THMC

(Thermal Hydrology Geo-Mechanics Reactive Chemical Model)

User Manual

Contents

1. Benchmark Problem 1	3
1.1. Geometry Mesh GUI	5
1.2 Model Parameter Input & Simulation GUI	22
1.3 Visualization GUI	34
2. How to use the Site Hydrology Information GUI?	?

1. Benchmark Problem 1

Figure 1.1 shows the screen that appears when you start THMC. The first screen is composed of of 6 menus (Project, View, Tools, Settings, Language, and Help) located at the top of the window, and 4 GUIs (Site Hydrogeology Information GUI \swarrow , Geometry Mesh GUI \circledast , Model Parameter Input & Simulation GUI \blacksquare , and Visualization GUI sc) positioned on the upper left side of the window. The Site Hydrogeology Information GUI is a GUI that generates meshes using shape files produced from GIS and borehole data. The Geometry Mesh GUI allows for the manual configuration of the mesh and designates nodes, elements, and element sides for initial and boundary conditions. The Model Parameter Input & Simulation GUI is for entering material properties, initial values, boundary values, time step sizes, and information necessary for numerical simulations, and it also conducts the numerical simulation. Lastly, the Visualization GUI outputs the numerical simulation results in the form of graphs or contours or in the form of an ascii file. To get detailed input information about the Benchmark problems, one can select 'Project' \rightarrow 'Benchmark' as shown in Figure 1.2.



Figure 1.1. The THMC start screen.

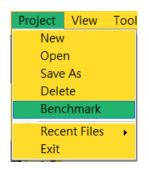


Figure 1.2. Project menu.

To start a new project, select 'Project' \rightarrow 'New'. Then, enter the project name. For Benchmark problem 1, since it is a 2D numerical simulation, select 'THMC2D' in Model as shown in Figure 1.3. Then, click 'Confirm'.

🖳 Create New Proje	ect	-		\times
Project Name	EX1			
Model	THMC2D		~	
Confirm		Car	cel	

Figure 1.3. Create New Project.

Next, if you click on the Geometry Mesh GUI positioned on the upper left side of the window, the screen appears as shown in Figure 1.4. The Geometry Mesh GUI includes 6 menus (File, Geo Setting, Consist, 3D Setting, Tools, and Canvas Setting).

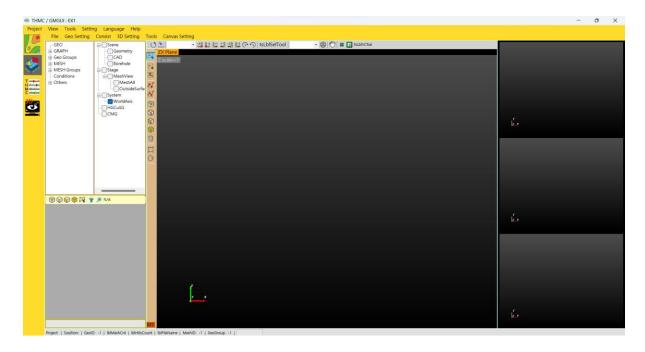


Figure 1.4. Geometry Mesh GUI.

1.1 Geometry Mesh GUI

The Geometry Mesh GUI consists of one panel on the left (Figure 1.5), a large window in the center (Figure 1.6), and three smaller windows on the right. As seen in Figures 1.5 and 1.6, the Panel on the left side and the Window in the center include many tools. The Window in the center has 12 tools on the left side (Add Selection \square , Minus Selection \square , Toggle Selection \square , Select in Geometry Nodes, Add new Node with Right Click in 'Edit Mode' \square , Select Geometry Lines \square , Select Nodes \square , Select Edges \square , Select Surfaces \square , Select Volumes \square , Delete all Selected \square , Square \square , Regular Polygon \square), and 15 tools on the top (Reset unselected species/reaction to selected \square , Save Solution \square , Solid/Wireframe \boxed{Solid} , YZ Plane, Set view direction to +X \square , YZ Plane, Set view direction to -Z \square , Rotate

90° clockwise (C, Rotate 90° counter clockwise (O, ObjectMode/EditMode ObjectMode , Mesh Generator (C, Labeling Vertex **#**, and Labeling Cell (D).

- GEO	∃- Scene
GRAPH	Geometry
🖶 Geo Groups	CAD
- MESH	Borehole
- MESH Groups	Stage
Conditions	MeshView
⊕ Others	MeshAll
	OutsideSurface
	System
	WorldAxis
	HGCuSG
	CMG
🗑 🖓 📦 🔀 🝟)
V V V V K	-

Figure 1.5. Panel on the left side in the Geometry Mesh GUI.

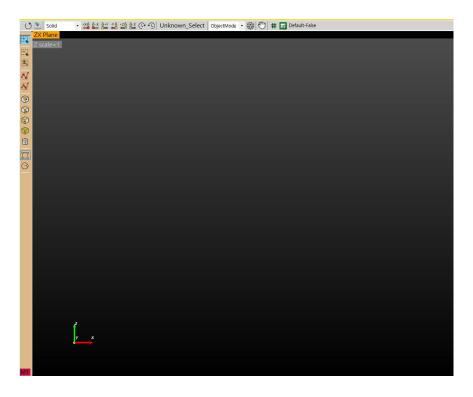


Figure 1.6. Large window in the center in the Geometry Mesh GUI.

1. To start a new solution, select 'File' \rightarrow 'New or Open Solution' as shown in Figure 1.7.

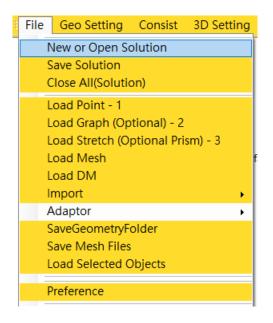


Figure 1.7. File menu.

2. In the 'New or Open Solution' window, enter the solution name and click 'Create New' as shown in Figure 1.8.

🖳 New or (Open Solution		- 🗆 X
New	Current Project Name: EX1	Create New	
	Existed Project Name:	Existed Solution Name:	
O Open	Dummy		Open Exist
			Delete Solution
		Restore Option	
		Restore Geometry Objects Restore Mesh Objects Restore Mesh Objects	
		E restore mean objects C restore mean mea	

Figure 1.8. New or Open Solution.

3. To set the interest area, click on the 'Square' tool include on the left side of the window. A panel will then appear as shown in Figure 1.9. After entering W=50 and H=200, click on the 'Preview' button followed by the 'Create' button. When you click the 'Create' button, an area of interest measuring 50 cm by 200 cm appears on the window as shown in Figure 1.10.

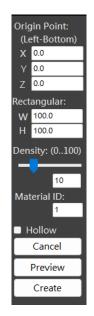


Figure 1.9. Square tool for setting the area of interest.



Figure 1.10. The area of interest.

