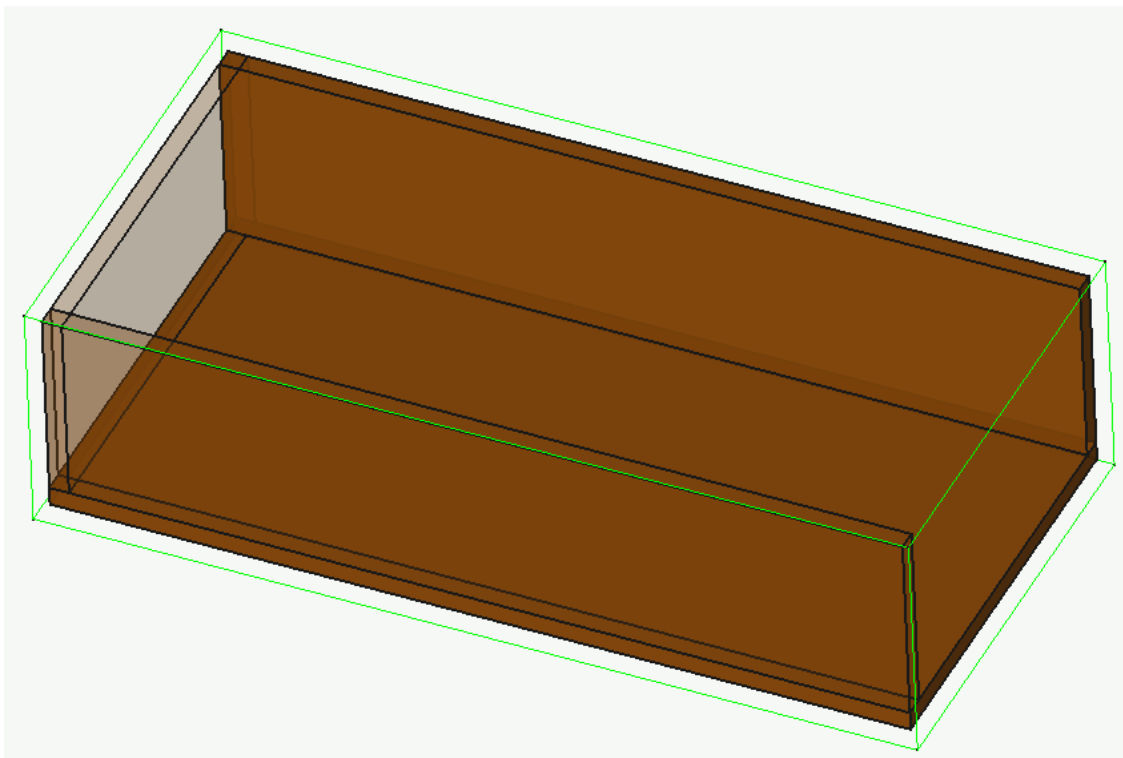
Hint: The thickness of the wall (width in the case of the "Front" wall) is unimportant as we will only use one face of the cube to generate particles later.

b. Creating the whole soil domain. To do this, we will first draw a 2D sketch in XZ space, then extrude the sketch along the Y direction.

  i. Go to the "Part Design" workspace.
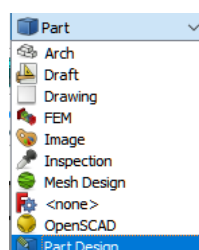
ii.   Click on "Create body" (⬡ Create body) , then "Create sketch" (⬚ Create sketch).

iii. Select XZ_Plane and Click OK.

OK	Cancel

⊠ Select feature   ⌃

XY_Plane (Base plane)
XZ_Plane (Base plane)
YZ_Plane (Base plane)
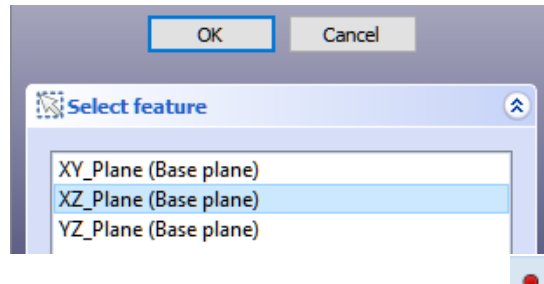
iv.   In the sketch space, use the create polyline tool (⚫⚫). Draft an initial geometry.
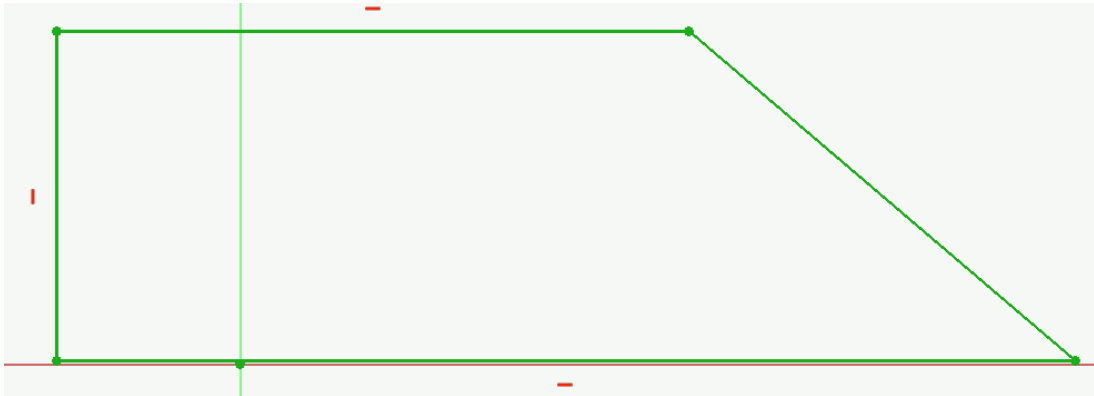
v.   To get the exact dimensions, other constraints can be used. Specifically, we used these constraints: "Constrain vertically", "Constrain horizontally", "Constrain horizontal distance", "Constrain vertical distance" and "Constrain angle".

