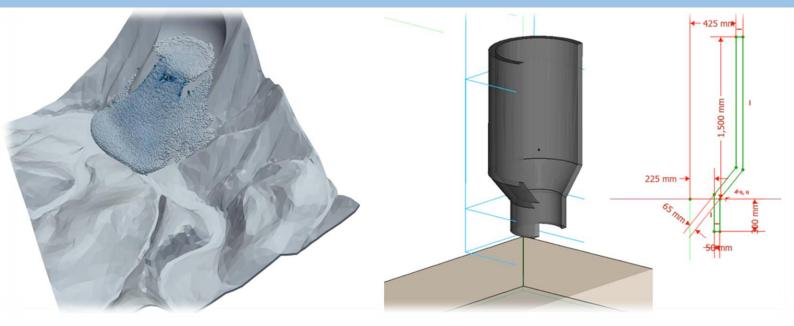




Particle-based continuum solutions for extreme geoengineering, geomechanics & geophysics applications



# GeoXPM Tutorial Manual

Ha H. Bui Tien V. Nguyen Giang D. Nguyen

### Introduction

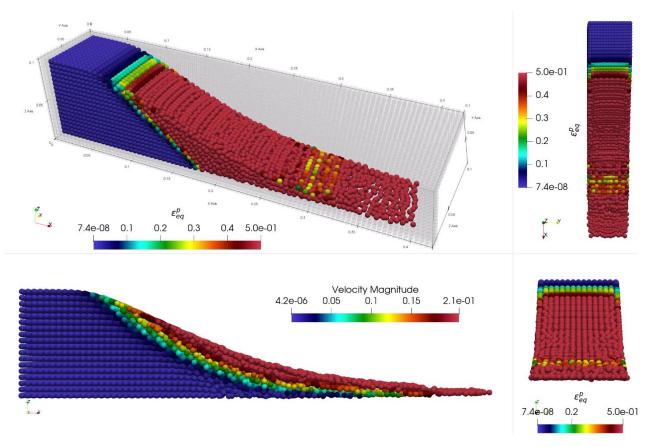
GeoXPM is a new generation of computational software package that was originally developed by Prof Ha H. Bui and his research team at Monash University in Australia based on the Smoothed Particle Hydrodynamics (SPH) method. It has been developed specifically to address the ever-evolving demands of solving intricate geotechnical engineering, geomechanics and challenges. particularly those involving fracturing, geophysics 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 Granular Flow



In this project, we will model a simple granular flow. You will learn about using:

- Basic drawing with primitive objects
- Basic functions of GeoXSPH, including Meshing, Staging, Material models and Visualisation

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.

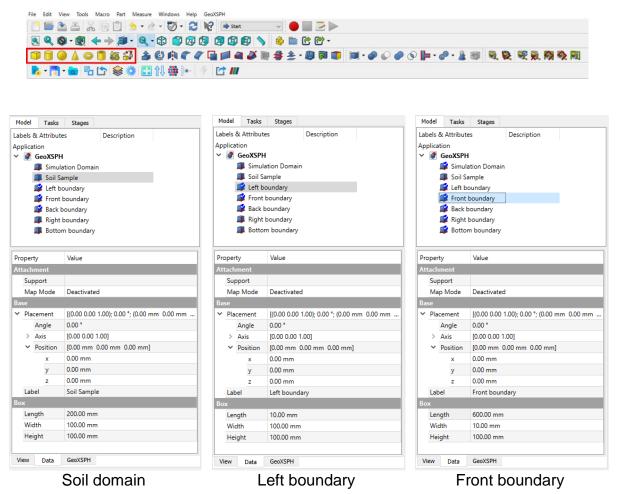
### Step 1. Create a new project

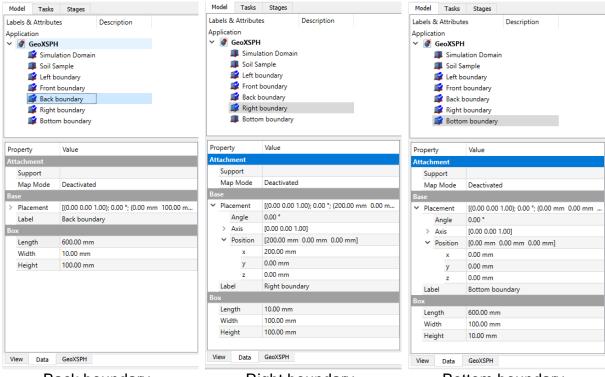
- a. Click the "New Tool" button to create a new project as the given dialogue below.
- b. Enter the project name and project description.