4. Click on the 'Mesh Generator' tool ⁶⁶ located at the top of the window. In Benchmark problem 1, the area of interest is composed of rectangular elements. Therefore, in the Mesh

Generator window, click on 'Quadrilateral'

💀 Mesh Generator			×
Proj: EX1 Sol: Build_0824_1716_23		Surface mesh nodes elevation from DTM	
🔂 Triangle	Triangle Prism	All mesh nodes offset from DTM	
Quadrilateral	D Quadrilateral Prism	Y-Z Axis Transpose Setting: ESRI Shape File Import: No	
Preview Text Files	Tetrahedron (NetGen)	Meshgen Pre-Process (2D only): No	
Triangle ~	Preview Clear Result	Meshgen Post-Process: Configure	
Result Preference Geometry.bd (Purge old generation files be Auto reload mesh file after es Add Boundary Points Rat Add Interior Points Ratio Quality Index (Skewness) Quadrilateral Mesh (2D/3D Add Boundary Points Rat Quality Index (Skewness) Quadrilateral Prism Genera GN/GE Sequence Comp For Example 15, 16 and	xecution neration Setting (Beta): (B	th Output Map	

5. After clicking on 'GEO (d1, n4)' in the Panel located on the left and then selecting 'domain 1', you can see that 4 Edges and 4 BGs have been created, as shown in Figure 1.14.

6. In Benchmark problem 1, since the area of interest is divided into 41 nodes in the z-direction, right-click on BG1 and input the 'Set Nodes Count' to 41. Then, click again 'Set Nodes Count' button as shown in Figure 1.15.

7. In Benchmark problem 1, since the x-direction consists of 2 nodes, right-click on BG2 and set the 'Nodes Counts' to 2 as shown in Figure 1.16.

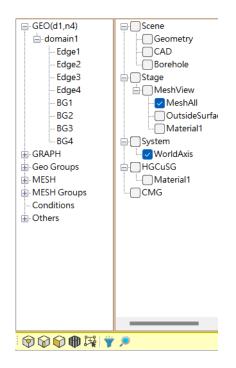
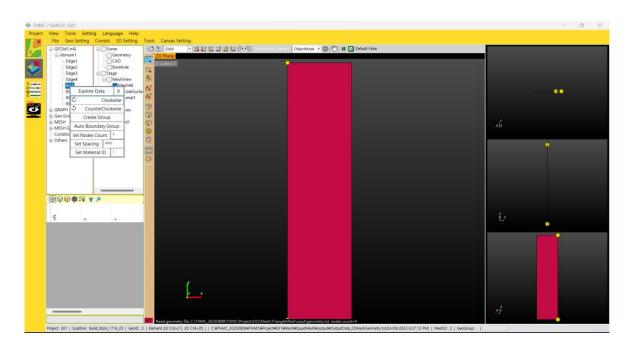
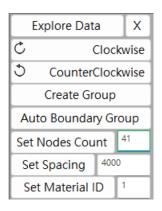


Figure 1.14. Domain 1 composed of 4 Edges and 4 BGs.



(a)



(b)

Figure 1.15. Divide BG1 into 41 nodes.

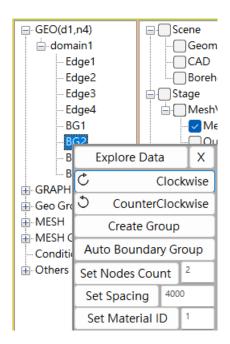


Figure 1.16. Divide BG2 into 2 nodes.

8. Click on the 'Mesh Generator' tool is located at the top of the window, and in the Mesh Generator window, click on 'Quadrilateral' Quadrilateral.

9. The result of dividing the area of interest into 41 nodes in the z-direction is illustrated in Figure 1.17.

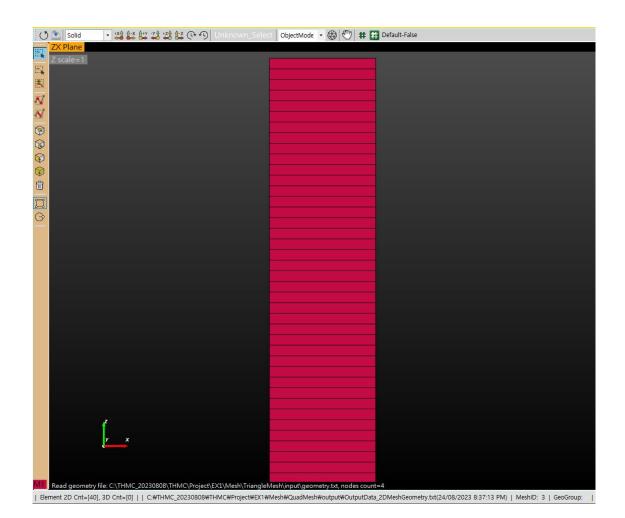


Figure 1.17. Divide the area of interest into 41 nodes in the z-direction.

10. If the mesh has been created as shown in Figure 1.17, one would want to set initial and boundary conditions for the area of interest. In Benchmark problem 1 for the initial condition, since different values were input for the top, bottom, and the rest of the area of interest, we will specify initial conditions for each of these three parts separately. First, with 'Add Selection' selected on the left side of the window, choose 'Select Nodes' .

11. To select the nodes located at the top of the area of interest, drag to select the two top nodes. The window screen appears as shown in Figure 1.18.

🕐 🏝 Solid	• 24 些 些 24 29 经 @ 49 Mesh_Select	ObjectMode • 🍪 🖑 🕯	# 😨 Node-True
ZX Plane Z scale=1		<u></u>	
		¥	<u> </u>
B			
24			
<u>×</u>			
B			
Z Plane Z scale = 1 I I			
G			
	Z		
	y x		
M1 Read geometry	file: C:\THMC_20230808\THMC\Project\EX1\Mesh\Triangle	Mesh\input\geometry.txt, no	nodes count=4

Figure 1.18. The window screen after the two top nodes have been selected.

12. In the panel, select 'CreateGroup' 🔀 as shown in Figure 1.19.

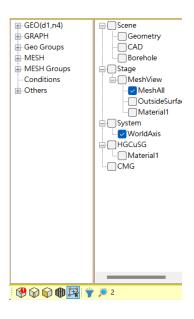


Figure 1.19. Panel view.

13. Then, as seen in Figure 1.20, 'Create Group' window appears.

💀 Create Group	×
 I. *Shape Type ● Point ○ Line ○ Face ○ Volume 	Reset
II. Conditions:	
1. O 2D () 3D	
2. Simulation Category:	~
3. Condition Name:	~
🕹 As	sign to Group Name
III. *Group Name and Number	
N	umber(Optional): 1
*Add (from Canvas)	
IV. *Selected Object Keys	
Count: 2	Import Range:
41 82	1 ~ 20000
	Import (copy paste)
	Remove All
	Export
Apply and Close Apply	Close

Figure 1.20. Create Group Window.

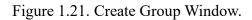
14. In Figure 1.20, click on the list box next to '2. Simulation Category:' and select 'Hydrological'. Then, click on the list box next to '3. Condition Name:' and choose 'Initial'. Then, click on 'Assign to Group Name'.

15. In Figure 1.20, if you enter 'top' into the text box under 'III. *Group Name and Number', the result appears as shown in Figure 1.21.

16. Then, click on 'Apply and Close'.

17. Then, as seen in Figure 1.22, you can observe that 'Hydrological_Initial_top' has been added under the 'Conditions' subcategory.

