

***MINEDW* version 9.7 tutorial**



ITASCA

***MINEDW* version 9.7 tutorial**



Prepared For:
MINEDW version 9.7 Users

July, 2026
ITASCA-MAN-1000.02

Prepared By:
Tonk M., Xiang J.



Avenida Pedro de Valdivia 291, Second
Floor, Providencia
Santiago de Chile, P.C. 7500524

+ (56) 224 341 300
Info.chile@oneitasca.com
www.oneitasca.cl

PREAMBLE

Disclaimer: The following tutorial is for training purposes only. The three tutorials provided are not designed to simulate real conditions or actual mine-dewatering systems. The problem descriptions are provided solely for the purpose of training a user to use the features of **MINEDW** and should not be assumed to be representative of actual groundwater modeling methodology or real-world problems. Professional judgment and experience are required to set up a well-defined groundwater flow model to produce meaningful results.

The following examples are prepared for new users of **MINEDW**. They provide information on the capabilities and functions of **MINEDW** as well as explain the step-by-step procedure for setting up a **MINEDW** model.

These tutorials do not cover detailed operations of each **MINEDW** feature; users should refer to the User Manual for detailed explanations.

Review	Prepared by	Reviewed by	Date	Comments
0	Jianwei Xiang Matías Tonk	Manuel Gutiérrez	July, 2026	For release

TABLE OF CONTENTS

PREAMBLE.....	i
TABLE OF CONTENTS	ii
1.0 GENERAL PROCEDURES FOR MODEL CONSTRUCTION AND SIMULATION.....	1
2.0 TUTORIAL 1. STEADY-STATE GROUNDWATER FLOW MODEL.....	2
2.1 Problem description.....	2
2.2 MINEDW Model Construction.....	2
2.2.1 Step 1. Generating a Mesh.....	13
2.3 Building the model in MINEDW.....	24
2.3.1 Step 2. Defining Project Properties.....	26
2.3.2 Step 3. Defining Time Steps.....	27
2.3.3 Step 4. Importing Ground-Surface Elevation.....	28
2.3.4 Step 5. Adding Layers	32
2.3.5 Step 6. Defining Hydrogeological Units.....	34
2.3.6 Step 7. Assigning Hydraulic Zones to Elements.....	34
2.3.7 Step 8. Defining "Pinch-Outs"	41
2.3.8 Step 9. Defining Initial Heads.....	46
2.3.9 Step 10. Assigning the Boundary Conditions	46
2.4 Constant-Head Boundary	47
2.5 River Boundary Condition.....	51
2.6 No-Flow Boundary.....	55
2.6.1 Step 12. Assigning Recharge.....	55
2.6.2 Step 13. Adding Evaporation	58
2.6.3 Step 14. Saving the Project.....	60
2.7 Running the Model	60
2.8 Results.....	61
2.9 Water-Table Contour.....	61
2.10 2-D or 3-D Head Contour.....	62
2.11 Geologic Cross Section and Water Table.....	63
3.0 TUTORIAL 2. OPEN PIT, ZONE OF RELAXATION, PUMPING WELLS, AND PIT LAKE.....	65

3.1	Problem Description	65
3.1.1	Step 1. Opening the Project File	65
3.1.2	Step 2. Defining Project Properties	65
3.1.3	Step 3. Defining Model Time Steps	66
3.1.4	Step 4. Defining Initial Heads	67
3.1.5	Step 5. Creating a Mining Plan for an Open Pit	67
3.1.6	Step 6. Creating a Zone of Relaxation for an Open Pit	71
3.1.7	Step 7. Open-Pit Backfilling	74
3.1.8	Step 8. Adding Pumping Wells	75
3.1.9	Step 9. Defining the Boundary Conditions	78
3.2	Running the Model	82
3.3	Results	82
3.4	Creating Monitoring Points	82
3.5	Creating Hydrographs	84
3.6	Pit-Node Elevation and Water Table	84
3.7	Head Distribution in 3-D	86
3.8	Head Distribution Cross Section	87
3.9	Exporting Pore Pressures to a Different Grid in 2-D	88
3.10	Exporting Pore Pressures to a Different Grid in 3-D	90
3.11	Seepage Component to Pit	91
3.12	Pit Lake – Water Level vs. Time	92
3.13	Pumping Wells – Discharge vs. Time	93
3.14	Particle Tracking	93
4.0	TUTORIAL 3. UNDERGROUND MINING	97
4.1	Problem Description	97
4.2	Transient Simulation	97
4.2.1	Step 1. Opening the Project File	97
4.2.2	Step 2. Defining Project Properties	98
4.2.3	Step 3. Defining Initial Heads	98
4.2.4	Step 4. Creating the ZOR file for Caving	99

