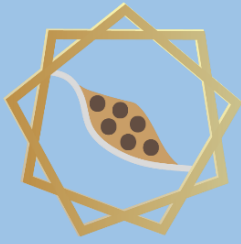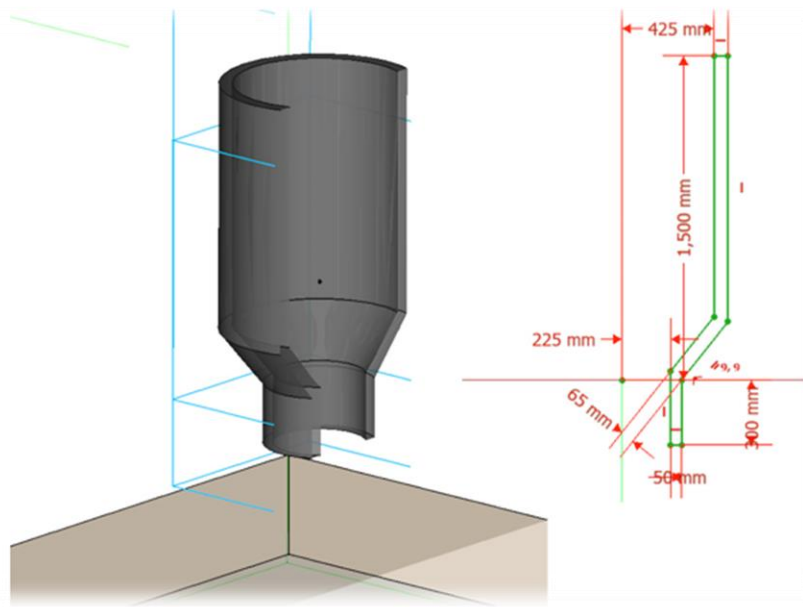# GeoXPM

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 Terrain using imported mesh

In this project, you will learn about using:

- **External Mesh objects**
- "**FillBox**" tool
- **Manipulating 3D objects**
- Creating **animation sequences**.

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.
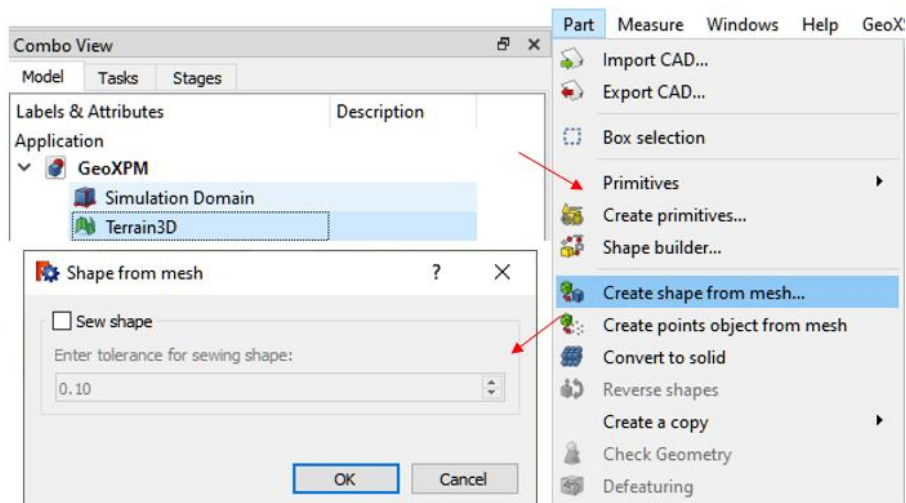b. Enter the project name and project description.
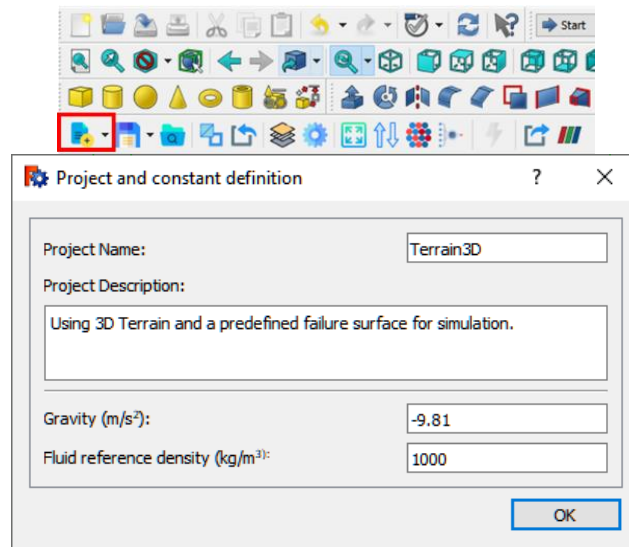

➢ Step 2. Create initial geometry

a. Using the "Import Terrain" tool to import the mesh file for the 3D Terrain.
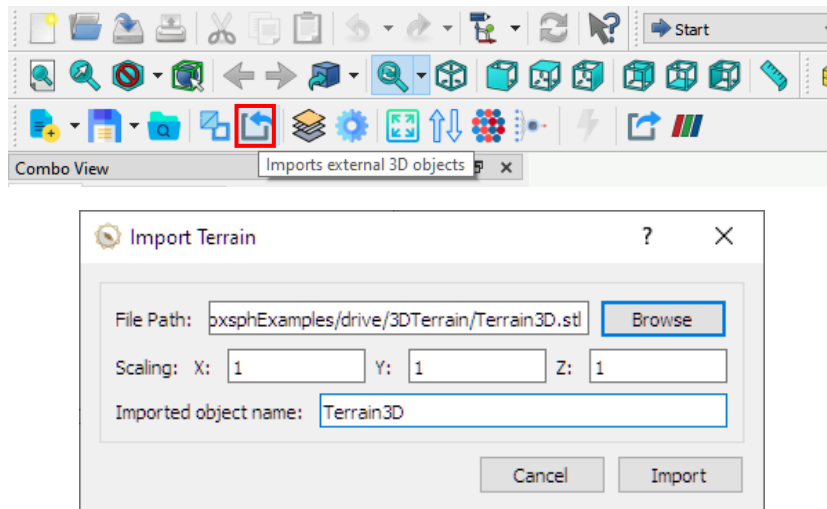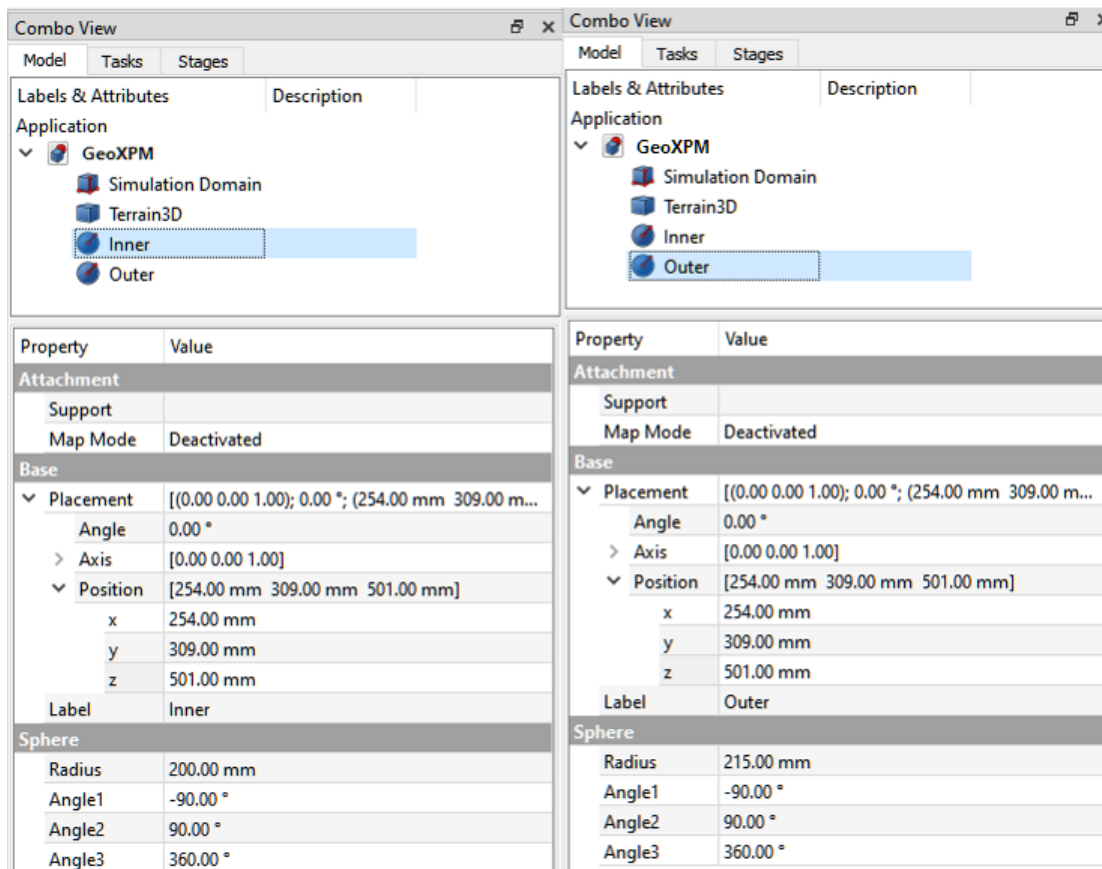