🛃 Create Group		×
I. *Shape Type Point O Line O Face O	Volume	Reset
II. Conditions:		
2. Simulation Category:	Hydrological	~
3. Condition Name:	Initial	~
	🛓 Assign	to Group Name
III. *Group Name and Number		
Hydrological_Initial_top		
	Numl	ber(Optional): 1
*Add (from Canvas)		
IV. *Selected Object Keys Count: 0		1 ~ 20000
		Import (copy paste)
		Remove All
		Export
Apply and Close A	pply	Close



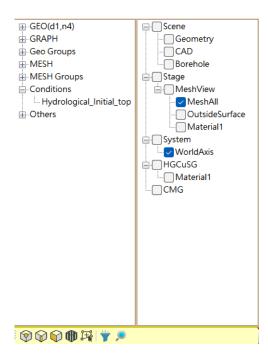


Figure 1.22. Panel view.

18. To set the initial conditions for the nodes located at the bottom, repeat steps 10 through 16. However, in step 11, select the two nodes at the bottom, and in step 15, enter 'bottom' in the text box under 'III. *Group Name and Number'.

GEO(d1,n4) GRAPH Geo Groups	Geometry CAD Borehole	● ● Sold ・ 認 認 認 認 認 認 認 認 認 認 認 認 認 認 認 和 記 (P+O) Mesh_Select ObjectMode - 級 (*) # 認 Node-True ZX Plane Z scale=1 T scale=1
MESH Grou	🔡 Create Group	X
Hydrolog	I. *Shape Type Point O Line O Face O	Volume Reset
	II. Conditions:	
	1. O 2D 🔿 3D	
	2. Simulation Category:	Hydrological V
	3. Condition Name:	Initial V
		🛓 Assign to Group Name
	III. *Group Name and Number	
	Hydrological_Initial_bottom	
		Number(Optional):
	*Add (from Canvas)	
*	IV. *Selected Object Keys	
	Count: 2	Import Range:
	1 42	1 ~ 20000
		Import (copy paste)
		Remove All
		Export
	Apply and Close Apply and Close	Close Close
		Inead geometry file. C:\THMC_20230808\THMC:Project\EX1\Mesh\TriangleMesh\input\geometry.txt, nodes count=4

19. In the final step of setting the initial conditions, follow the same steps 10 through 16 for the nodes excluding the top and bottom. During step 11, select all nodes except those at the top and bottom, and in step 15, enter 'rest' into the text box under 'III. *Group Name and Number'.

Geometry		< Plane
- CAD	12	
Borehole		scale=1
Stage	8 .	
MoshView	1	
🖷 Create Group		
Create Group		· · · · · · · · · · · · · · · · · · ·
I. *Shape Type		
● Point ○ Line ○ Face ○	Volume	Reset
II. Conditions:		
1. O 2D O 3D		
2. Simulation Category:	Hydrologi	ical ~
	-	-
3. Condition Name:	Initial	~
	📥 Assi	gn to Group Name
III. *Group Name and Number		
Hydrological_Initial_rest		
	Nu	mber(Optional): 1
*Add (from Canvas)		
IV. *Selected Object Keys		
Count: 78		Import Range:
2 3 4 5 6 7 8 9 10 11 12 13 14 15		
2 3 4 5 6 7 8 9 10 11 12 13 14 15 19 20 21 22 23 24 25 26 27 28 29		1 ~ 20000
19 20 21 22 23 24 25 26 27 28 29 3 33 34 35 36 37 38 39 40 43 44 45 4		(Income the Community)
		Import (copy paste)
49 50 51 52 53 54 55 56 57 58 59		
63 64 65 66 67 68 69 70 71 72 73	/4 /5 /6	Remove All
77 78 79 80 81		French
		Export
Apply and Close Ap	ply	Close
the second s		
	M1 Re	ad geometry file: C:\THMC_20

20. Then, we will explain how to set the boundary conditions. In Benchmark problem 1, we set a variable boundary condition for the top and a Dirichlet boundary condition for the bottom. First, to set a variable boundary condition for the top, with 'Add Selection' selected on the left side of the window and 'Labeling Vertex' **#** selected on the upper side of the window, choose 'Select Edges' .

21. To select the element side located at the top of the area of interest, drag to select the top element. The window screen appears as shown in Figure 1.23.



Figure 1.23. Window screen.

22. To verify the selected boundary element sides, click the tool \mathfrak{D} located at the bottom of the panel as shown in Figure 1.24.

23. In the table located at the bottom of the panel view in Figure 1.24, one can observe that the boundary sides (81, 82), (82, 41), and (41, 40) are selected. As indicated in Figure 1.23, the boundary side corresponding to the top boundary side is (82, 41). Therefore, boundary sides (81, 82) and (41, 40) should be removed.

24. As seen in Figure 1.25, select the boundary sides (81, 82) and (41, 40), right-click, and choose 'Remove Selected Mesh Objects'. Then, in the appearing 'Warning' window, click 'OK'.

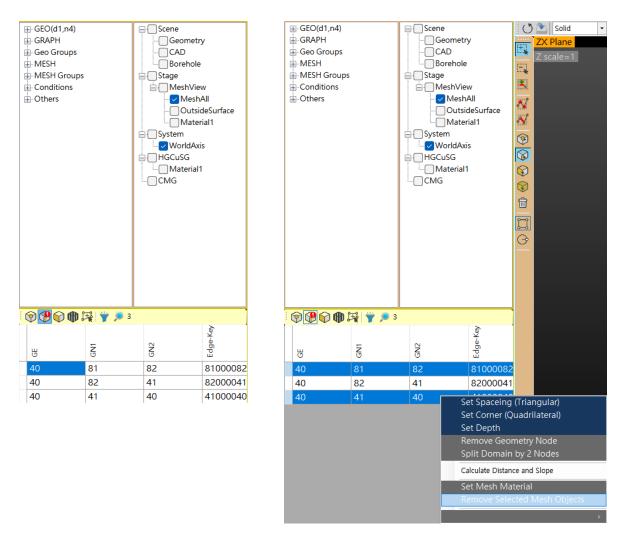


Figure 1.24. Panel view.

Figure 1.25. Panel view.

26. As a result, you can see that the top boundary side is selected, as shown in Figure 1.26.

27. Select 'CreateGroup' tool 🔀 located below the Panel, as shown in Figure 1.27.

28. As seen in Figure 1.28, 'I. *Shape Type' has 'Line' selected. For '2. Simulation Category:', choose 'Hydrological'. Then, for '3. Condition Name:', select 'Variable'. Click on 'Assign to Group Name', and type 'top' under 'III. *Group Name and Number'. Finally, click on 'Apply and Close'. Thus, the variable boundary condition setting for the top boundary side is completed.

29. The process of setting the Dirichlet boundary condition is very similar to the process of setting the initial condition in step 18. The only difference is that in step 14, you should select 'Dirichlet' for '3. Condition Name:' as shown in Figure 1.29.



Figure 1.26. Window screen for setting variable boundary condition.

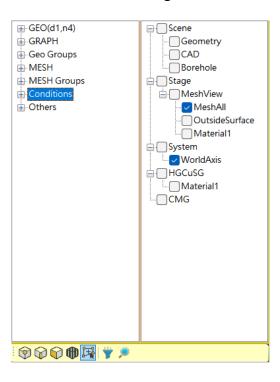


Figure 1.27. Panel view.