4.2.5	Step 5. Adding the ZOR for Caving to the Model.....	100
4.2.6	Step 6. Adding the Drain Nodes for the Cave Zone.....	101
4.3	Running the Model	103
4.4	Results	103

1.0 GENERAL PROCEDURES FOR MODEL CONSTRUCTION AND SIMULATION

A steady-state simulation includes the following steps:

- Constructing a mesh using a two-dimensional (2-D) Mesh Generator. (Note: The Mesh Generator is a third-party program, Rhinoceros). The 2-D mesh needs to be saved and then imported into the **MINEDW** graphical user interface (GUI) to create a three-dimensional (3-D) mesh.
- Assigning additional layers using the “pinch-out” method.
- Defining project properties, such as units (meters or feet), type of simulation (i.e., steady state), closure criteria, solver type, and solver parameters.
- Importing the 2-D mesh and adding layers for the 3-D mesh.
- Adding topographic information for land surface and geology.
- Defining hydraulic conductivity zone properties and then assigning the zones to elements.
- Assigning boundary conditions, which may include constant heads, drains, rivers, recharge, or evaporation.

Transient model runs include additional steps such as the following:

- Assigning initial conditions from a steady-state simulation.
- Creating a mine plan for open pit/underground mine(s).
- Defining a zone of relaxation (ZOR) around the open pit or underground excavation, if needed.
- Defining a pit lake if pit-lake formation will be considered.
- Defining pumping wells, pumping rates, and other mining-related structures, such as drain notes.
- Assigning boundary conditions, which may include variable fluxes, constant heads, drains, rivers, evaporation, or recharge.

2.0 TUTORIAL 1. STEADY-STATE GROUNDWATER FLOW MODEL

2.1 Problem description

Tutorial 1 simulates groundwater flow prior to mining under steady-state conditions. The finite-element mesh for the model is constructed so that the model boundary is well beyond the area that will be affected by drawdown. The model domain is approximately 10,000 meters (m) by 10,000 m and has an average thickness of 1,470 m. The surface topography is incorporated in the model with an elevation range of approximately 800–1,858 meters above mean sea level (mamsl). The grid consists of approximately 136,000 nodes and 260,000 elements within 19 main element layers. The number of elements and nodes will vary depending on how the mesh is constructed in the mesh generator and how many layers and pinch-outs are added in **MINEDW**. Finer discretization is used to resolve geology, geologic structures, rivers, and mining in the model domain. The simulated geology includes nine hydrogeologic units (overburden, alluvium, weathered bedrock, sandstone, granite, kimberlite, and two faults). The hydraulic parameters of the model were assigned as shown in Table 1.

Table 1. Hydraulic Parameters Simulated in the Groundwater Flow Model

Properties	Hydraulic conductivity (m/day)			Specific Storage (S_s)	Specific Yield (S_y)
	K_x	K_y	K_z		
Overburden	0.1	0.1	0.1	5e-06	0.005
Alluvium	0.6	0.6	0.6	5e-06	0.005
Weathered Bedrock	0.0009	0.0009	0.0009	5e-06	0.005
Sandstone	0.00125	0.00125	0.00125	5e-06	0.005
Granite	0.0005	0.0005	0.0005	5e-06	0.005
Kimberlite	0.0005	0.0005	0.0005	5e-06	0.005
Fault-1	0.002	0.002	0.002	5e-06	0.005
Fault-2	0.0002	0.0002	0.0002	5e-06	0.005
Contact	0.003	0.003	0.003	5e-06	0.005

2.2 MINEDW Model Construction

Starting in version 9.7 a new method to generate a MINEDW mesh is implemented. Using the Sketch tool from ITASCA software now it is possible to create a mesh valid for MINEDW models. In the following chapters, a step-by-step guide to use it is presented.

2.2.1 Step 1. Generating a Mesh Using MINEDW

Open MINEDW, click "File" on the upper menu, then go to "New" and click on "Sketch Set..." (Figure 1), a pop-up window will appear to define a name for the Sketch Set (Figure 2), write any name and click "OK".