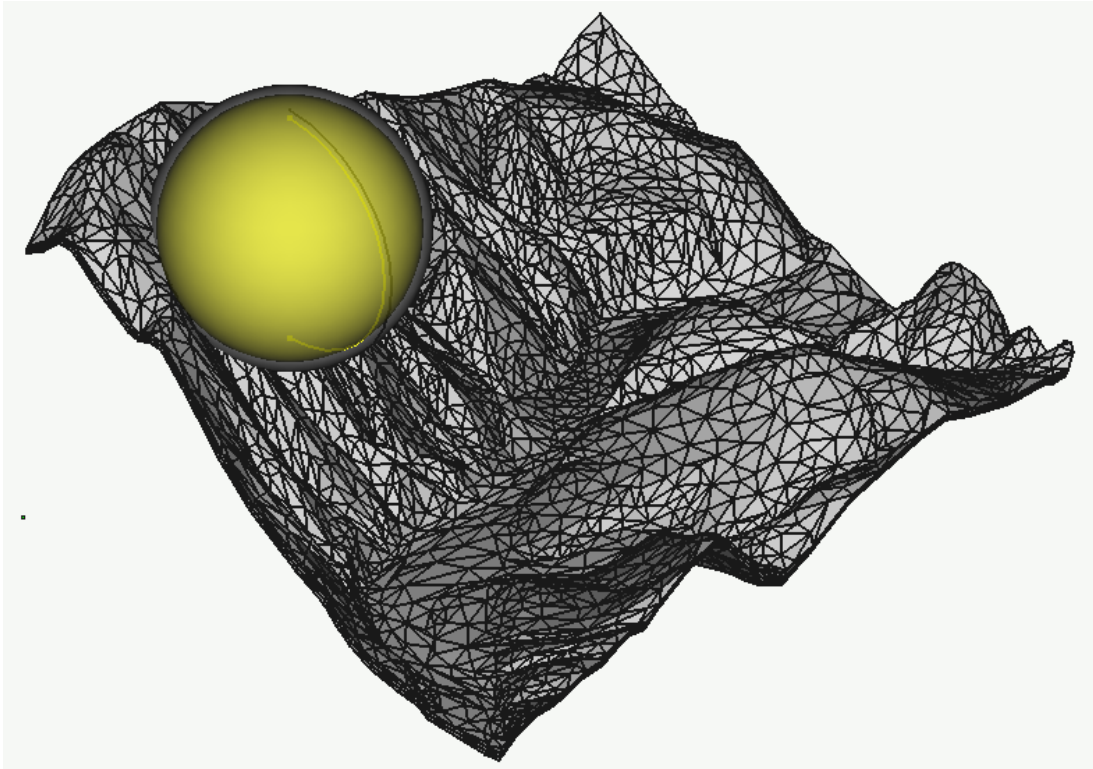b. Select the "Terrain3D" object in the "Model" tab. Go to "Part" menu → "Create shape from mesh" → OK.

c. Select the new shape "external_Terrain3D001" object in the "Model" tab. Go to the "Part" menu → select "Convert to solid".
d. Delete the mesh object "Terrain3D" and the shape object "external_Terrain3D001". Rename the solid object "external_Terrain3D001 (Solid)" to "Terrain3D".
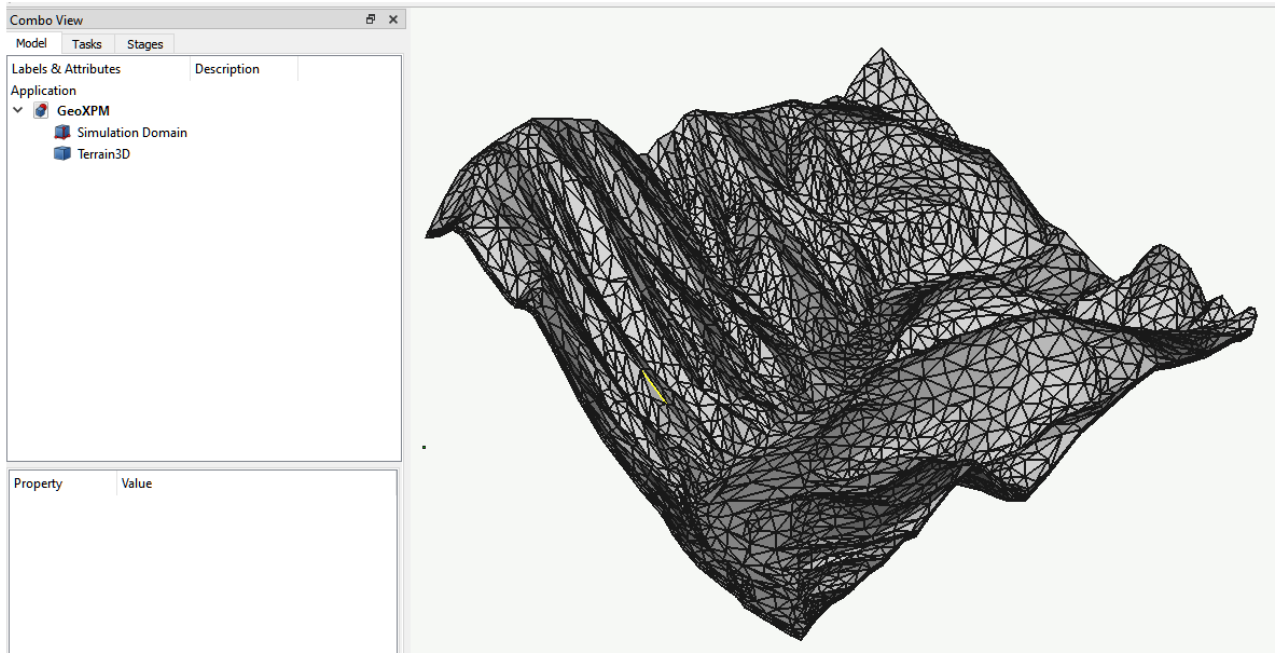

e. Using the Sphere ( ) tool to create two concentric spheres, named "Inner" and "Outer".

f.   Now we will use the "Terrain3D" to cut the "Outer" sphere, then join the Terrain3D and the lower cut part together.

  i.   The first step is to duplicate the Terrain3D. Select the "Terrain3D" and go to the "Edit" menu→ Duplicate selection.