🖳 Create Group	:	×
□ I. *Shape Type ○ Point ● Line ○ Face ○ Y	Volume Reset	
II. Conditions:		
2. Simulation Category:	Hydrological V	
3. Condition Name:	Variable ~	
	🛓 Assign to Group Name	
III. *Group Name and Number		
Hydrological_Variable_top]
	Number(Optional): 1	_
*Add (from Canvas)		
IV. *Selected Object Keys Count: 1 40	Import Range: 1 ~ 20000	
	Import (copy paste)	
	Remove All	
	Export	
Apply and Close App	Close	

Figure 1.28. Create Group window for setting variable boundary condition.

🖳 Create Group		×							
I. *Shape Type • Point Line Face \	/olume	Reset							
II. Conditions:									
2. Simulation Category:	Hydrological	\sim							
3. Condition Name:	Initial PointSourceSink	~							
Dirichlet III. *Group Name and Number Number(Optional); 1									
*Add (from Canvas)									
IV. *Selected Object Keys Count: 2 1 42	Import	Range: 1 ~ 20000							
	Import	t (copy paste)							
		Remove All							
		Export							
Apply and Close App	ly	Close							

Figure 1.29. Create Group window for setting Dirichlet boundary condition.

Through steps 1 to 30, we have completed the mesh generation for Benchmark problem 1 and finalized the settings for nodes and element sides for the initial and boundary conditions in the Geometry Mesh GUI. As shown in Figure 1.30, after setting the initial and boundary conditions for Benchmark problem 1, the subcategories are created under the 'Condition' category. You can click on each subcategory to verify if the selection has been made correctly. Lastly, to save the mesh information, initial and boundary conditions set from step 3 to 29, click the Save Solution located on the upper part of the program. Then, in the 'Save Geometry' pop-up window, click 'OK'.

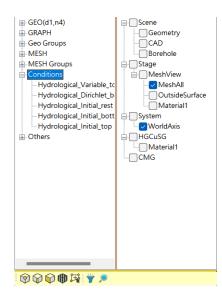


Figure 1.30. After setting the initial and boundary conditions, the subcategories created under the 'Condition' category.

Save Geometry			×						
 Triangle (or To Be DefinedOthers) Triangle Prism 	• Quadrilateral (or Hybrid)	O Tetrahedron	ОК						
		Change	to D:\temp						
Update to the latest changes (Even though the geometry files maybe are not correct enough to generate mesh!) PS: If you don't make sure what kinds of type, please use 'Triangle (or To Be Defined)'									



Now, we'll navig	ate to the 'Model	Parameter	Input	& Sim	ulation	GUI	C	loc	ated	on the
1 0 1 1	N.1 TT	•11	1		. 1	c		. 1		1

upper left side of the program. Here, we will set the time intervals for the numerical simulation, configure the material values, and establish the initial and boundary conditions, and then run the numerical simulation.

1.2 Model Parameter Input & Simulation GUI

The 'Model Parameter Input & Simulation GUI' consists of four tools: 'Import Mesh and Boundary Data' , 'Start Model Simulation', 'Stop Model Simulation', and 'Restart' ; as well as four tabs: 'Parameter Wizard', 'Simulation', 'Global', and 'Hydro' as shown in Figure 1.31.

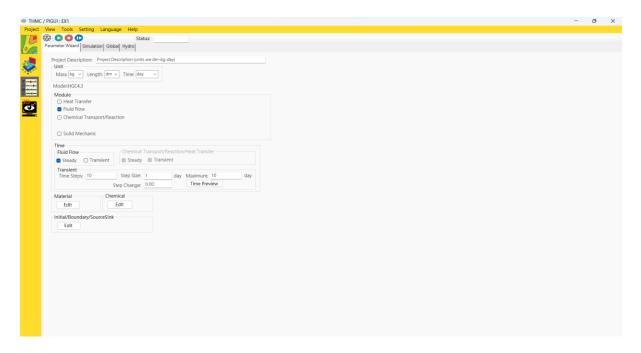


Figure 1.31. Model Parameter Input & Simulation GUI'.

1. To import the mesh information and the initial and boundary condition settings entered in the 'Geometry Mesh GUI', click on the 'Import Mesh and Boundary Data' tool

2. Then, the 'Mesh Import' window pops up as shown in Figure 1.32. Choose 'cm' in unit and click on 'Import'.

3. To proceed with the numerical simulation, select the 'Advanced' option in 'View' menu as shown in Figure 1.33.

4. In Figure 1.34, enter the 'Project Description' and 'Unit' in the 'Parameter Wizard' tab. For Benchmark problem 1, the units of cm, g, and day are used, so select the corresponding units. Then, in the Module, select 'Fluid Flow' and choose the 'Transient' fluid flow option.

Mesh Import			
Import Close]		
Mesh: GM GUI	~		
Unit: cm	~		
MeshType	SolutionName	TimeStamp	GeolD
1 Quad	Dummy	Timestamp	1
> 2 Quad	Build_0824_1716_23	2023-08-24 20:37:13	2
		2023-08-24 20:37:1	8

Figure 1.32. Mesh Import window.

Project	View		Tools	Setting	Lar	nguage	Help		
	~	То	olBar				Statu		
	×	Ac	lvanced		Global Hydro				
	×	Μ	Model Input File						
	~	М	odel Out	tput File		ct Description (un			
	~	М	odel Det	ail Input	cr bescription (units				
	_	_							

Figure 1.33. Check 'Advanced' option.

rameter Wizard Simu	Status:	
Project Description: Unit Mass: g v Le	Project Description (units are cm-g-day)	
Model:HGC4.3		
Module Heat Transfer Fluid Flow Chemical Tran		
Solid Mechan	lic	
Time Fluid Flow Steady Z	Chemical Transport/Reaction/Heat Transf	er
Transient Time Steps: 10	Step Size: 1 day Maximum:	10 day
	Step Change: 0.0D Time Prev	/iew
Material Edit	Chemical	
Initial/Boundary/S	SourceSink	

Figure 1.34. Parameter Wizard tab.

5. In Benchmark problem 1, the initial time step size is set at 0.05 days. Each subsequent time step is increased by a factor of 1.2 until it reaches a maximum size of 1.0 day. However, because there is a sudden shift in the flux value from 5 cm/day (infiltration) to -5 cm/day (evaporation) imposed on the top surface at t = 10 days, the time step size is reset to 0.05 days at that point. From there, it will increase again for each subsequent time step using the same 1.2 multiplier. The maximum time step size will never exceed 1 day, and the simulation will span a total of 20 days. Given these time step parameters, 44 time steps will be required for the simulation. To input the varying time step size, click on the 'Time Preview' button in Figure 1.34. This action will prompt a popup window of 'Simulation Time Setting', as shown in Figure 1.35.

Then, type 'NTIS (total number of the time steps) = 44, DELT (initial time step size) = 0.05, CHNG (multiplier factor - 1.0) =1.2-1.0 = 0.2, DELMAX (maximum allowable time step size) = 1.0, TMAX (maximum simulation time) = 20.0, KOUTSTEP_AUTO (number of time steps between outputs) = 1, TDTCH(I) (time at which the time step size is reset) = 10.0, and ITMSTO(I) = 1'. To check if the time step size has been entered correctly, click the 'Preview' button. Then, click on 'Save' button.