✖	Constrain coincident	C
⌐	Constrain point onto object	O
		Constrain vertically	V
—	Constrain horizontally	H
⫽	Constrain parallel	P
⊥	Constrain perpendicular	N
⬏	Constrain tangent	T
=	Constrain equal	E
✕	Constrain symmetrical	S
⊗	Constrain block	K, B
🔒	Constrain lock	K, L
⊢⊣	Constrain horizontal distance	L
I	Constrain vertical distance	I
✎	Constrain distance	K, D
⊘	Constrain radius or weight	K, R
⊘	Constrain diameter	K, O
⊘	Constrain auto radius/diameter	K, S
◁	Constrain angle	K, A
⟊	Constrain refraction (Snell's law')	K, W
⊛	Constrain internal alignment	
⊡	Toggle driving/reference constraint	K, X
⬚	Activate/deactivate constraint	K, Z
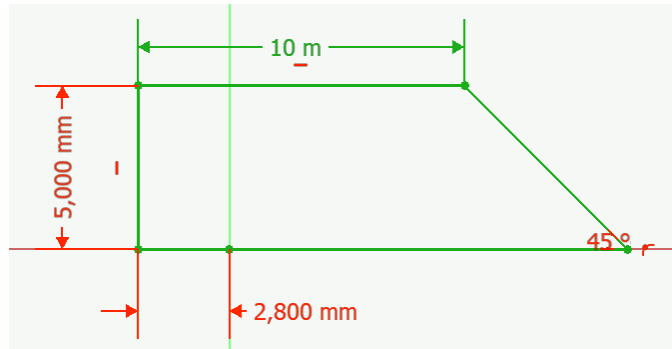
vi. After completing the sketch, click Close.

vii. In the "Tasks" tab, click "Pad" (🟫Pad) and enter 10500 mm in "Length" and tick on the "Reversed" button to extrude the sketch along the Y axis, then click OK.
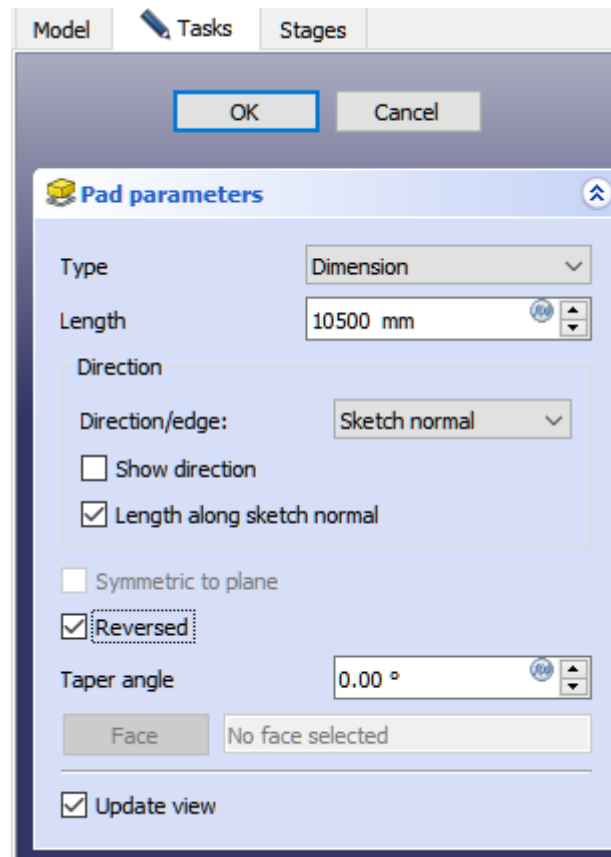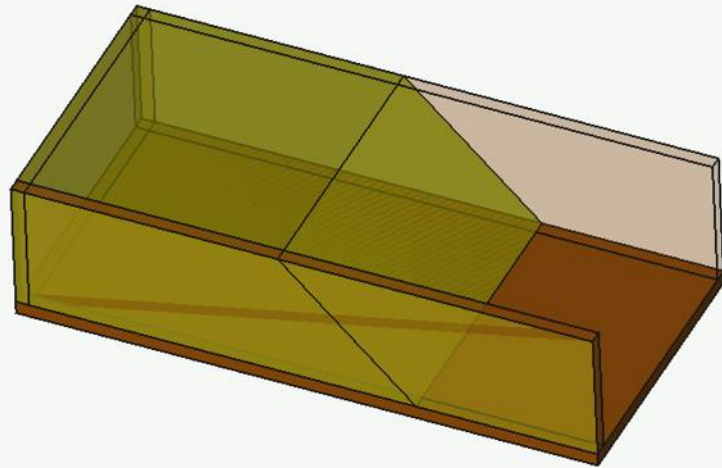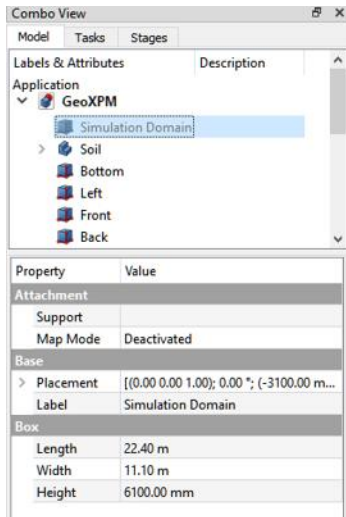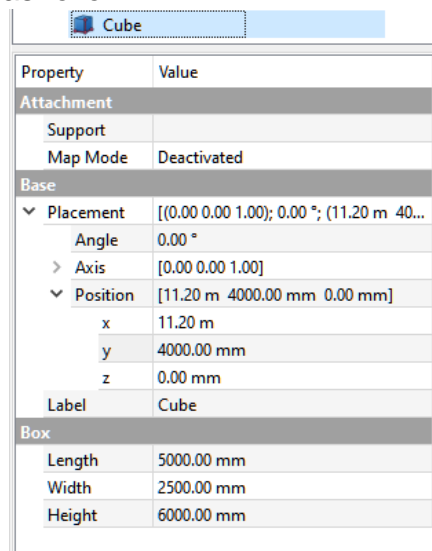

viii. Switch back to "Part" workplace.
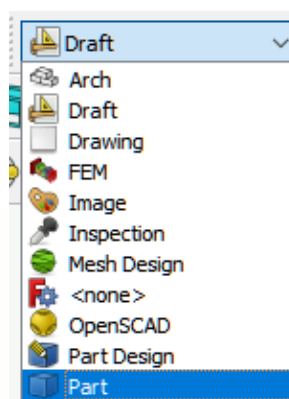

ix. Rename the "Body" object to "Soil".

c. The "Soil" object now needs to be divided into excavated parts and an embankment. To do this, we will create cubes and use these cubes to slice the "Soils" apart.