  ii.   In the "Model" tab, select the "Outer" sphere, then select the duplicated object "Terrain3D001". Go to "Part" on the menu → Split → Slice apart.
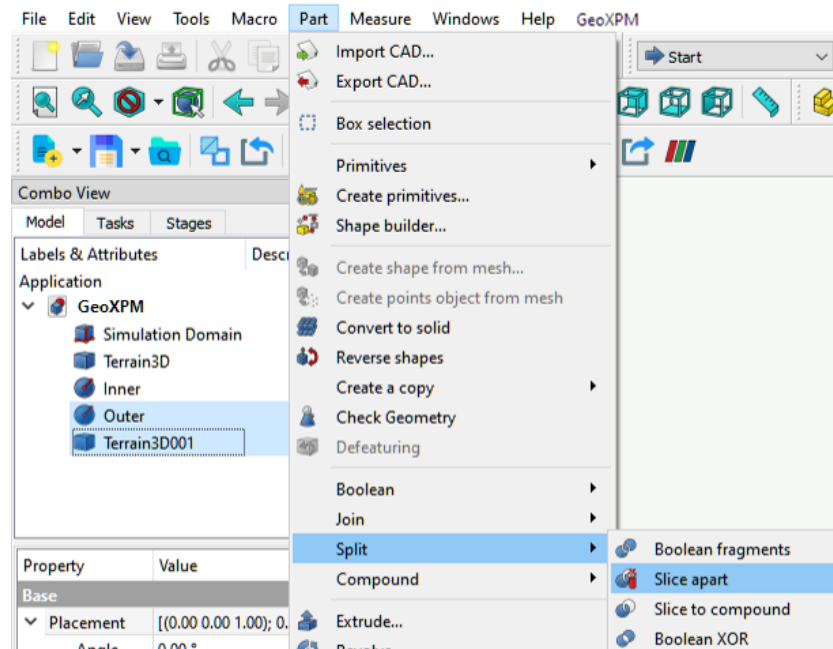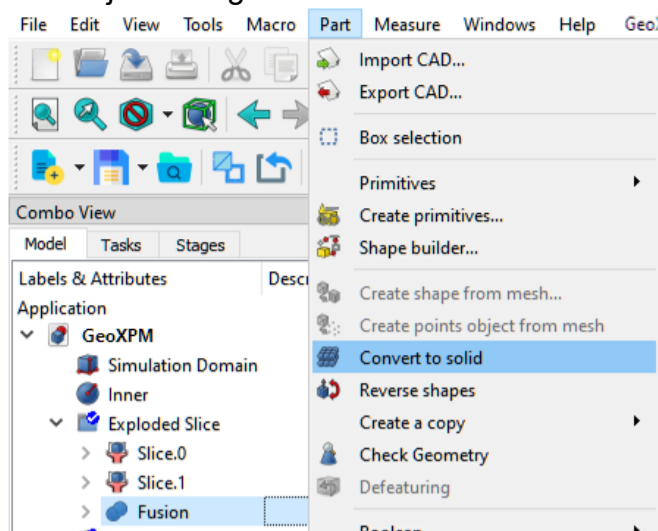

Hint: When selecting objects for slicing. The first object is the one being cut; the later objects are the cutting objects.

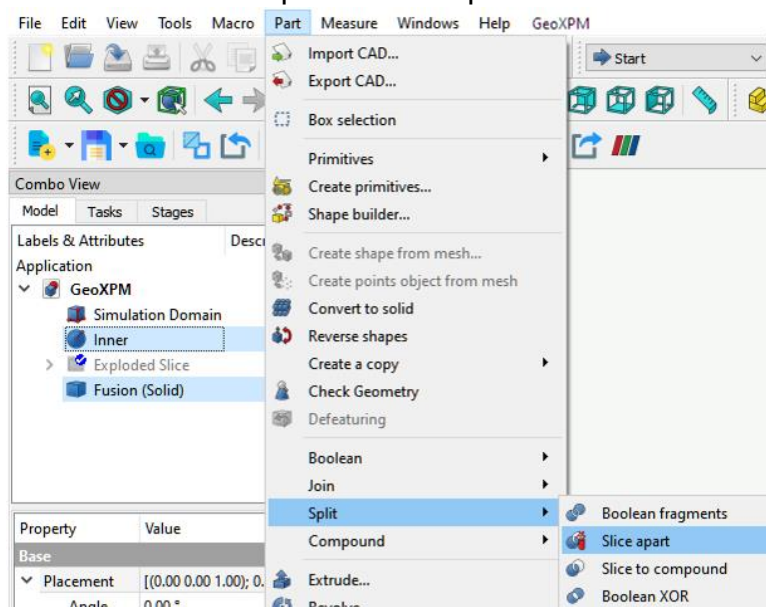 iii.   Select the Exploded "Slice.2" and "Terrain3D". Go to "Part" on the menu → Boolean → Union.
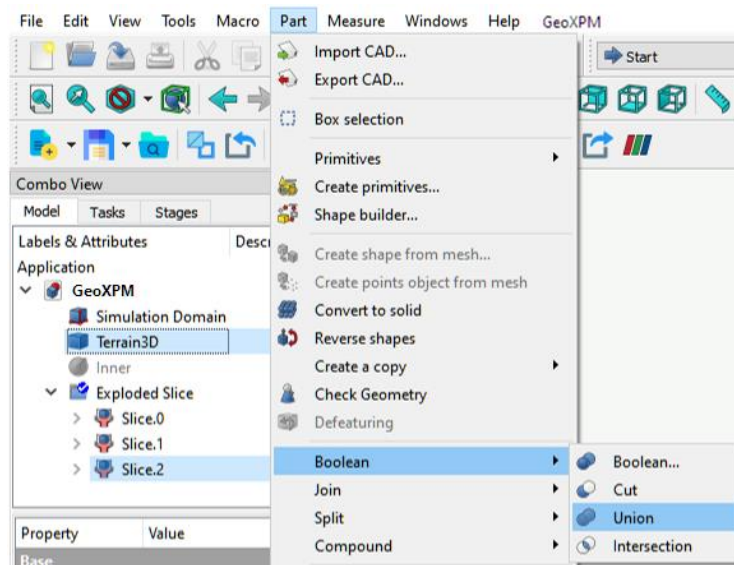
iv. Select the "Fusion" object and go to "Part" on the menu → "Convert to solid".
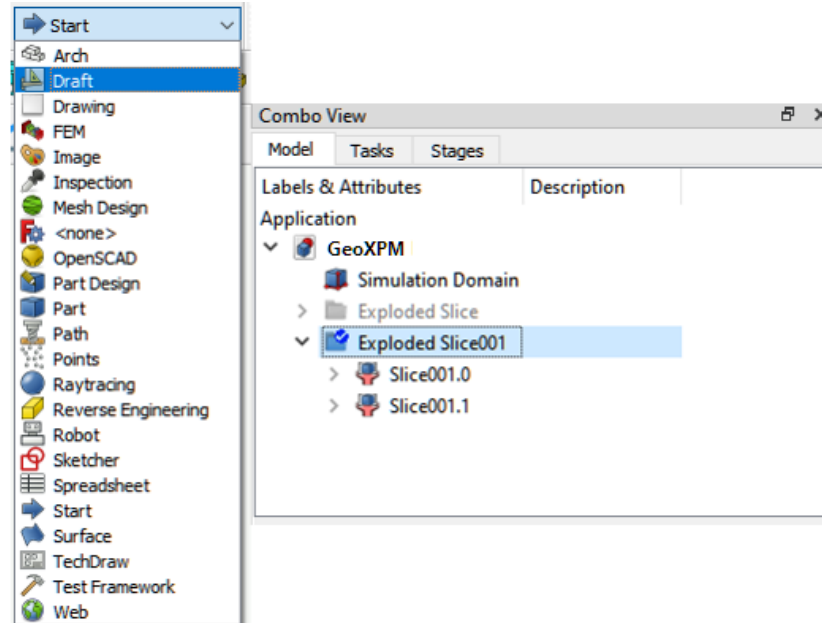

g. Now we will use the "Inner" sphere to Slice the "Fusion (Solid)" into two parts: Boundaries and Soils.