•	Si	Simulation Time Setting									_	×
	S	ave		Close	Preview	·						
		NTIS	DELT	CHNG	DELMAX	TMAX	KOUTSTEP_AUTO	TDTCH(I)	ITMSTO(I)			
►	1	44	0.05	0.2	1	20.0	1	10.0	1			
		Print	Step	Time	Delta							
•	1		1	0.0500	0.0500							- 1
	2	\sim	2	0.1100	0.0600							- 1
	3	\sim	3	0.1820	0.0720							- 1
	4	\sim	4	0.2684	0.0864							- 1
	5	\sim	5	0.3721	0.1037							- 1
	6	\sim	6	0.4965	0.1244							
	7	Image: A start of the start	7	0.6458	0.1493							
	8	\sim	8	0.8250	0.1792							
	9	\sim	9	1.0399	0.2150							
	10	\sim	10	1.2979	0.2580							
	11	 Image: A set of the set of the	11	1.6075	0.3096							
	12	\sim	12	1.9790	0.3715							
	13	\sim	13	2.4248	0.4458							
	14		14	2.9598	0.5350							
	15		15	3.6018	0.6420							

Figure 1.35. Simulation Time Setting window.

6. To input the material properties, click the 'Edit' button under 'Material'. This will bring up the 'Material' window as shown in Figure 1.36.

🖷 Material						_		×
Material						-	U	^
Save Close								
Material 2DExample	10 ~ +							
Material								
▶ 1 default								
Flow Property	 Hydraulic Conductivity 	 Saturated Permeability 	1					
Water Retention Curves	Modified Media Compressibility[1/cm]	Modified Water Compressibility[1/cm]	Saturated Water Content[-]	Hydraulic Conductivity xx[cm/day]	Hydraulic Conductivity zz[cm/day]	Hydraulic Cor xz[cm/day]	ductivity	y
▶ 1 Edit	0	0	0.45	0	10	0		

Figure 1.36. Material window.

7. In Benchmark Problem 1, the domain is assumed to contain soil with a saturated hydraulic conductivity of Kxx = Kxz = 0 cm/day and Kzz = 10 cm/day, a porosity of 0.45, and a field capacity of 0.15. Consequently, when the 'Material' popup window appears, check 'Hydraulic Conductivity' next to 'Flow Property'. Additionally, enter the following values:

- Modified Media Compressibility = 0
- Modified Water Compressibility =0
- Saturated Water Content =0.45
- Hydraulic Conductivity xx = 0
- Hydraulic Conductivity zz = 10
- Hydraulic Conductivity xz = 0
- Fluid Referenced Density = 1
- Fluid Referenced Dynamic Viscosity = 865.7
- Bulk Density = 1.2,

- Residual of Water Content = 0.15,
- Factor Exponent = 0.

8. Click the 'Edit' button located beneath 'Water Retention Curves' as shown in Figure 1.36, then the 'Water Retention Curves' window will pop up as shown in Figure 1.37.

🖳 Water Retention Curves					-	×
Confirm Cancel						
Water Retention Curves(-1	KSP = -1					
Water Retention Curves(KSP)	Description	KSI	P = -1, the following soil j	property functions are used:		
1	KSP = 1, Tabular soil parameter	$\theta =$	$\theta + (\theta - \theta) \frac{h - h_a}{h}$ if	f $h < 0$, $\frac{\theta - \theta_r}{\theta_r - \theta_r} = 1$ if $h \ge 0$	0: $K = \frac{\theta - \theta_r}{\theta_r}$	
▶ 2 -1	KSP = -1		$b_r + (b_s - b_r) h_b - h_a$	$\theta_z - \theta_r$	$(\theta_z - \theta_r)$	
3 -2	KSP = -2		SPP $(1,I,1) = \theta_{\gamma}$, residual moist	ure content ([WCR])		
4 -3	KSP = -3, Brooks and Corey		SPP $(2,I,1) = \theta_z$, saturated mois			
5 -4	KSP = -4, WRR, 15(5), p1089-1101, 1979		$SPP(3,I,1) = h_a([HAA])$			
6 -5	KSP = -5, van Genuchten		$SPP(4,I,1) = h_b ([HAB])$			
Parameter			Draw h-min: -200	h-max: 10	h-step: 5	 _
Parameter Description	Value					
	nt ([WCR],saturated water content [L3L-3]) 0.15					
	ent ([WCS],residual water content [L3L-3]) 0.45					
	to correct air entry pressure [L]) 0					
4 hb ([HAB],bubbling or ai	ir entry pressure [L]) -100					

Figure 1.37. Water Retention Curves window.

9. In Benchmark problem 1, the unsaturated characteristic hydraulic properties of the soil are given as

$$\theta = \theta_r + (\theta_s - \theta_r) \frac{h - h_a}{h_b - h_a}$$
 and $k_r = \frac{\theta - \theta_r}{\theta_s - \theta_r}$

where $\theta_r = 0.15$, $\theta_s = 0.45$, $h_a = 0$ cm, and $h_b = -100$ cm. Therefore, set the following values:

• Water Retention Curves (KSP) = -1

- WCR = 0.15
- WCS = 0.45
- ha = 0
- hb = -100

and click 'Confirm' to close the 'Water Retention Curves' window and click 'Save' and 'Close' buttons to exit the 'Material' window.

10. After completing the input for 'Material', skip the input for 'Chemical' since Benchmark Problem 1 simulates fluid flow only. Then, proceed to the input for 'Initial/Boundary/SourceSink' by clicking 'Edit' in Figure 1.34.

11. When you click the 'Edit' button, the 'Initial/Boundary/SourceSink' window will pop up as shown in Figure 1.38.

Initial/Boundary/SourceSink	-	×
Save Close		
Spatial Type Hydrodlogical/Initial Condition +		
Module Spatial Type Import Mode		
Conditions		
Import Species Species		
Domain		
File Import		

Figure 1.38. Initial/Boundary/SourceSink window.

12. In Benchmark Problem 1, the initial conditions are set as follows: a pressure head of -90 cm is imposed on the top surface of the column, 0 cm on the bottom surface of the column, and -97 cm elsewhere. Therefore, input the initial conditions separately for the top, bottom, and the remaining areas. As shown in Figure 1.39, select 'Hydrological/Initial Condition' in the list box for the 'Spatial Type' and then click the button +. After that, in 'Conditions', select 'Hydrological_Initial_top', 'Hydrological_Initial_bottom', and 'Hydrological_Initial_rest' in sequence. Then, enter the corresponding initial values into 'Initial Pressure' as shown in Figure 1.39.

🖳 In	itial/Boundary/SourceSink					-	×
Sa	ve Close						
Spati	al Type Hydrodlogical/Ele	ment	Source Sink	~	+		
▶ 1	Module Spatial Type Hydrodlogical Initial Conditi	_	Import Mode				
Cond	litions						
	Domain		Initial Pressure[cm]				
1	Hydrological_Initial_top	~	-90				
2	Hydrological_Initial_bottom	~	0.0	1			
⊳ 3	Hydrological_Initial_rest	~	-97				
* 4	Hydrological_Initial_top Hydrological_Initial_bottom Hydrological_Initial_rest Hydrological_Dirichlet_bottom]			

Figure 1.39. Initial/Boundary/SourceSink window.