    i. Create a Cube object as follow:

    ii. Select the "Cube", and go to "Edit" on the menu→ Duplicate selection. Repeated until there are 7 cubes in total corresponding to 7 excavation stages.

   iii. Move each cube in the -X direction so that each cube is 500 mm away from the other.
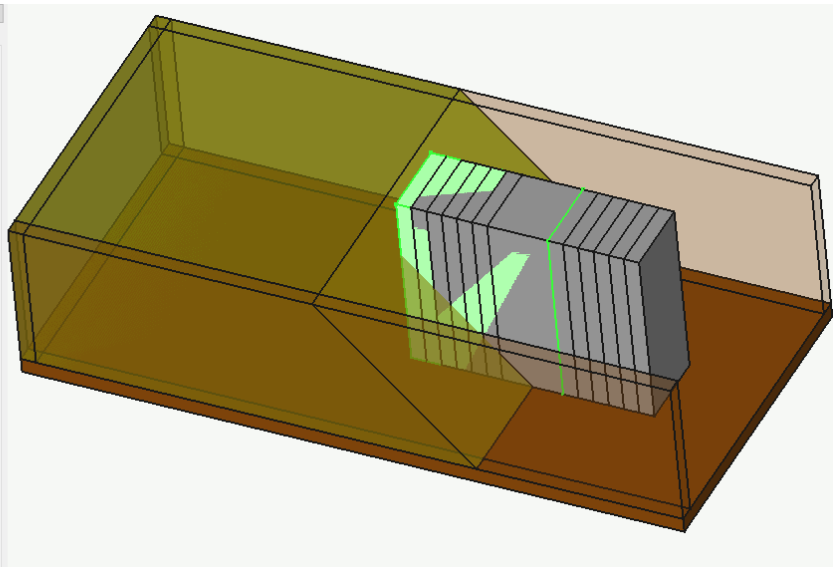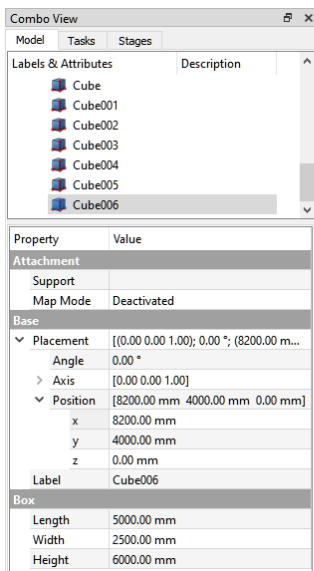
iv.   Select the "Soil" object, then select the 7 Cube objects. Go to "Part" on the menu → Split → Slice apart.

Hint: When selecting objects for slicing. The first object is the one being cut; the later objects are the cutting objects.

v.   Go to the "Draft" workspace. Select the Exploded Slice and click Downgrade ( ) to convert the slices into solid objects. Rename the solid objects correspondingly. You can now remove all the Slices objects.
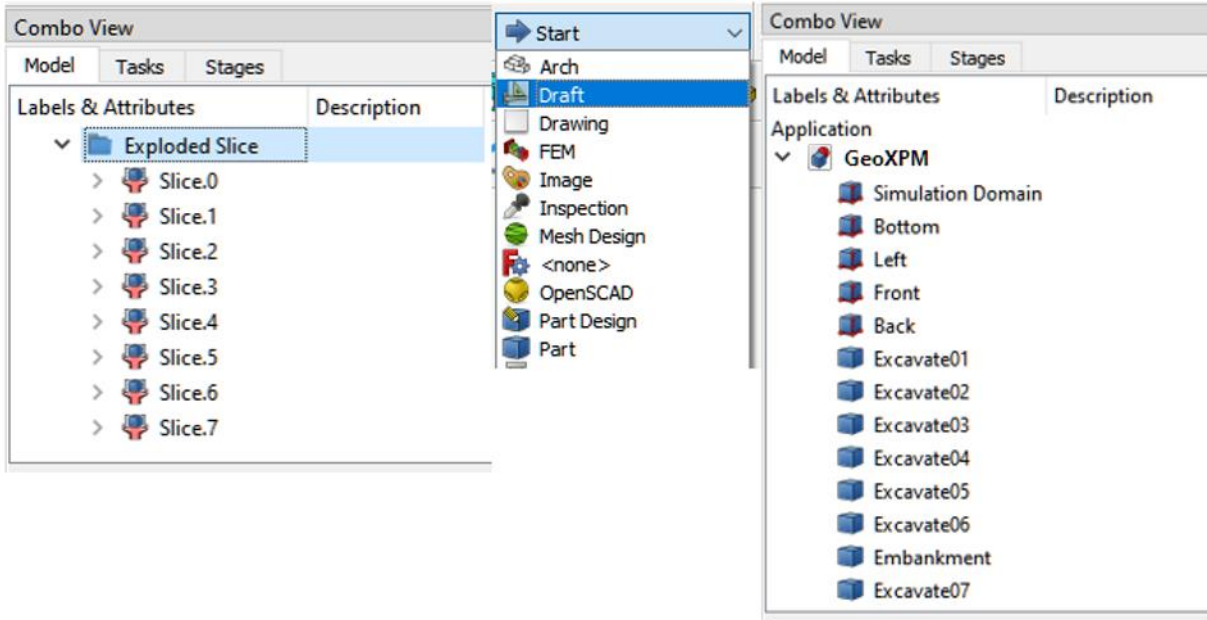
d.   Using the "FillBox" tool ( ) to fill the "Excavate01" object (First excavation of 1m) with Embankment particles. Create and rename the "Fillbox" object to "Excavate01F". Place "Fillpoint" inside the "Excavate01" object, and set the "Filllimit" to cover the "Excavate01".

Hint: For the Embankment and other Excavate objects, we are going to use the "Autofill" function. The autofill function will automatically define a bounding box around an object (FillLimit) and put the Fillpoint at the center of the bounding box. Though convenient, if the centre of the bounding box does not lie inside the object, Autofill will work incorrectly!