  i. In the "Model" tab, select the "Fusion (Solid)", then select the "Inner" sphere. Go to "Part" on the menu → Split → Slice apart.

ii. Go to the "Draft" workspace. Select the "Exploded Slice001" object and click Downgrade ( ) to convert the two slices into solid objects.

iii. In the "Model" tab, Rename "Solid" to Boundaries and "Solid001" to "Soils". You can now remove all other objects (except the Simulation domain, Boundaries and Soils) or hide them by choosing these objects and going to "View" on the menu → Toggle visibility.
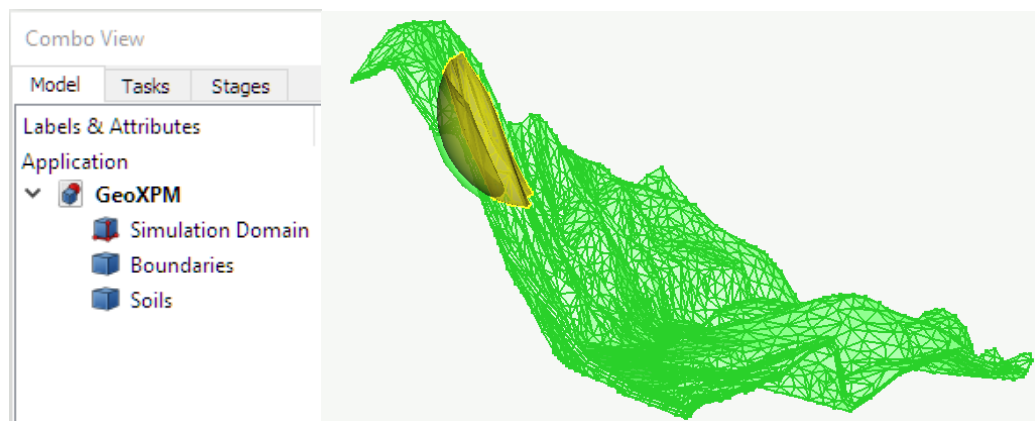
iv. Switch back to "Part" workplace.
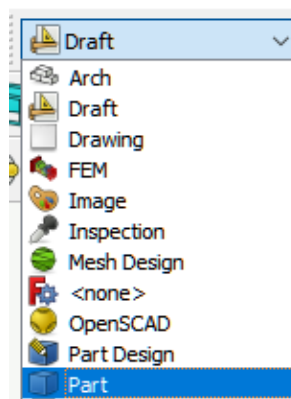
h. Using the "FillBox" tool to fill "Soils" objects with sand particles. "FillPoint" is placed inside the "Soils" object, and the "FillLimit" is set to cover the "Soils".
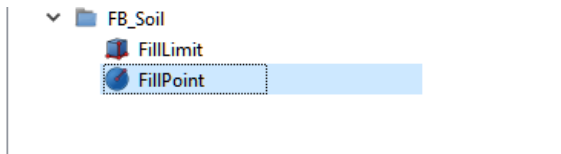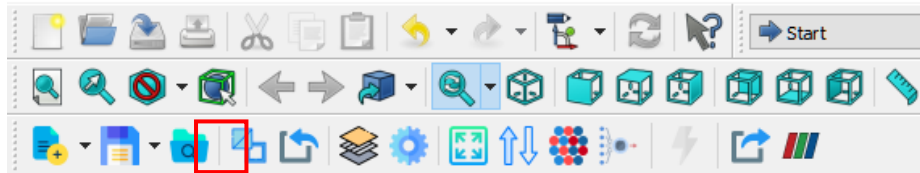
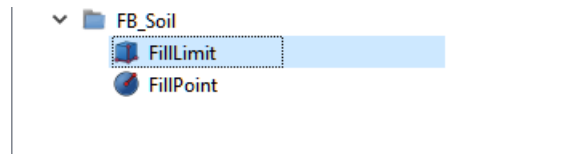| Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
∨ Placement	[(0.00 0.00 1.00); 0.00 °; (141.00 mm 231.00 m...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
∨ Position	[141.00 mm 231.00 mm 383.00 mm]
x	141.00 mm
y	231.00 mm
z	383.00 mm
Label	FillPoint
Sphere	
Radius	10.00 mm
Angle1	-90.00 °
Angle2	90.00 °
Angle3	360.00 °

FillPoint

Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
∨ Placement	[(0.00 0.00 1.00); 0.00 °; (13.00 mm 111.00 mm...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
∨ Position	[13.00 mm 111.00 mm 283.00 mm]
x	13.00 mm
y	111.00 mm
z	283.00 mm
Label	FillLimit
Box	
Length	300.00 mm
Width	300.00 mm
Height	300.00 mm

FillLimit

i. Similarly, Create another "FillBox" to fill the "Boundaries" with boundary particles.
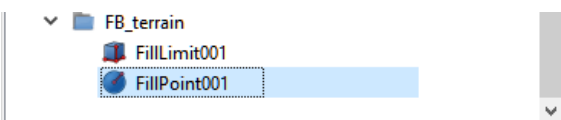
Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
∨ Placement	[(0.00 0.00 1.00); 0.00 °; (900.00 mm 24.00 mm...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
∨ Position	[900.00 mm 24.00 mm 61.00 mm]
x	900.00 mm
y	24.00 mm
z	61.00 mm
Label	FillPoint001
Sphere	
Radius	5.00 mm
Angle1	-90.00 °
Angle2	90.00 °
Angle3	360.00 °
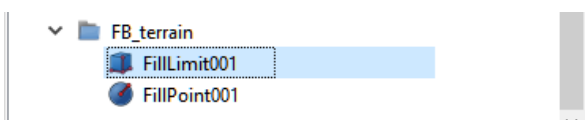
FillPoint

| Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
∨ Placement	[(0.00 0.00 1.00); 0.00 °; (-172.00 mm -76.00 m...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
∨ Position	[-172.00 mm -76.00 mm 0.00 mm]
x	-172.00 mm
y	-76.00 mm
z	0.00 mm
Label	FillLimit001
Box	
Length	1500.00 mm
Width	1500.00 mm
Height	800.00 mm

FillLimit

j. Setup the simulation domain. (Particles goes out of the defined simulation domain will be removed from the simulation).

Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
⌄ Placement	[(0.00 0.00 1.00); 0.00 °; (15.00 mm ...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
⌄ Position	[15.00 mm 15.00 mm 20.00 mm]
x	15.00 mm
y	15.00 mm
z	20.00 mm
Label	Simulation Domain
Box	
Length	950.00 mm
Width	950.00 mm
Height	700.00 mm