13. In Benchmark problem 1, the boundary conditions are given as: no flux is imposed on the left and right surfaces of the column; pressure head is held at 0 cm on the bottom surface; and variable condition is used on the top surface of the column with a ponding depth of zero, minimum pressure of -90 cm, and a rainfall of 5 cm/day for the first ten days and a potential evaporation of 5 cm/day for the second ten days. Here, no flux boundary doesn't require any

additional input, so the input for no flux boundary can be skipped. The bottom boundary is set as a Dirichlet boundary with a hydraulic head of 0cm. As shown in Figure 1.40, choose 'Hydrological/Dirichlet Boundary' in the list box from the 'Spatial Type', then click the button +. In 'Conditions', select 'Hydrological_Dirichlet_Bottom' and click 'Edit' button.

😸 Initial/Boundary/SourceSink		-	×
Save Close			
Spatial Type Hydrodlogical/River Bou	indary ~ +		
Module Spatial Type 1 Hydrodlogical Initial Condition 2 Hydrodlogical Dirichlet Boundary	Import Mode		
Conditions			
Domain Hydrological_Dirichlet_bottom 2 Hydrological_Initial_top Hydrological_Initial_bottom Hydrological_Initial_rest Hydrological_Initial_Post	Profile ID Edit		

Figure 1.40. Initial/Boundary/SourceSink window.

14. By clicking the 'Edit' button, the 'ProfileInputForm' window will pop up. Click the button

+ next to 'Profile' and type '0' and '0' for Total Heads in 'Data Points'.

Save Close Profile Selected Profile: Profile Name	
Profile Name	
1	
Data Points Draw	
Time[day] Total Head[cm]	
2 1.0e+38 0	
2 1.0e+38 0	

Figure 1.41. ProfileInputForm window.

15. To set a Variable boundary condition on the top surface, select 'Hydrological/Variable Boundary' from the 'Spatial Type' dropdown menu. Then, click the button +, and in the 'Conditions' section, choose 'Hydrological_Variable_top' as shown in Figure 1.42. Enter '0' for 'Ponding Depth' and '-90' for 'Minimum Pressure Head'. After that, click the 'Edit' button to input the 'Profile'.

💀 Initial/Boundary/SourceSink				-	×
Save Close					
Spatial Type Hydrodlogical/Cau	chy Boundary	+			
Module Spatial Type 1 Hydrodlogical Initial Conditio 2 Hydrodlogical Variable Bound 3 Hydrodlogical Dirichlet Bound	dary 🔳				
Conditions	Profile Pondi Depth	ng Minimum [cm] Pressure Head[cm]	Profile ID		
Hydrological_Variable_top		-90			
• 2	<u>,</u>				

Figure 1.42. Initial/Boundary/SourceSink window.

16. By clicking the 'Edit' button, the 'ProfileInputForm' window will pop up. To input the 'Profile', click the button +. Because there's a rainfall of 5 cm/day for the first ten days and a potential evaporation of 5 cm/day for the second ten days in Benchmark problem 1, enter the values for 'Time' and 'Rainfall/Evaporation rate' as shown in Figure 1.43. Then, click the 'Save' button to close the 'ProfileInputForm' window, and click the 'Save' and 'Close' buttons to exit the 'Initial/Boundary/SourceSink' window.

🖳 F	ProfileInputForm					-	- 0
5	ave Close						
Dro	file + Selected Profile: 1						
FIC	file + Selected Profile: 1						
	Profile Name						
▶ 1	1						
			Draw				
Dat	a Points		Draw				
	Time[day]	Rainfall/Evaporation Rate[cm/day]					
	0	5					
	2 10	5					
	3 10.001	-5					
	1 20	-5					
• :	5						

Figure 1.43. ProfileInputForm window.

17. Through steps 10 to 16, the input for the initial and boundary conditions has been completed. Under the 'Simulation' tab, you can view the input file generated based on the data entered so far as an ASCII file named 'Sim.inp' in the 'Model Input File' section, as shown in Figure 1.44.

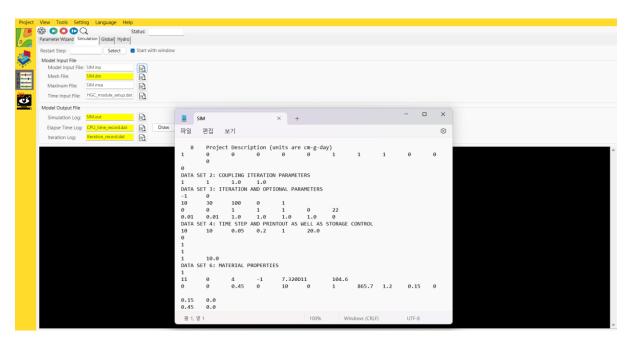


Figure 1.44. 'Sim.inp' file under Simulation tab.

18. In the 'Global' tab, as seen in Figure 1.45, you can view and modify the inputs for title, coupling iteration parameters, time step, material property, and unit.

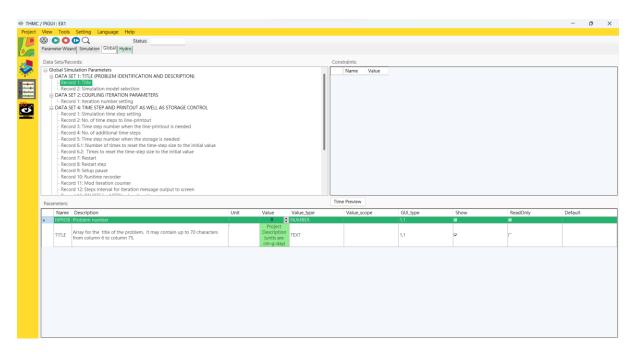
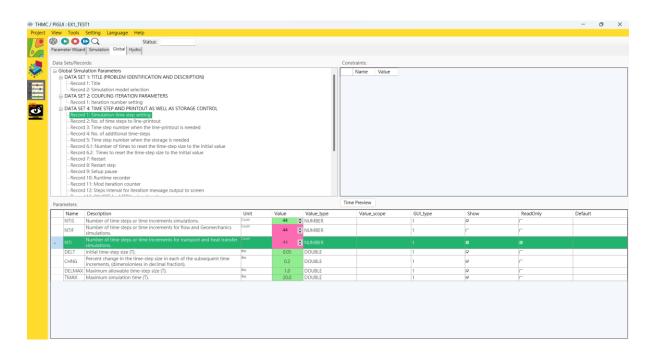


Figure 1.45. 'Global' tab.

In the 'Global' tab, confirm that 'Data set 4' \rightarrow 'Record 1' has 'NTIF = NTIS = 44'.



19. In the 'Hydro' tab, you can view and modify inputs related to iteration and optional parameters, material properties, initial conditions, sink/source, and boundary conditions. Since Benchmark problem 1 simulates the vertical flow of fluid flow, it should be set as 'Data set 3 \rightarrow Flow Record 4 \rightarrow KGRAV = 1' as shown in Figure 1.46. For matrix solution, choose the direct band-matrix solver (IPNTSF = 0). Furthermore, since the tolerance is set to 0.02, it should be configured as 'Data Set 3 \rightarrow Flow Record 6 \rightarrow TOLAF = 0.02, TOLBF = 0.02'.