So far, our simulation should look like this:
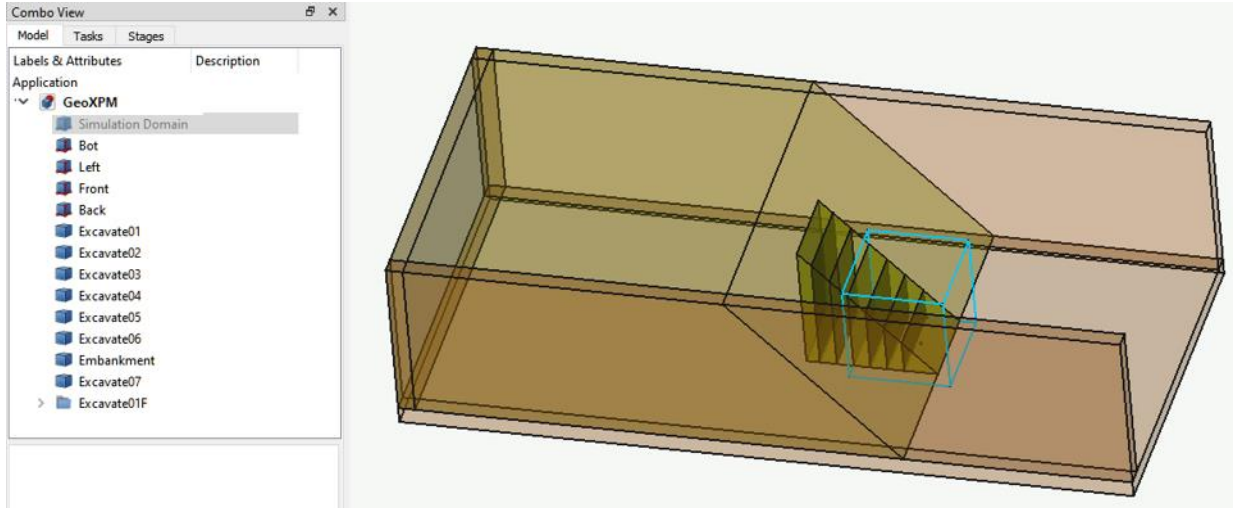
Hint: If you want to save on calculation time, You can also reduce the number of particles in half by simulating only half of the soil domain as follow:
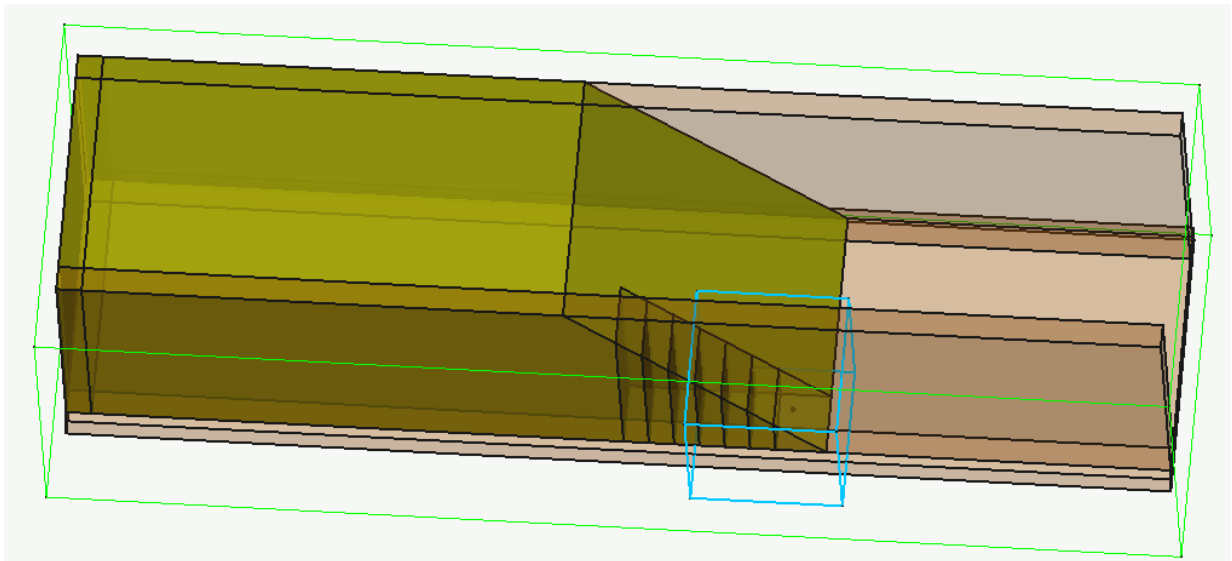
➢ Step 3. Input SPH parameters
a. Enter the SPH parameters into the dialogue.
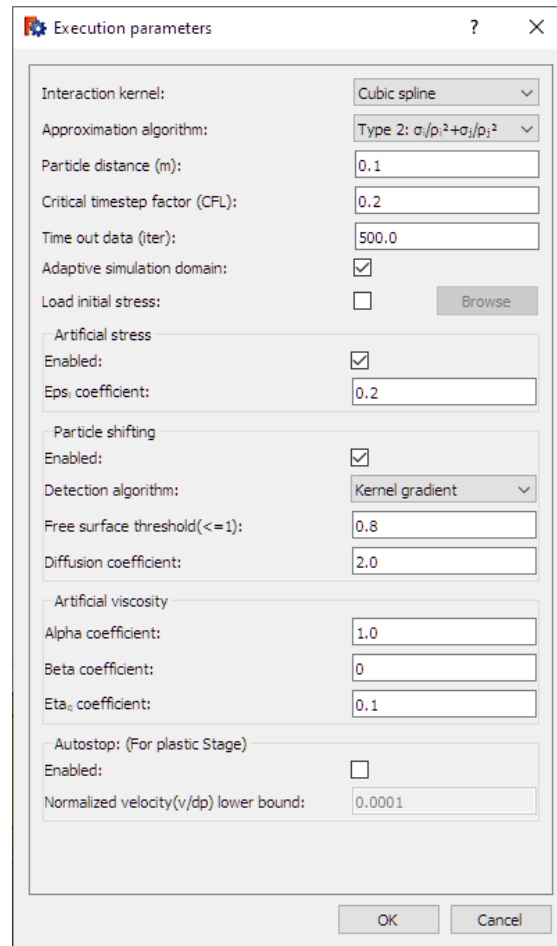
➢ Step 4. Input material properties