File Edit View Tools Macro Part Measure Window	s Help GeoXSPH		
📑 🖕 🏝 👗 🕞 📋 🍮 • 🕭 • 🐯	- 🔁 🎼 🕩 Start 🗸 🔴 📰 🔁 🕨		
🖲 🍳 🕲 · 🛃 🔶 🔿 🖉 · 🔍 · 🛱 🧯	) 🖾 🖄 🕼 🕼 🕼 🔌 🗎 🕑 🖓 ·		
■ 🗋 🥥 🛦 🗢 🗂 編 🐉 🍐 🙆 🕀		🗐 • Ø Ø Ø 🖗 • Ø •	A 39 9. 9. 9. 9. 9. 9. 9. 10
💦 • 📑 • 📷 🕾 🖆 🏶 🔛 îl 🏶			
	Reproject and constant definition	? ×	
	Project Name:	Granular flow	
	Project Description:		
	3D Granular flow simulation		
	Gravity (m/s <sup>2</sup> ):	-9.81	
	Fluid reference density (kg/m <sup>3):</sup>	1000	
		ОК	

### Step 2. Create initial geometry

- a. Using the "Primitives" toolbox (Cube tool of soil (named "Soil Sample" object) and Boundaries (named "Bottom boundary", "Front boundary", "Back boundary", "Left boundary" and "Right boundary" objects).
- b. Define the dimensions and positions of each object.





Back boundary

Right boundary

Bottom boundary

### > Step 3. Input SPH parameters

a. Enter the SPH parameters into the dialogue.

File Edit View Tools Macro Part Measure Windo			
	🖔 - 🕄 隆 Þ Start 🗸 🔴 🛄 🖻		
	N <i>C C</i> <b>□ □ □ 0 0 0 0 0 0 0 0 0 0</b>		
🔥 • 📑 • 🖮 🕾 🖆 😂 🄃 🄃 🖗	₿• %  <b>% C*</b> ₩		7
	🐘 Execution parameters	? ×	
	Interaction kernel:	Cubic spline $\checkmark$	
	Approximation algorithm:	Type 2: $\sigma_i/\rho_i^2 + \sigma_j/\rho_j^2  \lor$	
	Particle distance (m):	0.005	
	Critical timestep factor (CFL):	0.2	
	Time out data (iter):	200.0	
	Adaptive simulation domain:		
	Load initial stress:	Browse	
	Artificial stress		
	Enabled:		
	Eps: coefficient:	0.5	
	Particle shifting		
	Enabled:		
	Detection algorithm:	Kernel gradient $\sim$	
	Free surface threshold(<=1):	0.8	
	Diffusion coefficient:	2.0	
	Artificial viscosity		
	Alpha coefficient:	0.1	
	Beta coefficient:	0	
	Eta <sub>o</sub> coefficient:	0.1	
	Autostop: (For plastic Stage)		
	Enabled:		
	Normalized velocity(v/dp) lower bound:	0.0001	
	L		
		OK Cancel	

### > Step 4. Input material properties

a. Click the Material property button, select the used constitutive model (i.e., Elasticperfectly plasticity (DP) model) provided in the "new material" button, and enter the material parameters.

Edit View Tools Macro Part Measure Windows Help GeoXSPH						
) 🔚 🏝 📇 😹 📄 📋 🧆 • 🖉 • 🐷 • 🔀 🙀 🗭 Start	v 🔴 🔲 🖻					
Q Q - 🕅 🗲 🔿 🔍 - Q - B 🗇 🖾 🛱 🔇 🛇	🔌 🖿 🕐 🕐	-				
I 🗍 🥥 🛦 🗢 🗂 🍇 🦨 🍐 🕼 🦿 🗳 🖉 🗎 🖉 🖉 🖉	🛊 🛓 · 🔘 🖗	I 🗊 🗐 • 🥔 🖉 🙆	🖓 ⊨ • 🥔 • 🎥 I	5		
• 👘 • 📷 🕾 🖆 🐼 🏶 🔛 🄃 🗰 🐓 👉 💷						
Materials configuration				?	×	
Materials		Material Properties				
New Material		Model:	Drucker-Prager			
Soil material	Û	Young Modulus (N/m2):	840000.0			
		Poisson Ratio:	0.3			
		Friction Angle:	19.8		<b>F</b>	
		Dilatancy Angle:	0.0		-	
		Strength reduction factor:	: 1.0			
		Cohesion (N/m2):	0.0			
		Unit Weight (kg/m3):	2650.0			