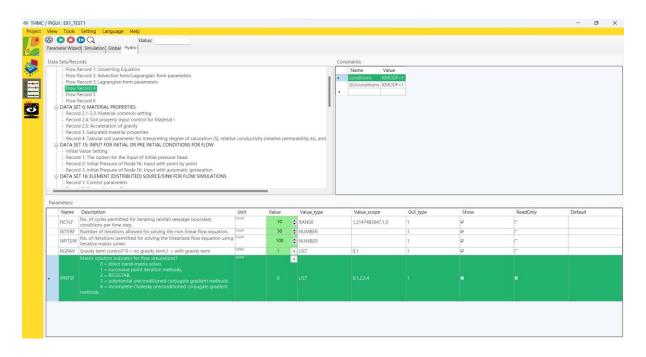


Figure 1.46. 'Hydro' tab.

20. The input for the 'Model Parameter Input & Simulation GUI' has been completed. Then, click 'Start Model Simulation' 🖸 located on the upper left side of the program.



Now, the numerical simulation is to be run through the 'Visualization GUI' the upper left side of the program, and the results are to be examined.

located on

1.3 Visualization GUI

The Visualization GUI has the following five tools: Save B, Add Task •, Remove Task •, Start •, and Pause • as shown in Figure 1.47.

- 1. First, to display the numerical simulation results, click the 'Add Task' tool •
- 2. After selecting the appropriate Task, click the 'Start' tool **•**.
- 3. Click the 'Open' button Open next to 'ParaView' in Figure 1.47.

	Setting Lar	nguage	Help								
	00										
ualization Ta	asks:										
- Task N	Name	Status	Update E	Date Open .dat with	Open .vtu with						
					Open ParaView V Open						
sk Detail Vari	iable ParaView	Settings	nterpolatio	n Tool							
Task Name:		EX1 TES	T1 - Task 1								
	Dethu			D8\THMC\Project\EX1_TEST1\Solver	ANCCAROUTENT		Select	Reset			
Source Folde											
Visualization	n Path:	C:\THM	C_2023080	D8\THMC\Project\EX1_TEST1\Solver	r\HGC4.3\OUTPUT		Select				
Visualization	n Format	Ext	ension	Application							
		🛃 .vtu		ParaView							
		.dat		Tecplot							
Target Time	Step:	45 / 45	steps.								
	Step: Time	45 / 45 : Nodes	steps. Cells	File Name		 					
p ID	Time			File Name 000000000.0000000E+00.dat		 			 	 	
p ID	Time 1 0	Nodes	Cells 0			 			 		
ID 1 ID 1 Initia	Time 1 0 0.05	Nodes 0	Cells 0 0	0000000i0.000000E+00.dat		 			 	 	
2 2 1	Time 0 0.05 0.11	Nodes 0	Cells 0 0 0	00000000i0.000000E+00.dat 00000001_5.0000000E-02.dat		 			 	 	
ID 1 Initia 2 1 3 2 4 3 5 4	Time 0.05 0.11 0.182 0.2684	Nodes 0 0 0	Cells 0 0 0 0	0000000000.0000000E+00.dat 00000001_5.0000000E-02.dat 00000002_1.1000000E-01.dat		 			 		
2 0 10 3 0 2 4 0 3 5 0 4 6 0 5	Time 0.05 0.11 0.182 0.2684 0.37208	Nodes 0 0 0 0	Cells 0 0 0 0 0 0 0 0	00000000000000000000000000000000000000	I	 			 	 	
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.496496 0.496496	Nodes 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	000000000.0000000000000000000000000000					 	 	
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6 8 7	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.496496 0.6457952 0.6457952 0.6457952	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	0000000015.0000000E-eO0.dat 00000001_5.0000000E-02.dat 00000002_1.1000000E-01.dat 00000003_1.820000E-01.dat 00000005_3.7208000E-01.dat 00000005_4.949600E-01.dat 00000007_6.4579520E-01.dat					 	 	
ID 1 Initia 2 I 3 2 4 3 5 4 6 5 7 6 8 7 9 8	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.6457952 0.82495424 0.82495424	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	000000000 0000000 - 00.dat 00000001_5.000000E - 02.dat 00000002_1100000E - 01.dat 00000005_3.720000E - 01.dat 00000005_3.720000E - 01.dat 00000006_4.964900E - 01.dat 00000006_4.9549502E - 01.dat 00000000_8.2495424E - 01.dat		 			 		
ID 1 Initia 2 1 3 2 4 3 5 4 6 5 7 6 8 7 9 8 10 9	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.6457952 0.82495424 1.0399451	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	00000000 0000000 -00 dat 0000000 5.0000000 - 02.dat 00000002 1.0000000 -01.dat 00000004 2.68400000 - 01.dat 00000006 4.96496000 - 01.dat 00000005 6.4579520 - 01.dat 00000005 6.4579520 - 01.dat 00000005 6.4579520 - 01.dat		 			 		
ID 2 Initial 2 I 3 2 4 3 5 4 6 5 7 6 8 7 9 8 10 9 11 0	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.6457952 0.6847952 0.82495424 1.0399451 1.2979341	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	0000000015.0000000E+00.dst 000000015.000000E+0.dst 000000021.100000E+0.1.dst 000000023.1820000E+01.dst 000000054.9649600E+01.dst 000000054.979502E+01.dst 000000005.197952E+01.dst 000000005.1297952E+01.dst						 	
2 0 1 2 0 1 3 0 2 4 0 3 5 0 4 6 0 5 7 0 6 8 0 7 9 0 8 10 0 9 11 0 10 12 0 11	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.6457952 0.82495424 1.0399451 1.2979341 1.6075209	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	000000001_5000000E+00.dst 00000001_5000000E+02.dst 00000002_1100000E+01.dst 00000002_1820000E+01.dst 00000005_42.684000E+01.dst 00000005_49249600E+01.dst 00000005_4929524E+01.dst 00000000_11099451E+00.dst 00000000_12979341E+00.dst		 					
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6 8 7 9 8 10 9 11 10 12 11 13 12	Time 0 0.05 0.11 0.182 0.2684 0.37208 0.496496 0.6457952 0.82495424 1.0399451 1.6075209 1.6975209 1.6075209 1.9790251	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	00000001000000000000000000000000000000							
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6 8 7 9 8 10 9 11 10 12 113 13 12 14 13	Time 1 0 0.05 0.11 0.182 0.2684 0.37208 0.6457952 0.6457952 0.62495424 1.0399451 1.2979341 1.675209 1.9792521 2.4248301 2.4248301	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	00000001 0000001 410 0181 00000001 5.0000000-01.484 00000002 1.1000000-01.484 00000002 1.200000-01.484 000000000 1.240400-01-01.484 00000000 1.494980-01-01.484 00000000 1.24979341-01.048 00000001 1.2979341-01.048 00000001 1.2979341-01.0484 00000001 1.2979341-01.0484 00000000 1.2979341-00.0484 00000000 1.2979341-00.0484 0000000000000000000000000000000000							
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6 8 9 10 9 11 10 12 11 13 12 14 13 15 14	Time 1 0.05 0.01 0.11 0.182 0.2684 0.37208 0.6457952 0.82495424 1.0399451 1.2979341 1.8075209 1.59700251 2.4248301 2.9597962 0.24954724	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells O 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	0000000100000014:e10-001 00000001_50000004:e02.68 00000002_10000008:e10.48 00000002_14000008:e10.48 00000002_1400008:e10.48 00000002_140793208:e10.48 00000000_140793208:e10.48 00000001_150793208:e10.48 0000001_12079341:e00.48 0000001_12079341:e00.48 0000001_2219790251:e00.48 0000001_22424801:e00.48							
ID 1 Initial 2 1 3 2 4 3 5 4 6 5 7 6 8 7 9 8 10 9 11 10 12 113 13 12 14 13	Time 1 0 0.05 0.11 0.182 0.2684 0.37208 0.6457952 0.6457952 0.62495424 1.0399451 1.2979341 1.675209 1.9792521 2.4248301 2.4248301	Nodes 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Cells O 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	00000001 0000001 410 0181 00000001 5.0000000-01.484 00000002 1.1000000-01.484 00000002 1.200000-01.484 000000000 1.240400-01-01.484 00000000 1.494980-01-01.484 00000000 1.24979341-01.048 00000001 1.2979341-01.048 00000001 1.2979341-01.0484 00000001 1.2979341-01.0484 00000000 1.2979341-00.0484 00000000 1.2979341-00.0484 0000000000000000000000000000000000							