a. Click the Material property button, select the appropriate constitutive model (i.e., Elastic-perfecly plasticity (DP)) provided in the "new material" button, and enter the material parameters.
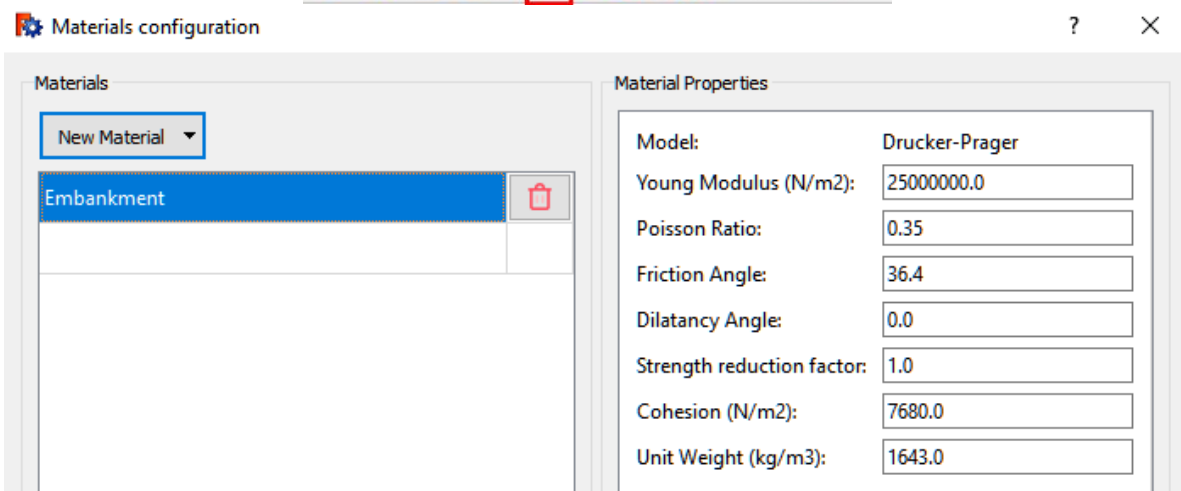
Hint: The timestep for the elastoplastic model depends heavily on the sound speed, which depends on the material's Young Modulus and Unit Weight. Decreasing Young

Modulus and increasing Unit Weight will increase the timestep, making the simulation faster but risking extreme particle overlapping and penetration.

## ➢ Step 5. Include objects in simulations and assign material properties

a. Soil objects:
   i. Select the "Embankment" in the Model tab and click "Add to Simulation" under the "GeoXPM" tab and select the "Type of object" as "Soil". In the next step, click the "Assign" button to assign the "Embankment" model to the object in the material configuration dialogue. Repeat the process for the seven excavations and the fillbox "Excavate01F" objects.
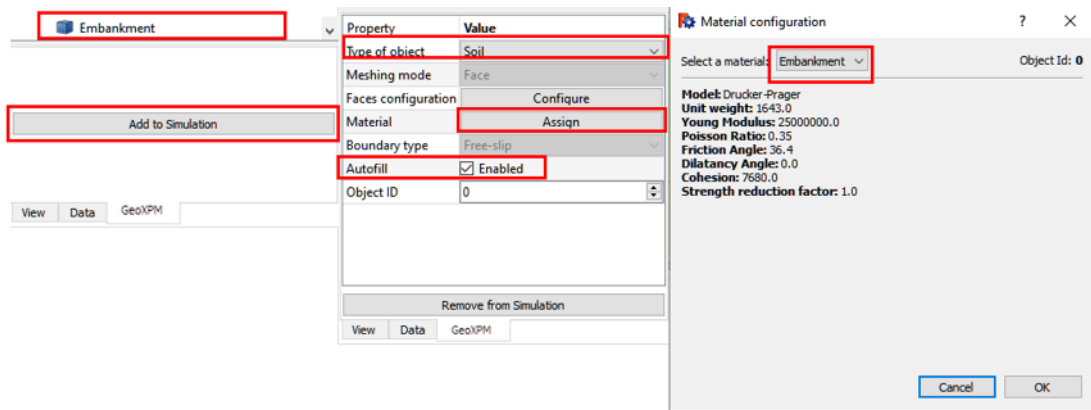

   ii. Select and tick the "Enabled" autofill option for the Embankment and all the excavation objects except Excavation01, where we used Fillbox instead in Step 2d.

b. Boundary objects:
   i. Select the "Front" object, and click "Add to Simulation" under the "GeoXPM" tab. Then select the "Type of object" as "Boundary", and the boundary type is "Free-slip". In the next step, define the Front face of the "Front" Box object as the meshing face by ticking the "Front face" option. Input 0,-1,-2 into the Layers textbox to create three boundary layers, with the first layer on the Box's Front face (0) and the subsequent two layers in the inward direction of the "Front" Box object (-1,-2). Repeat the process for the "Left" and "Back" objects.
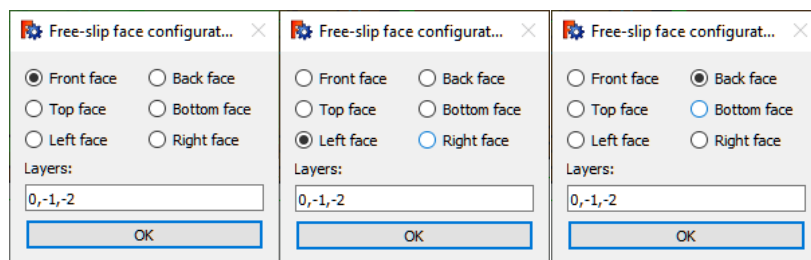

   ii. Select the "Bottom" object, and click "Add to Simulation" under the "GeoXPM" tab. Select the "Type of object" as "Boundary", the meshing mode as "Face", and the boundary type as "No-slip". Click the "Configure", and tick the "Bottom face". Input 0,-1,-2 into the Layers textbox.
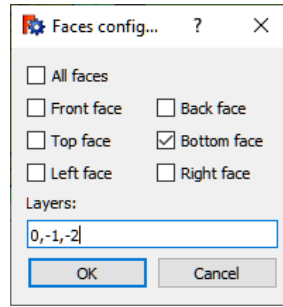
➢ Step 6. Define construction stages