**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 object: Select the "Soil sample" in the "Model" tab and click "Add to Simulation" under the "GeoXSPH" tab. And then, select the "Type of object" as "Soil" and the meshing mode as "Full". In the next step, click the "Assign" button to assign the "Soil material" to the soil in the dialogue of the material configuration.

Model Tasks Stages	Model Tasks	Stages					
Labels & Attributes Description Application Constraints	Soil S Left b Front Back Right						
	Property Type of object Meshing mode	Value Soil Full	4	~			
	Faces configurat	ion	Configure		Material configuration		? X
	Material		Assign	6	Select a material: Soil material V	7	Object Id: 0
	Boundary type	No-slip				'	Object Id: U
	Autofill	🗌 Enabl	led		Model: Drucker-Prager Unit weight: 2650.0		
	Object ID	0		\$	Young Modulus: 840000.0		
Add to Simulation 3					Poisson Ratio: 0.3 Friction Angle: 13.8 Dialatancy Angle: 0.0 Cohesion: 0.0 Strength reduction factor: 1.0		
		Remove	from Simulation				
View Data GeoXSPH 2	View Data	GeoXSPH				Cancel	OK

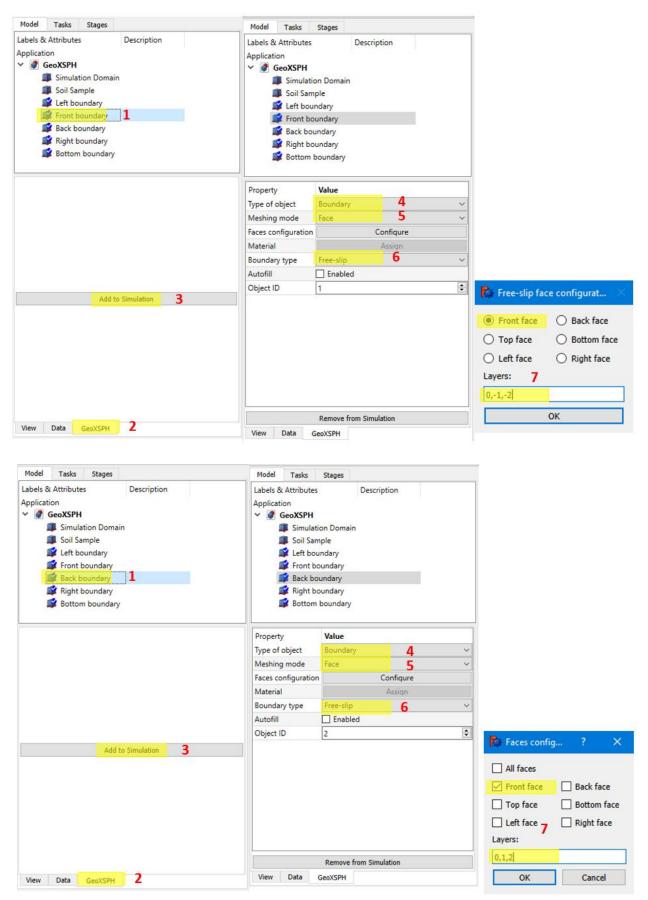
b. "Bottom boundary" object: Select the "Bottom boundary" in the "Model" tab and click "add to Simulation" under the "GeoXSPH" tab. And then select the "Type of object" as "Boundary", meshing mode as "Face", and the "Boundary type" as "No-slip". In the next step, click the "Configure" button to define the boundary surface in the dialogue of the face configuration.