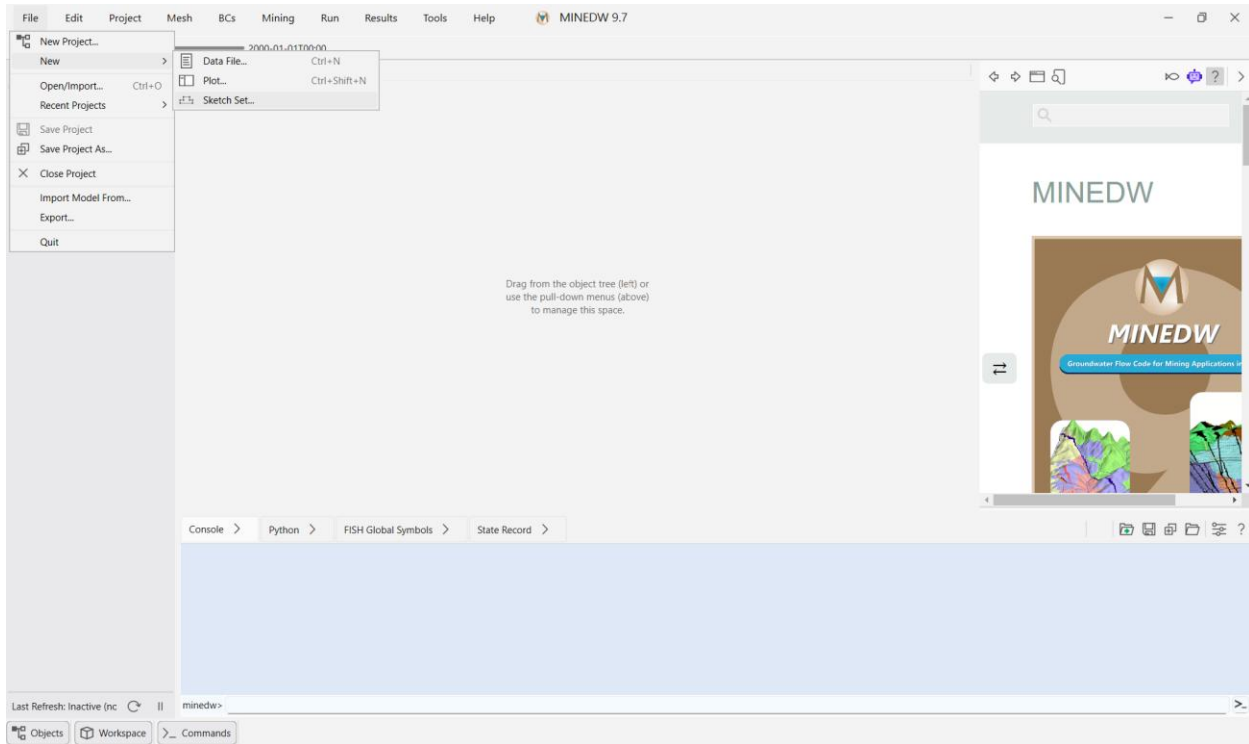


Figure 1. MINEDW Main window

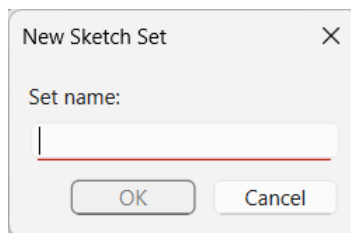


Figure 2. Pop-up window to define name of the Sketch Set.

Then, the Sketch Set user interface will open, allowing the user to draw lines, polylines, points, among other geometries to define the mesh. For this tutorial, the user should import the pre-defined geometry set for the mesh contained in the file "MeshInput-R0.dxf". To import this file, click on "Import background data..." (Figure 3) and the select file window will appear, select the file and click "Open" (Figure 4).