a. Go to the Stage tab and click "Add stage".
b. Add Stage 0 for the stress initialisation using the elastic loading method.
c. Set the "Type of calculation" to "Elastic".
d. Set the "Time of simulation" by double-clicking on the number and inputting 5000 steps.
e. Tick the checkbox next to "Reset displacement to 0" to reset displacement after stress initialisation.
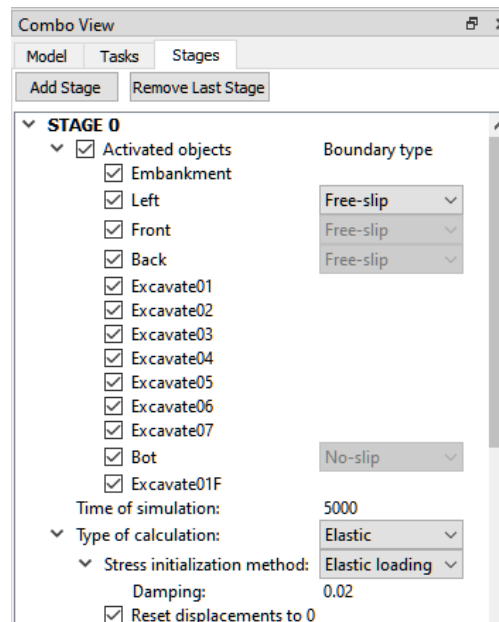
f. Add Stage 1 for the first excavation stage. Set the "Type of calculation" to "Plastic". Set the "Time of simulation" to 3000.
g. Untick Excavate01 and Excavate01F to remove the 2 objects during this Stage.
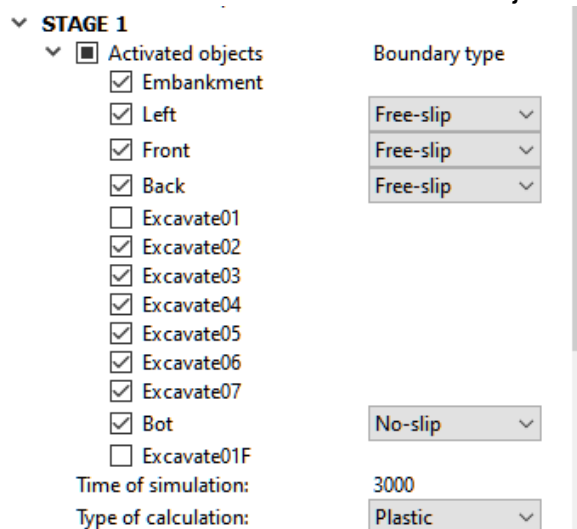
h. Repeat the process to add Stage 2,3,4,5,6,7, which further removes Excavate02,03,04,05,06 and 07 objects in each stage. The corresponding "Time of simulation" are 3000, 3000, 5000, 10000, 15000, and 20000.

Hint: You can change the Boundary condition from Free-slip to No-slip between Stages. For example, if next to the "Front" wall is a rigid boundary, then from "Stage 1" onward, you can change its Boundary type to No-slip.

➢ Step 7. Define the meshing sequence

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.
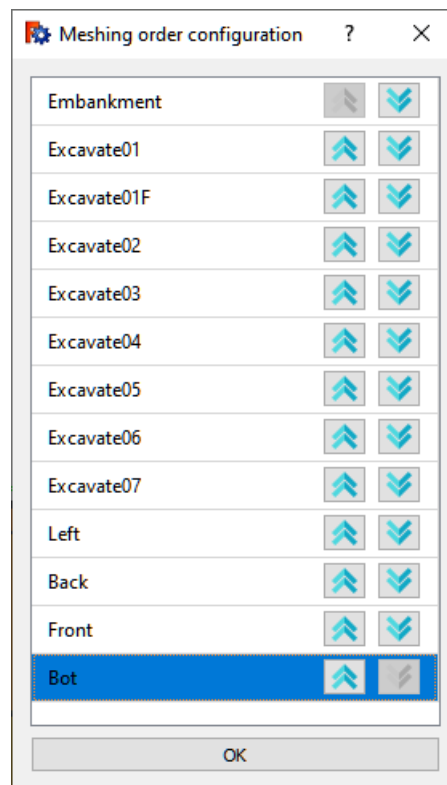

Meshing order configuration

Hint: Fillbox will fill only work after meshing its container.

➢ Step 8. Generate particles for computations

a. Before generating particles, click the save button to save the project as 2D or 3D based on your need.

c. Click the "Autofit simulation domain" button ( ) to fit all simulated objects in the calculation domain.