Figure 1.47. Visualization GUI window.

- 4. Then, the 'ParaView' window will pop up. Choose 'Pressure' in the list box.
- 5. In Figure 1.49, click the play button 🕨 to display the pressure head distribution.

III ParaView 5.10.1																	-	\Box ×
<u>File Edit View Sources Filters Extract</u>	tors Ic	ools <u>C</u> at	alyst <u>M</u> a	cros <u>H</u> elp														
📫 🛍 🕹 🔀 🞻 🕸 🕸) 🖌		e .	: 💓 🛛 🗸				Time: 0			0 🛊 🛛	্ ্						
📕 🚂 🚔 🛱 🛱 🛱 🔹	Den	sityFlow		* Surfa	ice Wit	Edges	Ŧ	X X 🛞 I	i (0,	** 1	s 🏥 💥	1 +Z 1 1-2	C.	<u></u> ?!	• 🕑	Q		
	 Moi: Porc 	stureCont osity		. 🖬 🚹		0	₩ {	} 🤌										
Pipeline Browser	 Pres 	ure																
	Satu			121 W1 191 191	un 121-	(17) (10)	1400 - 241		2								BenderView1	
	Tota					10 10	28 18	< A A ⊉ .	·		200-		-200				Kenderviewi	
	 Velo 			1: 0 (day)							200-		-200					
1		inalCellI									180-		-180					
	Time	eValue																
											160-		-160					
											140-		-140					
Properties Information																		
Properties 28											120-		-120					
e [#] Apply Ø Reset ¥ Delete ?																		
										1	(dm)100-		-1002	(dim)				
Search (use Esc to clear text)											80-		-80					
- Properties (EX1_TE													~~~					
											60-		-60					
✓ Cell/Point Array Status ✓ f OriginalCellID																		
V Criginal CelliD											40-		-40					
MoistureContent		ť														_		
V Porosity		× *									20-		-20			Densit		
V Presure											1.0	e+00 1.000	1.00	0004 1.00	006 1.0000	8 1.0001 1.000	012 1.00014 1.00016 1.00018 1.0002 1.000	2 1.0e+00
V Saturation											0							
V TotalHead																	SpreadSheetView1	
Velocity	Show		TECT1 -	Took 1 - Attribut	ter Dol	at Data	- 0.	ecision: 6 🗘 🛔	0 0 =	1 T R 🚙								
Time Array TimeValue		oint ID D	ensityFlow	MoistureConten		Point		Points_Magnitud		y Presur				Velocity		locity_Magnitude		-
– Display (Unstructur 🗊 🗊	0 0	1		0	0	0	0	0	0	0	0	0	0	0	0 0			
Representation Surface With Edges	1 1	1		-28.2962	0	0	5	5	0	-97	0	0	0	0	0 0			
Coloring	2 2	1		-53.7189	0	0	10	10	0	-97	0	0	0	0	0 0			
DensityFlow	3 3	1		-73.6351	0	0	15	15	0	-97	0	0	0	0	0 0			
🔒 Edit 🚳 🟥 🟥 📬 👻 🗸	4 4	1		-87.8186	0	0		20	0	-97	0	0	0	0	0 0			
																		Ψ.

Figure 1.48. ParaView window.

III ParaView 5.10.1																					-	
<u>File Edit View Sources Filters Extrac</u>	tors I	ools <u>C</u> ata	alyst <u>M</u> acros <u>H</u>	elp																		
📫 🖆 🚣 🔀 🐗 🕸 🕸	1	50	e 🗼 🖗					📫 Time: 20,29	35		44 🗘 🖪	্ ্										
📘 🌬 📸 🛱 🛤 🖬 🖬	Presu	ire	•	Surfa	ce With	lay Euges	•	🗙 💥 🌚 🖠	i (q. 1	×1 1-×	14 4	*Z	C	్లి 🗜	: 📀	ତ G						
1 🔊 🖗 🕸 🔊 💮	ê 2	2 🛞 1	🧿 I 🛯 🗖	1 🔛		<u>©</u> æ (€ {	}														
Pipeline Browser 🛛 🖗		.ayout #16	8 +																			
builtin:	(A) (A)	2	10 🔍 🔿 🗕 🚉		e (1):	彩彩	1	X A A 2 2	+ - 1	t.										Rende	erView1	
EX1_TEST1 - Task 1 AnnotateTime1	Ð	<1_TEST	1 - Task 1: 2	20 (day)							200-		-200									
											160 140		-160									
Properties Information																						
Properties @ 8	8										120-		-120									
e [#] Apply ⊘ Reset ¥ Delete ?										z	(dm)100		-1002	(dim)								
Search (use Esc to clear text)											80-		-80									
- Properties (EX1_TE											80-		-80									
Cell/Point Array Status Cell/Point Array Sta		۲									60- 40-		-60 -40									
Moistrecontent Porosity Presure Saturation											20- -9 .7 0-	e+01 -90	-20	-80	-70	-60	Presure -50	-40	-30	-20	-10	0.0e+0
✓ TotalHead ✓ Velocity																				SpreadShe	etView1	
v velocity						t Data	* Pr	ecision: 6 🛊 👖														
Time Array TimeValue			ensityFlow Moist			Points		Points_Magnitude						Velocity		Velocity_Magn	tude					
- Display (Unstructur	0 0	1	1.0850	6e+162	0	0	0	0	0.45	0	0.990361	0	0	0	-1.305	1.30595						
Representation Surface With Edges	1 1	1	6.0484	6e-163	0	0	5	5	0.45	-4.33	0.971117	0.662553	0	0	-1.296	1.29616						
Coloring	2 2	1	8.2615	ie+92	0	0	10	10	0.45	-8.64	0.942405	1.35531	0	0	-1.294	1.29497						
Presure	3 3	1	-4.673	86e-48	0	0	15	15	0.45	-12.9	0.913909	2.08048	0	0	-1.292	1.29295						
■ Edit 🚳 🟥 🟥 📬 💗 🕶	4 4	1	-4.422	49e+293	0	0	20	20	0.45	-17.1	0.885646	2.84055	0	0	-1.290	1.29007						
		1.																				

Figure 1.49. Pressure head distribution.