Model Tasks Stages	Model Tasks	Stages			
Labels & Attributes Description Application GeoXSPH Simulation Domain Soil Sample Left boundary Sack boundary Back boundary Bottom boundary Bottom boundary	Labels & Attributes Application Ø GeoXSPH Soil Sam Left bou Front bc Back bo Wight bc Bottom	Desc on Domain nple indary pundary undary pundary	ription		
	Property	Value			
	Type of object	Boundary	4	~	
	Meshing mode	Face	5	~	
	Faces configuration		Configure		
	Material		Assign		
	Boundary type	No-slip	6	~	
	Autofill	Enabled			
	Object ID	4		\$	🔯 Faces config ? 🛛 🗙
Add to Simulation 3					All faces Front face Back face Top face ØBottom face Left face Right face Layers: 7 0,-1,-2
		Remove from Sin	nulation		
View Data GeoXSPH 2	View Data	GeoXSPH			OK Cancel

c. "Left boundary" object. Select the "Left boundary" in the "Model" tab and click "Add to Simulation" under the "GeoXSPH" tab. And then select the "Type of object" as "Boundary", meshing mode as "Face", and the "Boundary type" as "Free-slip". In the next step, define the boundary surface in the dialogue of the face configuration.

Combo View & X	Model Tasks	Stages					
Model     Tasks     Stages       Labels & Attributes     Description       Application     ✓       ✓     Ø GeoXSPH       ■ Simulation Domain       ■ Soil Sample       Ø Left boundary       Ø Back boundary       ■ Right boundary       ■ Bottom boundary       ■ Bottom boundary	Labels & Attribute: Application	s tion Domain	Description	n			
	Property	Value					
	Type of object	Boundary	-	4	~		
	Meshing mode	Face		5	~		
	Faces configuratio	n	Con	fiqure			
	Material		As	sign			
	Boundary type	Free-slip		6	~		
	Autofill	Enable	d				
	Object ID	0			\$	Distance in the second	
Add to Simulation 3						Free-slip fac	ce configurat
						O Front face	O Back face
						O Top face	O Bottom face
						Left face	O Right face
						Layers: 7	0
						0,-1,-2	
		Remove f	rom Simulatio	n			
View Data GeoXSPH 2	View Data	GeoXSPH					ОК

d. Repeat step c for the rest of the free-slip boundary objects, including: "Front boundary", "Back boundary" and "Right boundary".



Model Tasks Stages	Model Tasks	Stages			
Labels & Attributes Description Application Soil Sample Left boundary Right boundary Back boundary Left boundary Back boundary Left boundary Back boundary Description Descrip	Labels & Attributes Application GeoXSPH Simulati Soil Sam Left bou Front bc Back bo Right bc Bottom	on Domain nple undary undary undary bundary	ption		
	Property Type of object Meshing mode Faces configuration Material Boundary type Autofill	Free-slip	4 5 Configure Assign 6	× × ×	
Add to Simulation 3	Object ID	3		÷	🍢 Faces config ? 🛛 🗙
View Data GeoXSPH 2	View Data	Remove from Simu GeoXSPH	ulation		All faces Front face Back face Top face Bottom face U Left face Right face Layers: 7 0,1,2 OK Cancel

#### > 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. Set Stage 0 for the stress initialisation using the elastic loading method. Set the "Type of calculation" to "Elastic", "Stress initialisation method" to "Elastic loading" and the "Time of simulation" to 8000.
- d. Tick the checkbox next to "Reset displacement to 0" to reset displacement after stress initialisation.
- e. Add Stage 1 for the elastoplastic analysis. Click the checkbox "Activated objects" to activate all objects in the simulation, then untick "Right boundary" to deactivate (remove) the right wall in this stage.
- f. Set Stage 1's "Type of calculation" to "Plastic" and the "Time of simulation" to 30000.

Model Tas	sks Stages			
Add Stage	Remove Last St	tage		
✓ STAGE 0				
Y 🗹 Act	tivated objects		Boundary type	
	Soil Sample			
	Left boundary		Free-slip	$\sim$
	Front boundar	у	Free-slip	$\sim$
	Back boundary	/	Free-slip	~
	Right boundar	у	Free-slip	$\sim$
	Bottom bound	lary	No-slip	$\sim$
Time o	f simulation:		8000	
🗙 Туре о	f calculation:		Elastic	$\sim$
✓ Str	ess initializatior	method:	Elastic loading	$\sim$
	Damping:		0.02	
	Reset displace	ments to 0		
✓ STAGE 1				
	tivated objects		Boundary type	
	Soil Sample		<b>F F</b>	
	Left boundary		Free-slip	~
	Front boundar	у	Free-slip	$\sim$
	Back boundary	/	Free-slip	~ ~ ~
	Right boundar	у	Free-slip	$\sim$
	Bottom bound	lary	No-slip	$\sim$
Time o	f simulation:		30000	
Туре о	f calculation:		Plastic	$\sim$