b. Click the "Generate particles" button( ) to mesh the computational domain.
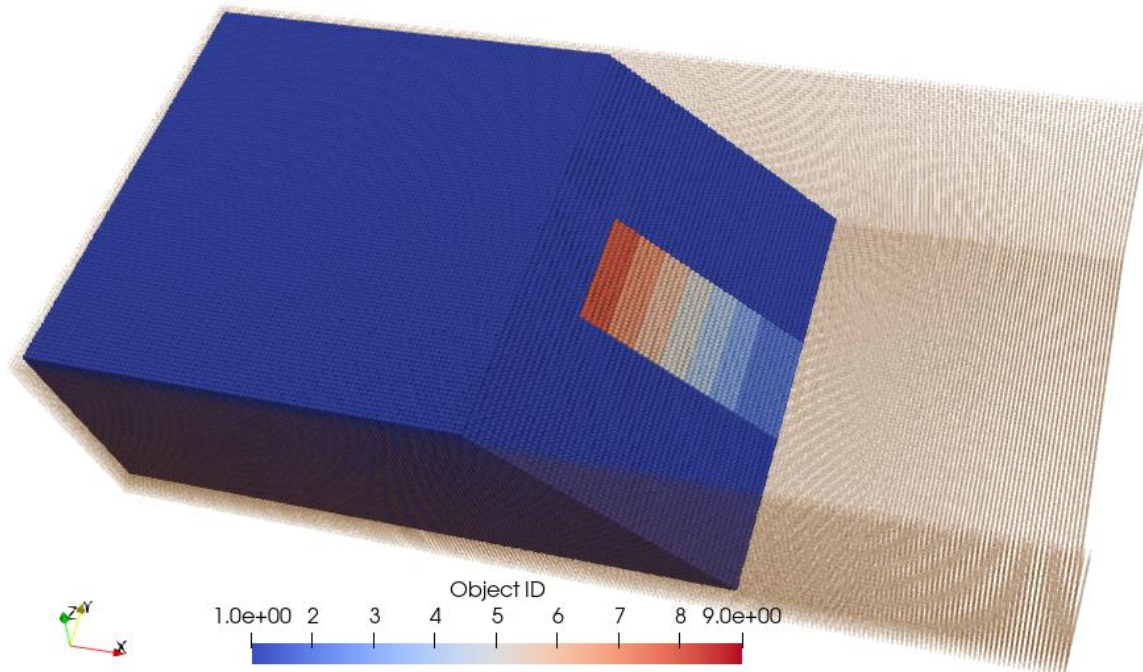
c. After meshing, click "Open with Paraview" to view the particle configuration.
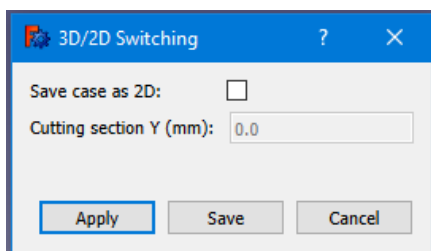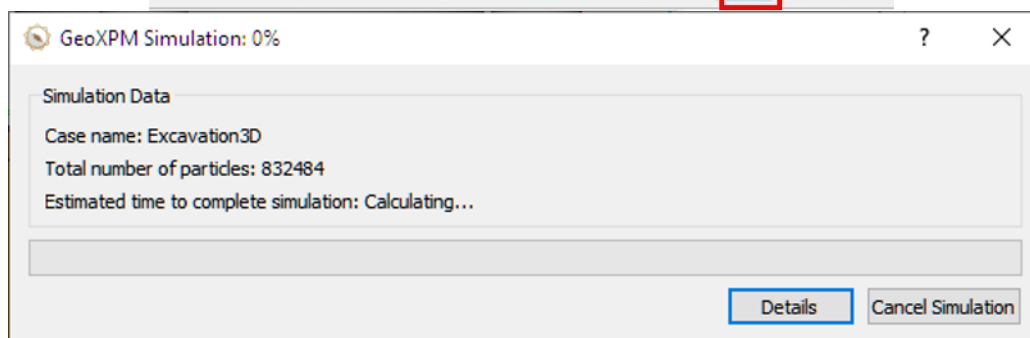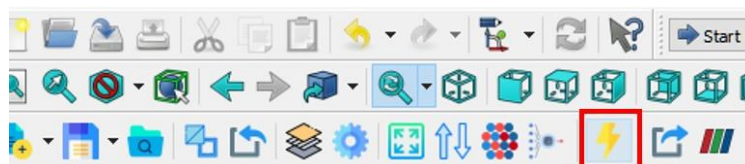
Save the project as 3D
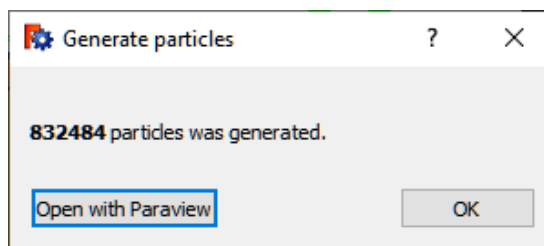
Check the particle configuration

Hint: You can use the Paraview Slice tool ( ) to cut the geometry and look inside the soil domain for irregularity.

## ➢ **Step 9. Start the Calculation**

Click the "Run" button and start the calculations.

➢ Step 10. Visualisation

a. Click on the tool "Export output data to VTU format" to start exporting data to Paraview anytime during the simulation or after completion.

b. When the export is done, Paraview will open with already imported results. All the data can be customised and visualised from here.
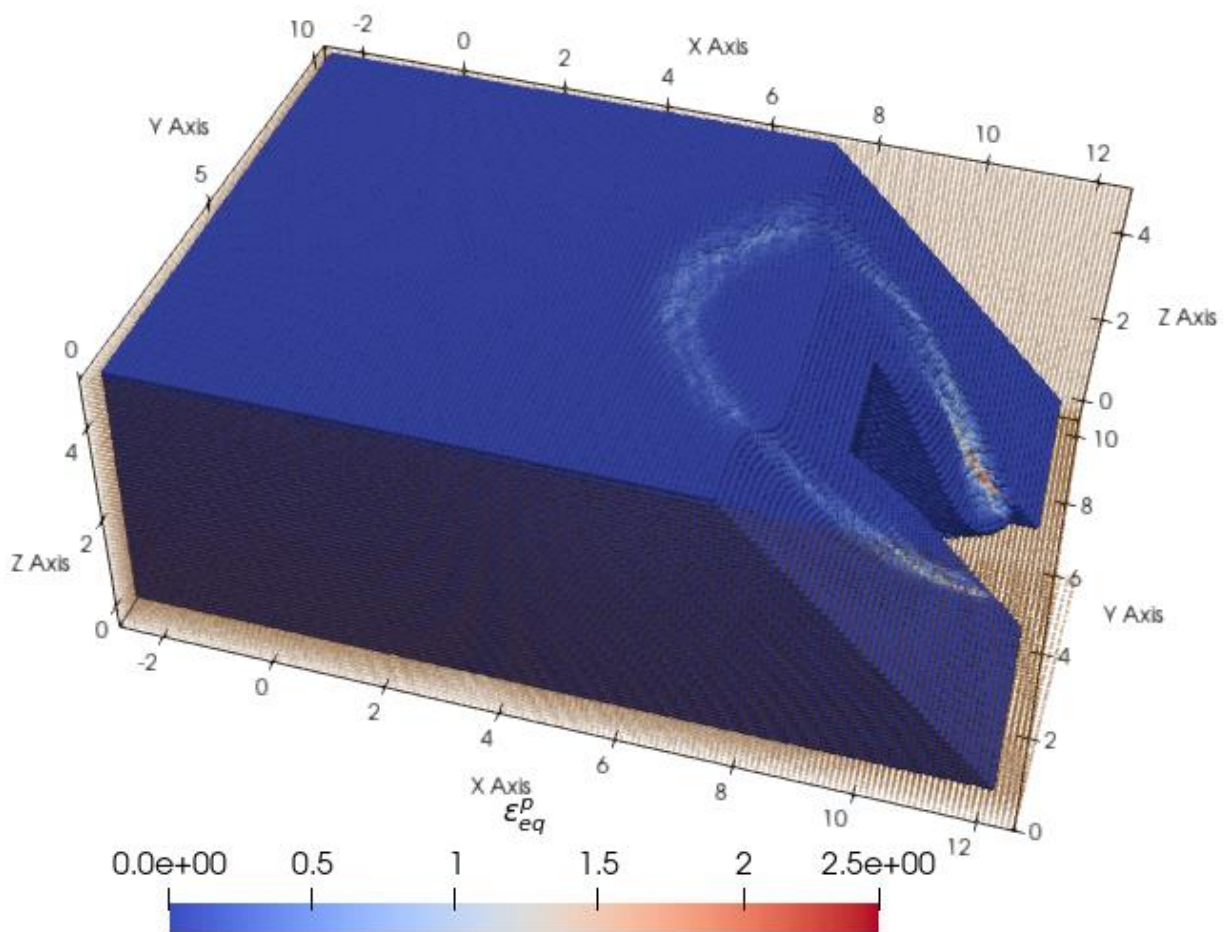
Bonus: Data extraction in Paraview

Let's say we put a displacement sensor on the top of the soil domain. How are we going to get that information?