| Property	Value
Attachment	
Support	
Map Mode	Deactivated
Base	
⌄ Placement	[(0.00 0.00 1.00); 0.00 °; (-172.00 m...
Angle	0.00 °
> Axis	[0.00 0.00 1.00]
⌄ Position	[-172.00 mm -76.00 mm 0.00 mm]
x	-172.00 mm
y	-76.00 mm
z	0.00 mm
Label	FillLimit001
Box	
Length	1500.00 mm
Width	1500.00 mm
Height	800.00 mm
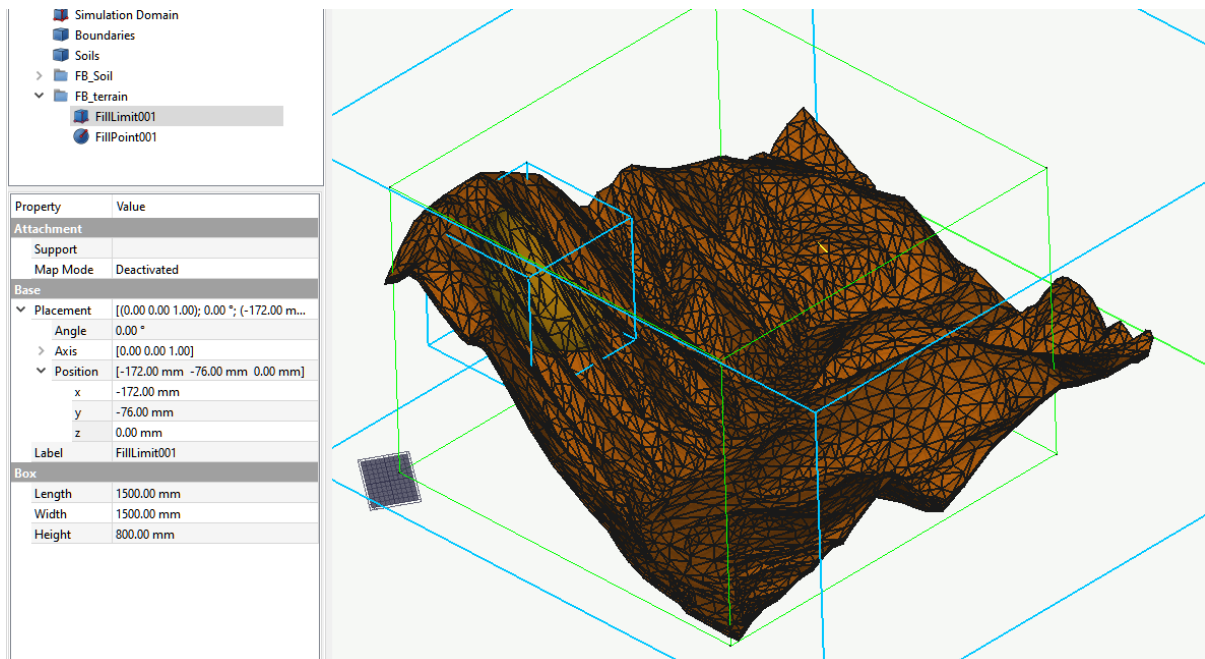
Hint: You can also follow the "3D Hourglass" or "3D Excavation" Example to plot control structures to investigate the effects of different types of baffles or check-dams, for example.
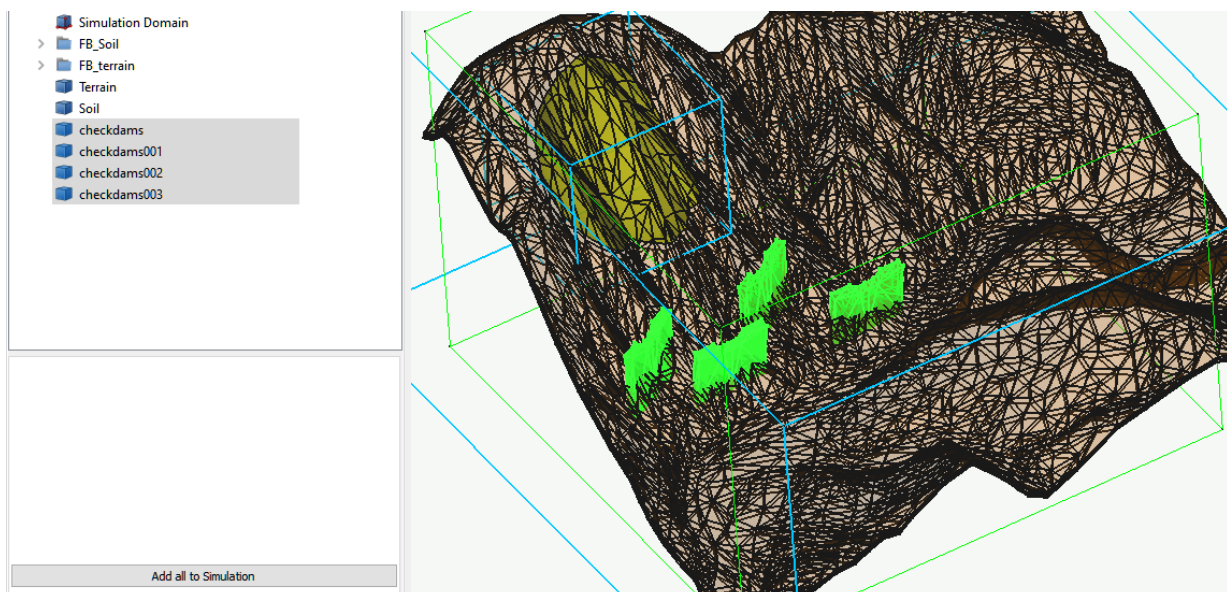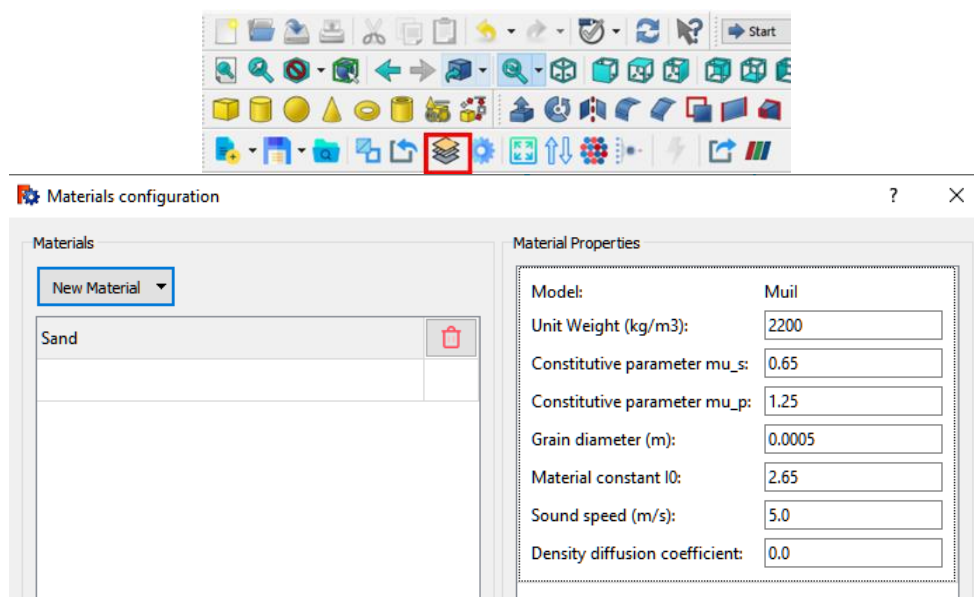
➢ Step 3. Input SPH parameters

a. Enter the SPH parameters into the dialogue.
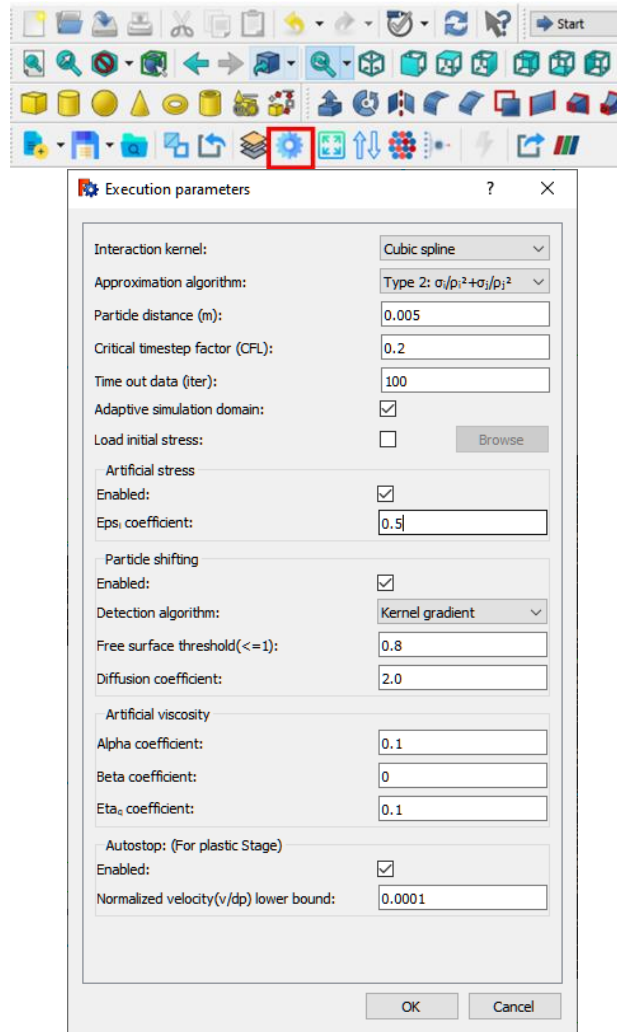

➢ Step 4. Input material properties

a. Click the Material property button, select the appropriate constitutive model (i.e., $\mu(I)$ Rheology model) provided in the "new material" button, and enter the material parameters.