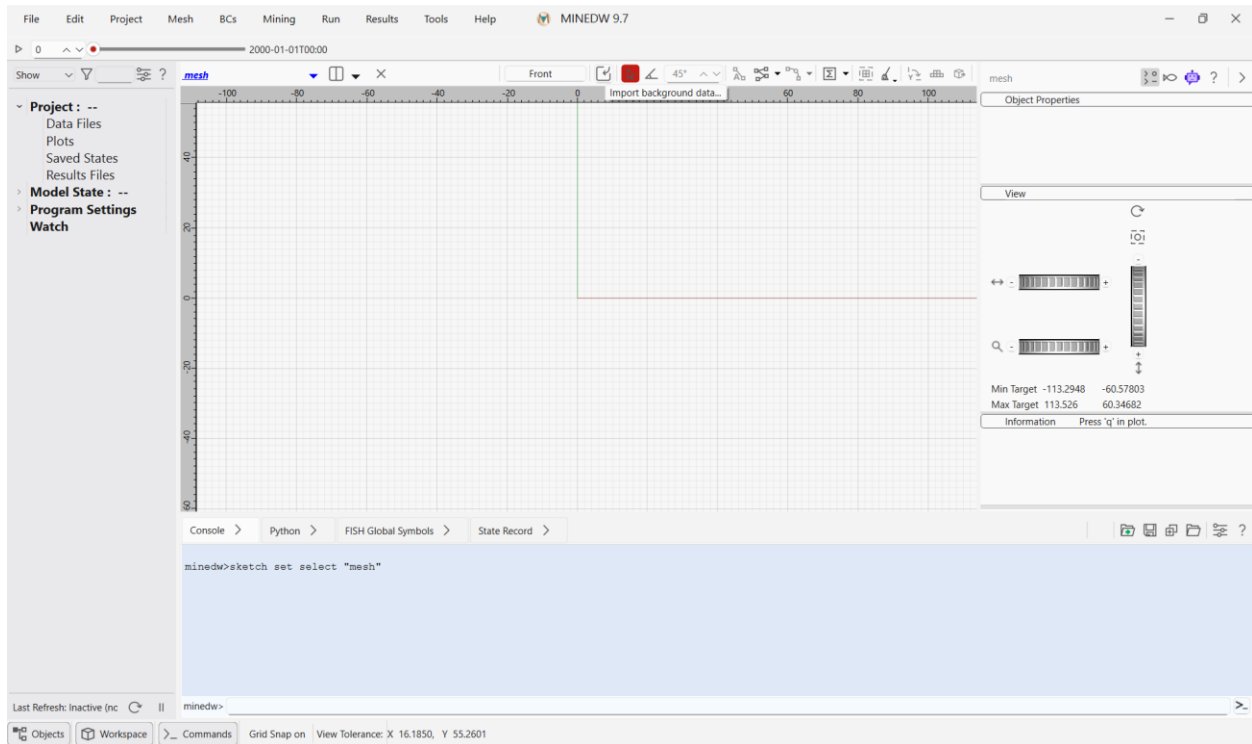


Figure 3. Sketch set interface.

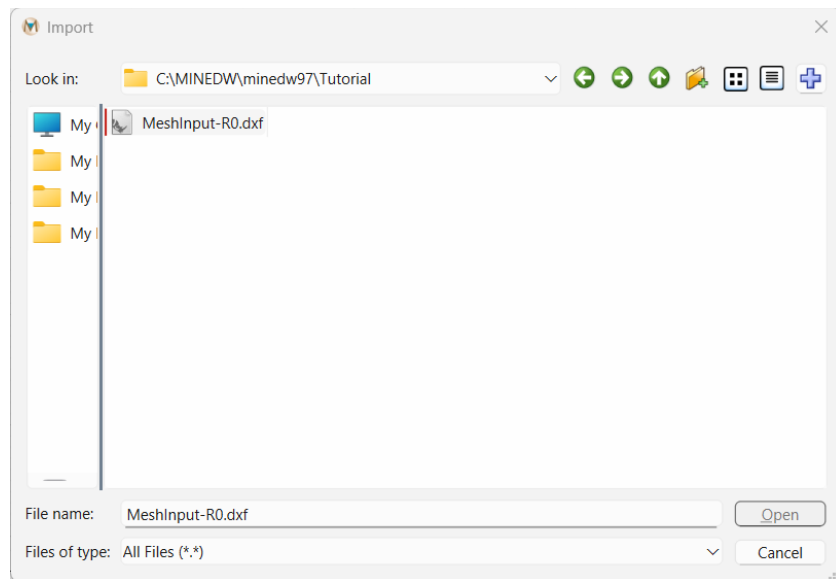


Figure 4. Select file "MeshInput-R0.dxf".

Once open, the geometries contained in the file will be displayed in the GUI (Figure 5). In the upper tools menu click on "Automatically create edges from background geometry" (Figure 5) and the software will detect points and lines that will be used in the mesh development process (Figure 6).