### > 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.

File Edit	View	Tools Mac	ro Part	Measure	Window:	s Help	GeoXSPH	4																
P 🖆	2			<b>5 -</b> d	- 🖏	- 2	12	Start		~			Þ											
	<b>0</b> - 1	🗶 🔶	-> 🔊	- Q		) 🐼 I	3 🗊		) 📏	6	•	• 🕑 •												
		0	1 🚋 🔓	1	6 🏚	<b>a</b>			<b>X</b>	\$	ا - ا	) 🗊 🕻		- 🌒	00	0	<b> -</b> - d	- 8	1	9. 0	R .	1. 🕅	🌒 🕅	0
-	<b>-</b>	5		۵ 🕸	îî 🗱	]•-	1																	

🏟 Meshing order configuration	?	×
Soil Sample	A.	<b>&gt;</b>
Left boundary	*	<b>V</b>
Front boundary	*	<b>¥</b>
Back boundary	*	*
Right boundary	*	<b>V</b>
Bottom boundary	*	4
ОК		
UK		

Meshing order configuration

### > Step 8. Generate particles for computations

- a. Before meshing, click the "Save" button (<sup>1</sup>) to save the project as 2D or 3D based on your need.
- b. Click the "Autofit simulation domain" button () to fit all simulated objects in the Simulation domain.
- a. Click the "Generate particles" button( ) to mesh the computational domain.
- b. After meshing, click "Open with Paraview" to view the particle configuration.

🔯 3D/2D Switching	?	×
Save case as 2D: Cutting section Y (mm): 0.0		
Apply Save	Can	cel

🔯 Generate particles	?	×
31761 particles was generated.		
Open with Paraview	Oł	C
	(*	

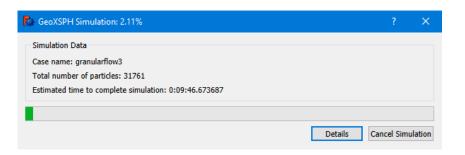
Save the project as 3D

Check the particle configuration

### > Step 9. Start the Calculation

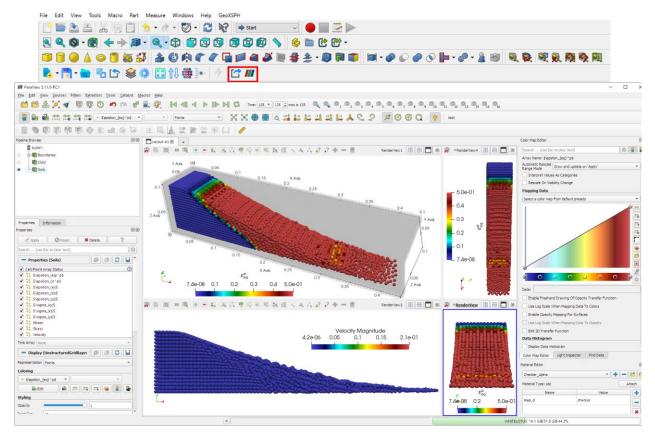
Click the "Run" button and start the calculations.

File	Edit	View	Tools	Macro	Part	Measure	Windows	Help	GeoXSPH	-						
	-	2	5 2		Û	5-0	- 🖉	3	12	i Start		~	•			
	Q	0.		← →		- Q	3	1 🐼 1	9 6		7 💊	1		6	er -	
		0 4	0		5	1	6 🏚	<b>a</b>			<i>🍑</i> 🔳	#	*	- 🔘	। 🗊 🗊 🗑 🏉 🌑 🌔 🗭 խ + 🗬 - 🎍 🗐 🔍 💱 💱 🎘 🦓 🖏 🕅	
P.	-	- 🗖	1		8	<b>*</b> 53	îl 🏶		1							



### > 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.



### **References:**

- Bui H.H, Nguyen V.T & Nguyen G.D (2023). GeoXPM: Particle-based continuum solutions for extreme geoengineering, geomechanics & geophysics applications, Tutorial Manual. <u>Link</u>
- 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
- 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 elasticplastic soil constitutive model, International Journal for Numerical and Analytical Methods in Geomechanics, Vol.32(12), pp.1537-1570. <u>Link</u>