Hint: The timestep for μ(I) model depends heavily on the sound speed. Setting it low makes the simulation faster but risks extreme particle overlapping and penetration. A good initial estimation for the sound speed can be induced from the maximum velocity of a free fall body ($v=\sqrt{2gH_{max}}$) times a factor of 2 to 3.

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

a. Soil object: Select the "Soils" in the Model tab and click "add to Simulation" under the "GeoXPM" tab. And then, select the "Type of object" as "Soil". In the next step, click the "Assign" button to assign the "Sand" model to the soil in the dialogue of the material configuration. Repeat the process for the fillbox "FB_Soil".

b. Boundary object: Repeat the above process for "Boundaries" and "FB_terrain" objects; however, keep "Type of object" as Boundary.
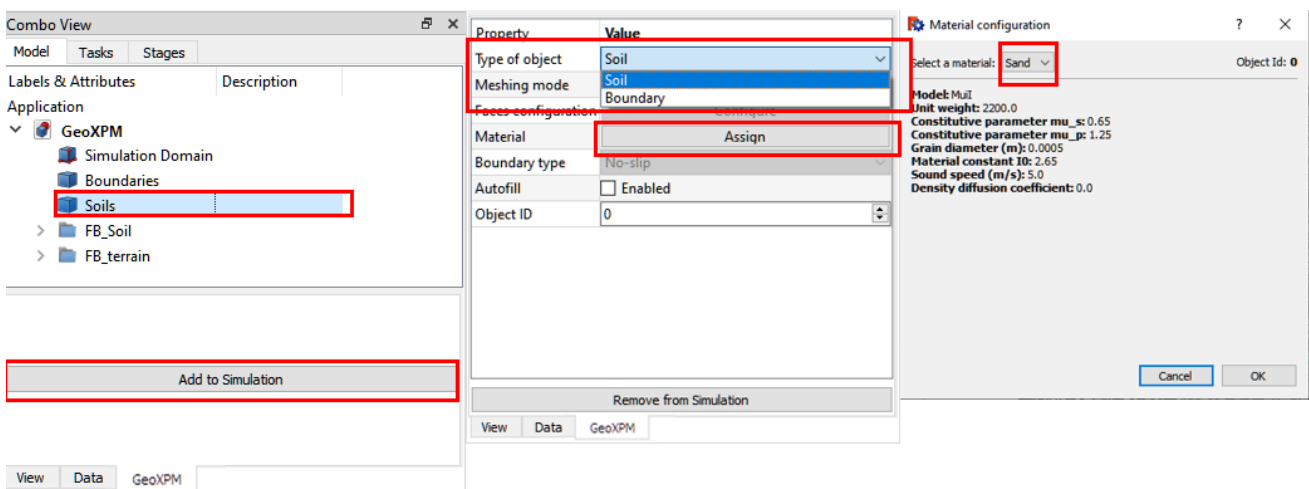

➢ Step 6. Define construction stages

a. Go to the Stage tab and click "Add stage".

b. Click the checkbox "Activated objects" to activate all objects in the simulation.

c. Double-click the number next to "Time of simulation" and enter 30000 steps for the elastoplastic analysis.
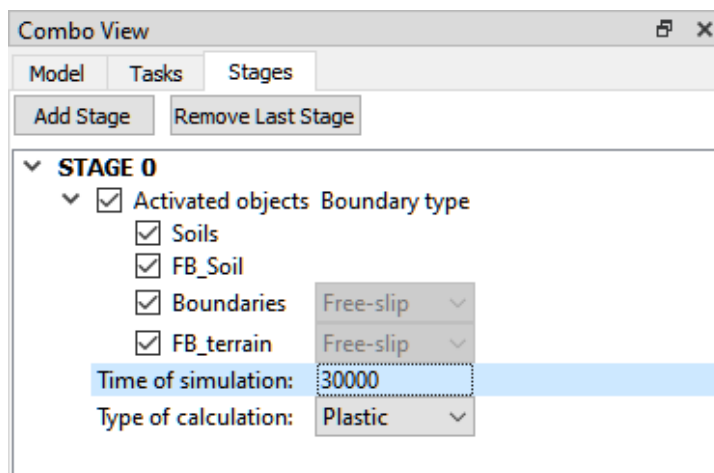
➢ Step 7. Define the meshing sequence

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.
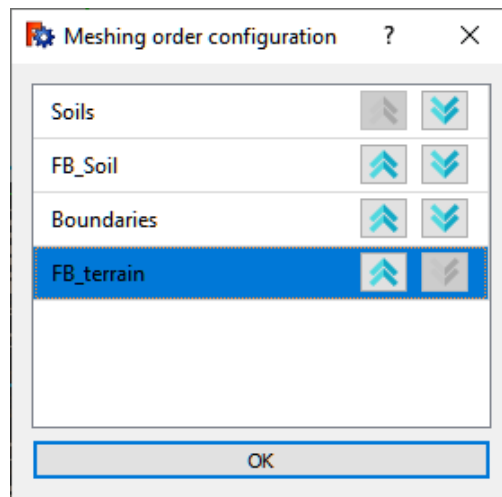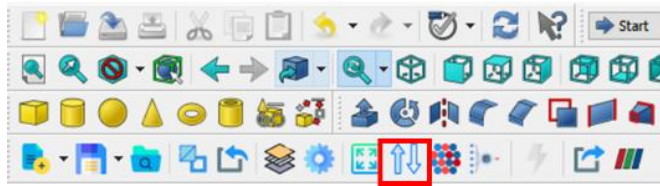

Meshing order configuration

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

➢ Step 8. Generate particles for computations

a. Before meshing, click the save button to save the project as 2D or 3D based on your need.
b. Click the "Generate particles" tool to mesh the computational domain.
c. After meshing, click "Open with Paraview" to view the particle configuration.
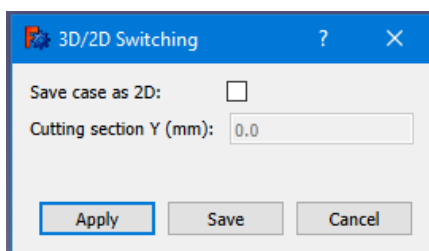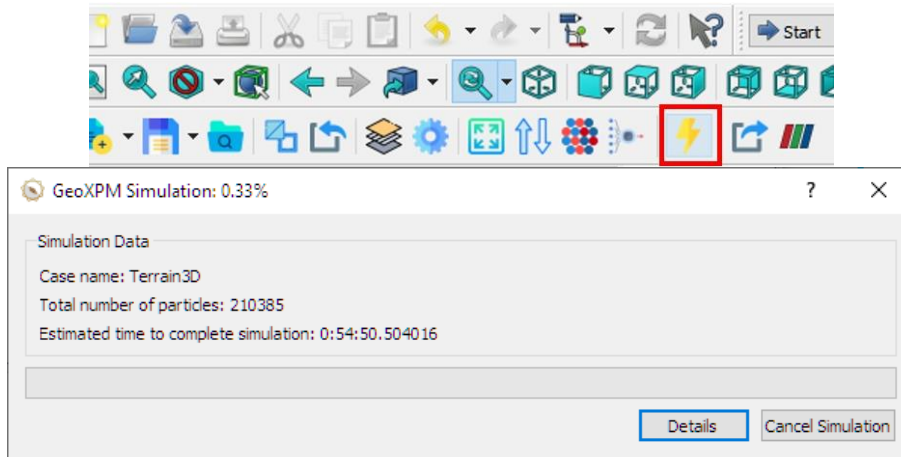

Save the project as 3D
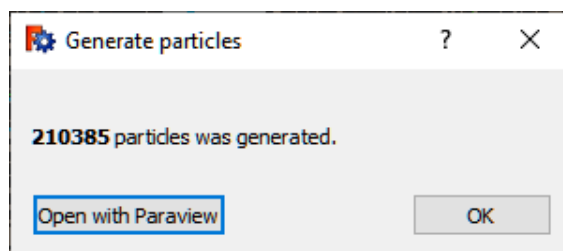
Check the particle configuration

Hint: Typically, SPH boundaries require a minimum of 2-3 consecutive layers to be effective. Inadequate thickness of the boundaries can result in them being penetrated.