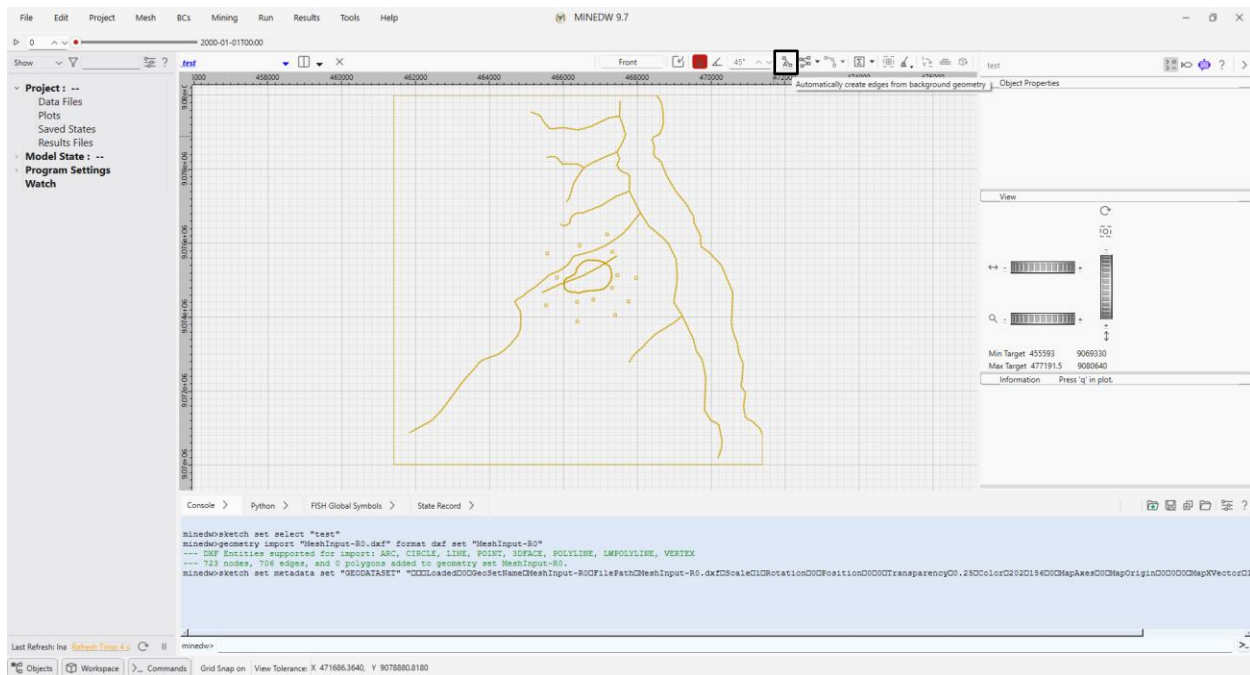


Figure 5. DXF file geometries shown on Sketch Set.