Assuming we put the sensor at the coordination X=6.5m, Y= 5.3m and Z=5m, we want to find the closest particle to that point in space and track its movement through time.

To do this, we can follow the following steps:

a. Go to View → Find Data to activate the "Find Data" tab.
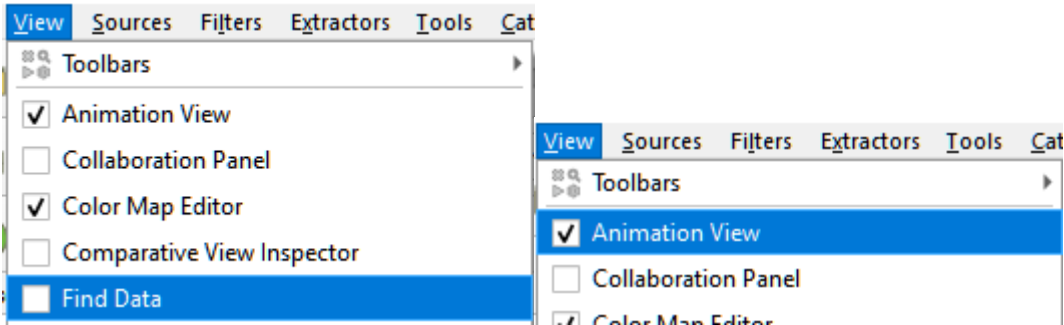
b. In the "Find Data" tab. Input the Selection Criteria as follow and Click the "Find Data" button. The resulting point will be shown in the table at the bottom, and the corresponding point will be highlighted as pink on the View Layout.
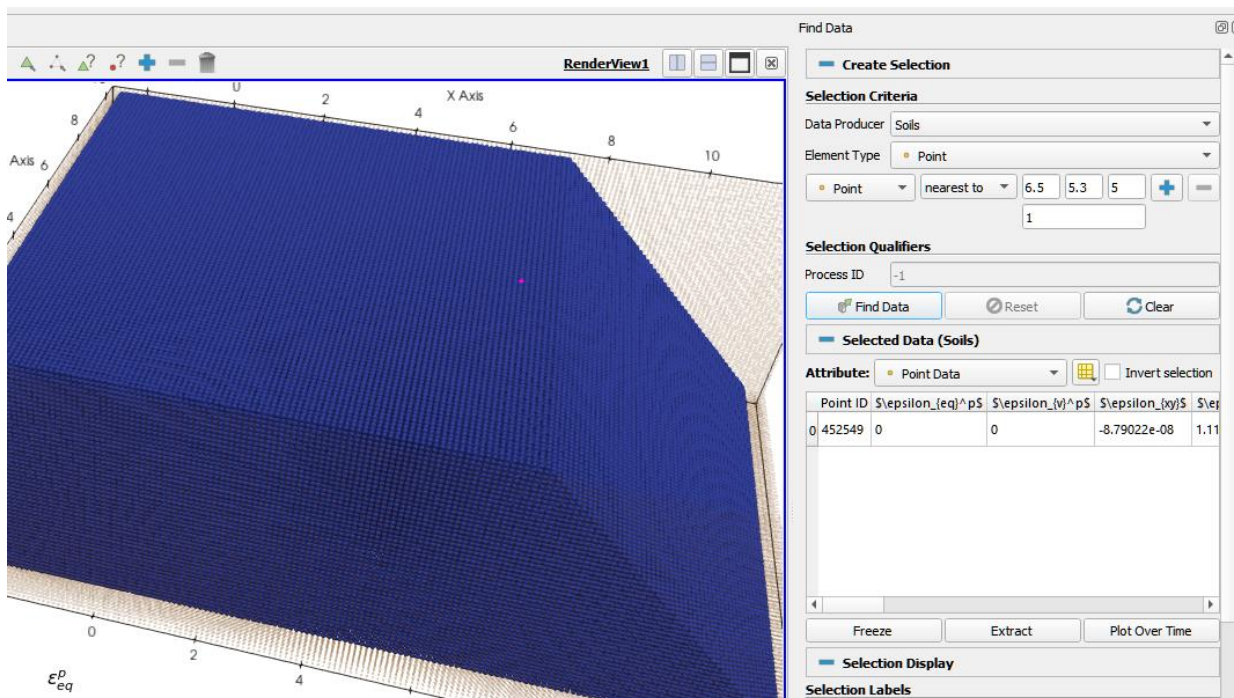
c. Click on the "Plot Over Time" button, then click "Apply" in the "Properties" Tab.

Hint: If the amount of information is significant, Paraview will take some time. Be patient!

    d. Scroll down on the "Properties" tab. In the "Series Parameters" section, deselect everything except for the "Z (stat)".

Hint: You can select data points manually using the "Interactive Select Points On" tool (). You can also select multiple points.

References:

1. Bui H.H, Nguyen V.T & Nguyen G.D (2023). GeoXPM: Particle-based continuum solutions for extreme geoengineering, geomechanics & geophysics applications, Tutorial Manual. Link
2. Bui H.H, Nguyen G.D (2021). Smoothed particle hydrodynamics (SPH) and its applications in geomechanics: From solid fracture to granular behaviour and multiphase flows in porous media, *Computers and Geotechnics* 138, 104315. Link
3. Bui H.H, Fukagawa R, Sako K & Ohno S (2008). Lagrangian mesh-free particles method (SPH) for large deformation and failure flows of geomaterial using elastic-plastic soil constitutive model, International Journal for Numerical and Analytical Methods in Geomechanics, Vol.32(12), pp.1537-1570. Link