➢ Step 9. Start the Calculation

Click the "Run" button and start the calculations.

➢ Step 10. Visualization

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 customized and visualized from here.
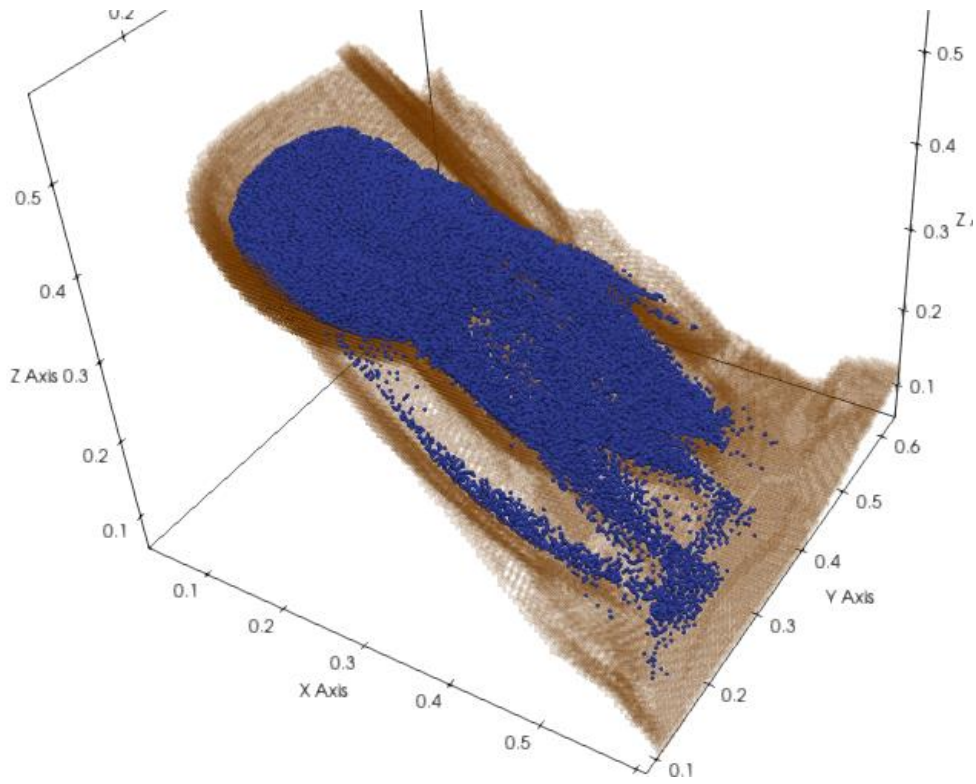
Hint: If you activate the "Adaptive simulation domain" in step 3, the simulation will only generate particles around the soil domain.

Bonus 1: Animation sequence in Paraview

We can use the Paraview Animation view to create an animation sequence with a moving camera point of view.

a. In ParaView, click the eye icon next to Boundaries in the "pipeline Browser" to hide Boundaries particles.

b. Click the "Open" button ( ) and go to the "*_out" folder of the project, and open the "*__Actual.vtk" object → Click "Apply" in the "Properties" tab.
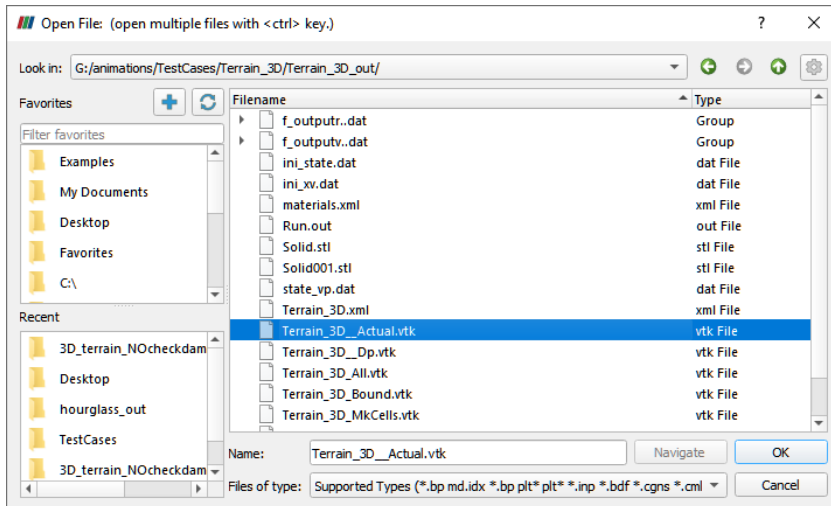

Hint: Before creating an animation, you can adjust the scaling size (Properties tab, Point size in Styling section) and particle colour scale (Properties tab, Colouring).