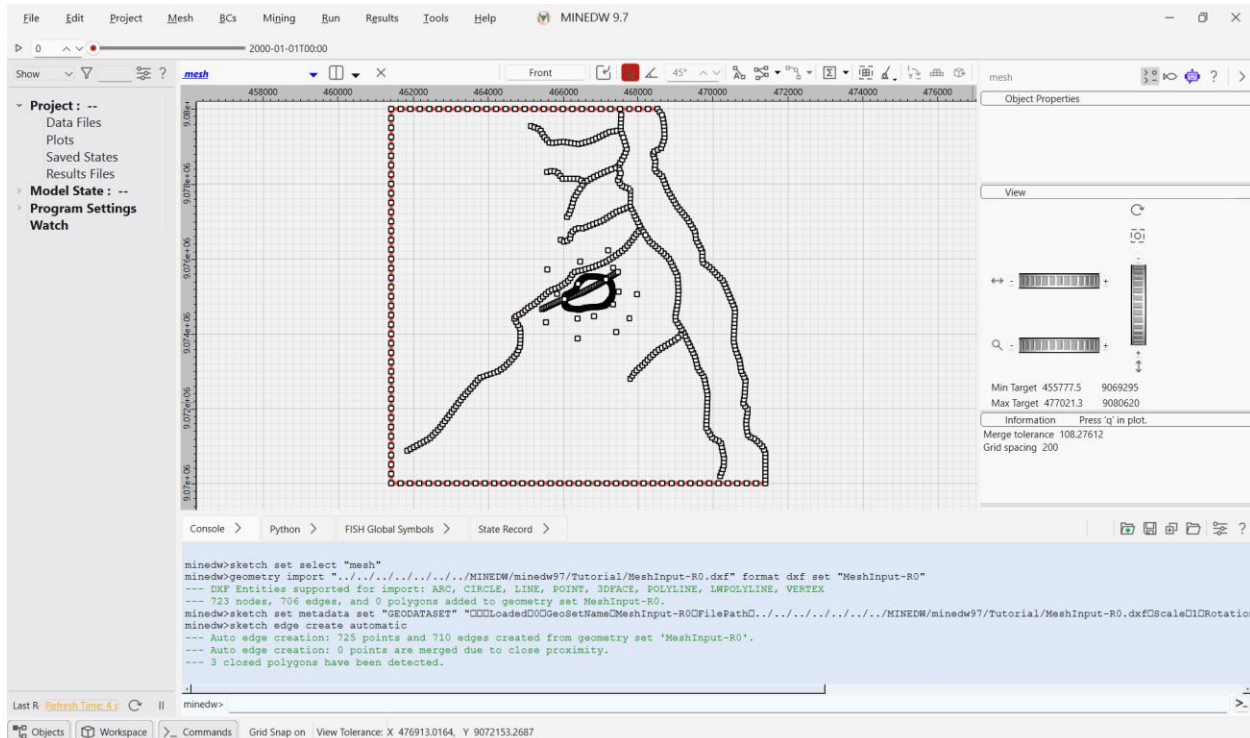


Figure 6. Automatically detected points and lines for the mesh.

Then, drag a square and select all lines and points in the view and change the zone number from default (4) to 1 (Figure 7).

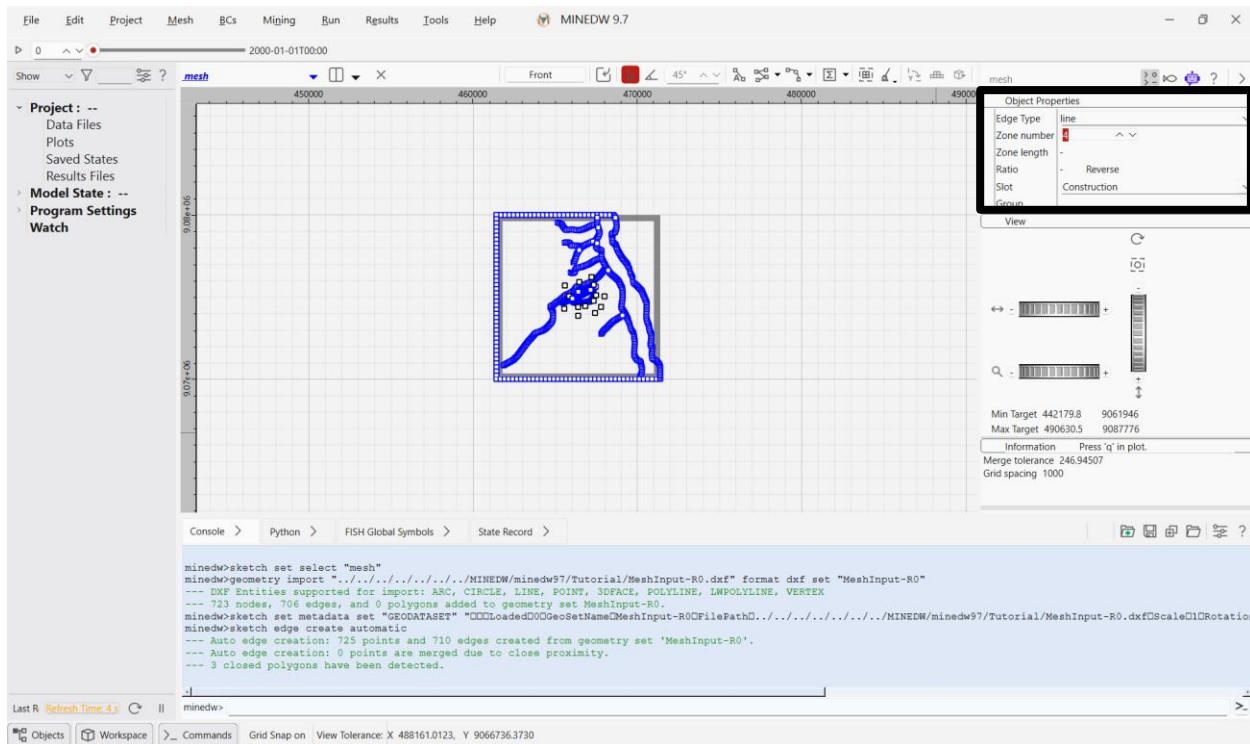


Figure 7. Selection of all the points and lines and change of zone number.