c. Go to "View" on the menu → Animation View to activate the "Animation View" tab.
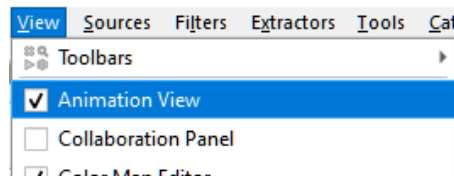
d. Adjust the initial View and Initial timestep in the "Animation View" tab. Select Camera and Interpolate cameras, then click the Plus icon to add the rendered view.
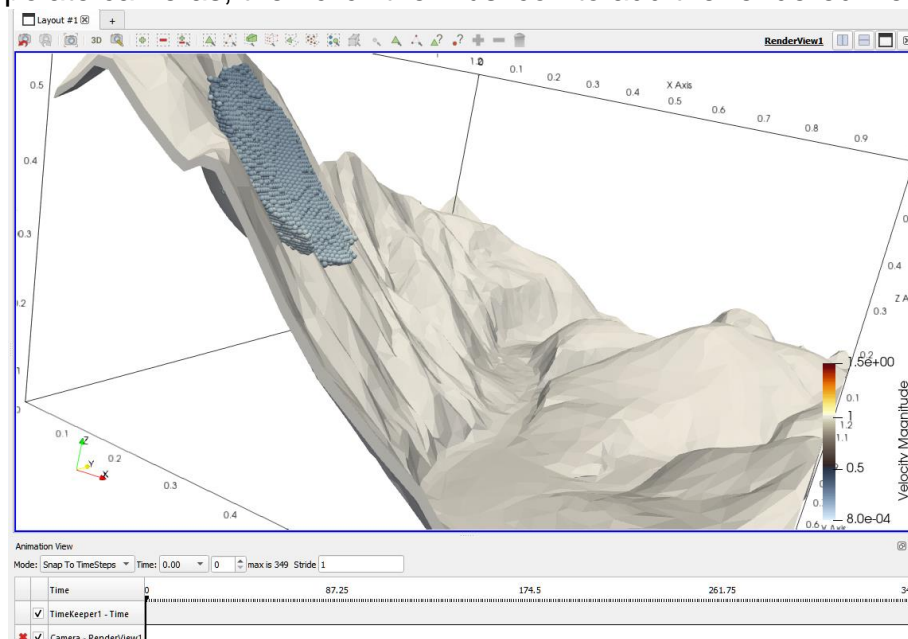
e. Double-click on the whitespace next to "Cemera-RenderView1" to open the "Animation Keyframes" dialogue. In the "Animation Keyframes" dialogue, double-click on "Position …." of the first keyframe (1st row) and click "Use Current" → OK → OK.
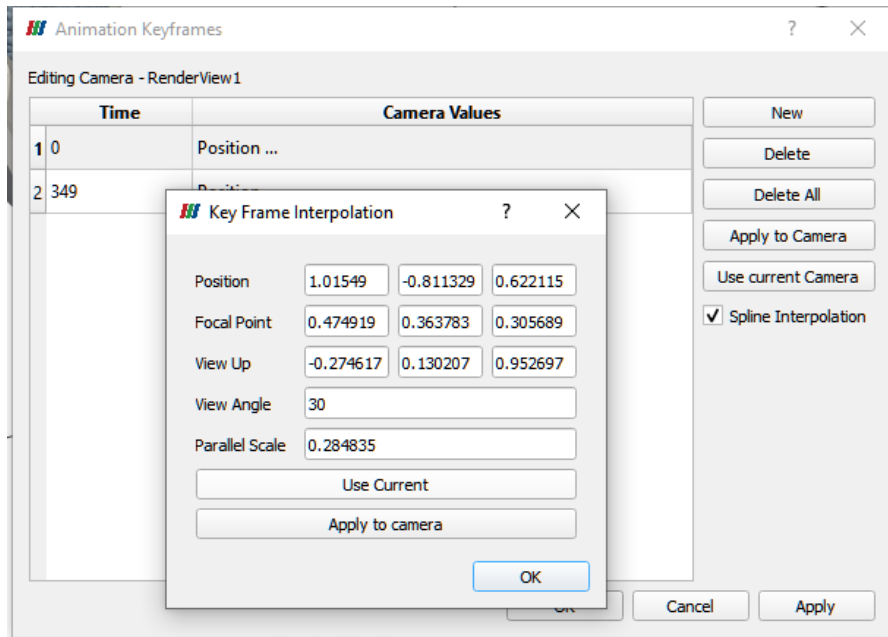
f. Adjust the final view in the last frame. Double-click on the whitespace next to "Cemera-RenderView1" to open the "Animation Keyframes" dialogue. In the "Animation Keyframes" dialogue, double-click on the "Position …." of the last keyframe (2nd row) and click "Use Current".
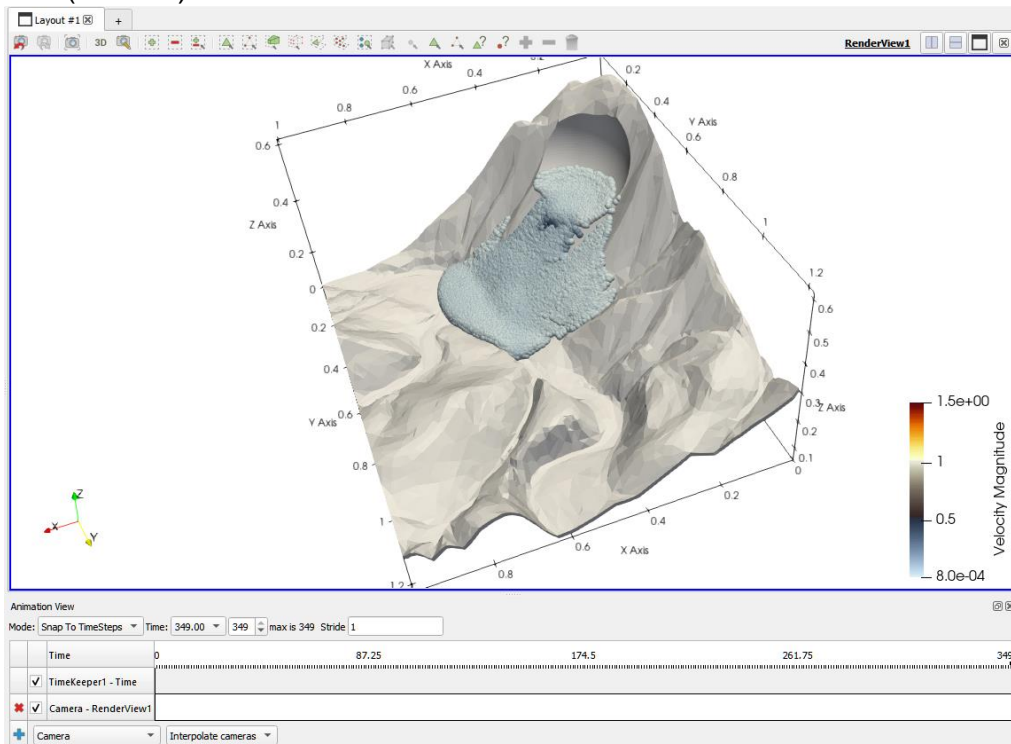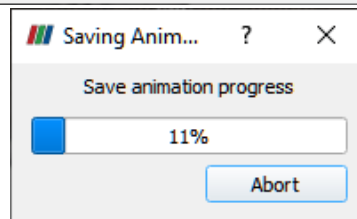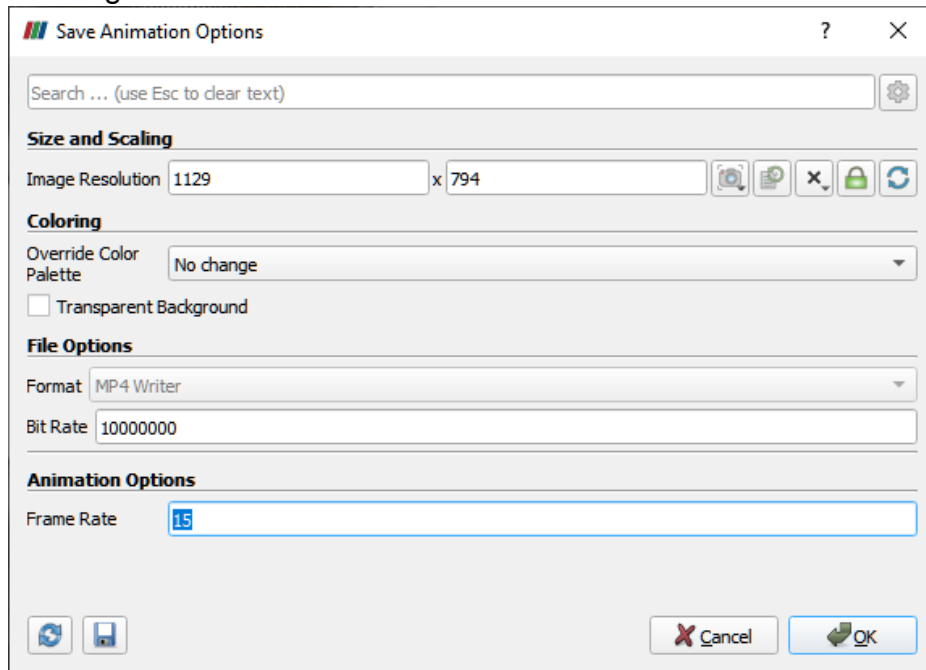

Hint: You can add as many keyframes as needed in the intermediate timeframe.

g. You can now press Play to test the animation.

h. Go to "File" on the menu → "Save Animation". In the "Save Animation Options", input the desired Frame Rate, Bit Rate and Image Resolution → Press OK and wait for the rendering to finish.

Bonus 2: Getting 3D terrain from around the world.

You can follow this video to convert available online point clouds data to 3D mesh files:

https://www.youtube.com/watch?v=0DRiqX20-68

Or even extracting Google Maps information and converting to 3D mesh files using Blender (Including even roads and building information):
https://www.youtube.com/watch?v=7vhV5kKe66g

Note: Using Blender, you can even extract textures and import them into Paraview for better animation and rendering. Imagination is the limit!

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