By default, when using the “Automatically create edges from background geometry” tool, each line between two points is then separated into four elements (zone number), but in this case, the length between points is enough to generate a well discretized mesh and it is not necessary to generate more elements.

After this change, zoom in to the pit polygon and select the lines inside it, then delete those lines (Figure 8). If those lines are kept in the Sketch Set, those will be considered in the mesh, but inside the pit polygon our goal is to have an uniform zone length, for this reason we want to delete other geometries that could generate a different zone length.

When this is ready, the mesh could be created, to do this, click on “Mesh all polygons” (Figure 9), or, if some change is needed in the mesh criteria, click on “Set meshing parameters” to modify those parameters (Figure 10), and then it is possible to click “Mesh now” to generate the mesh.

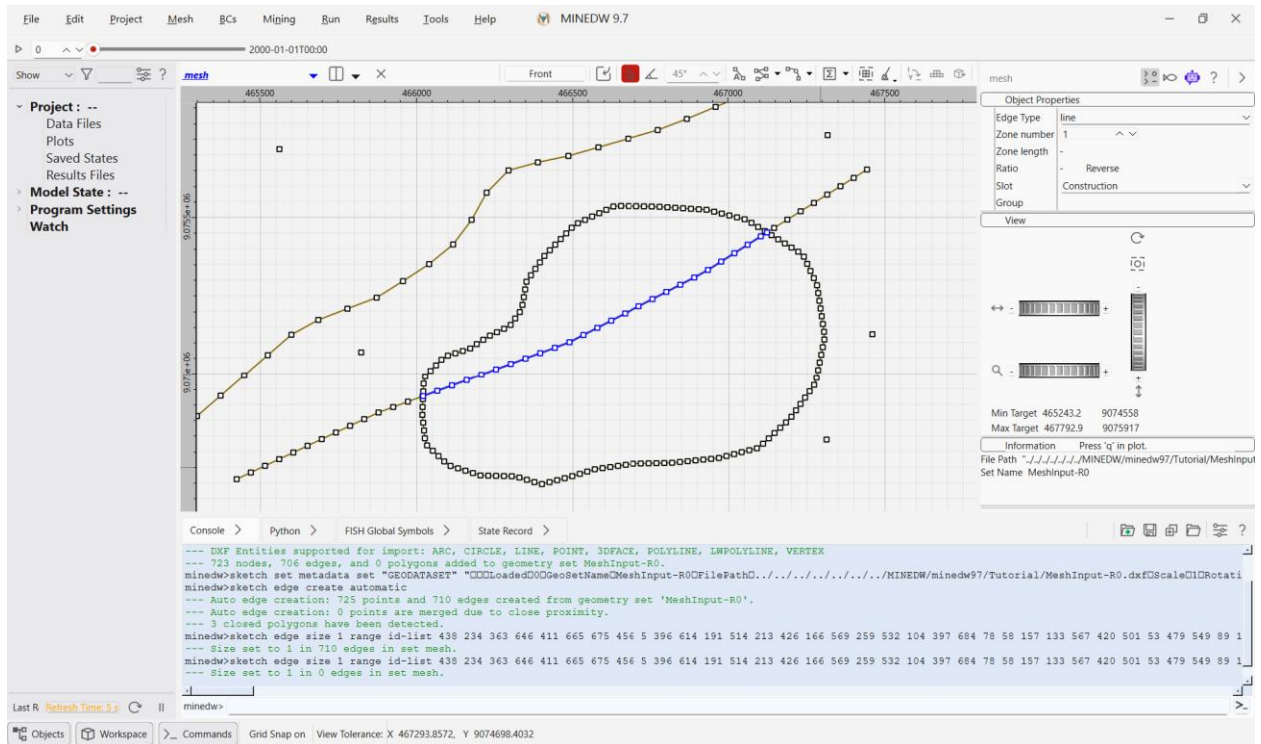


Figure 8. Selection of lines inside the pit polygon.

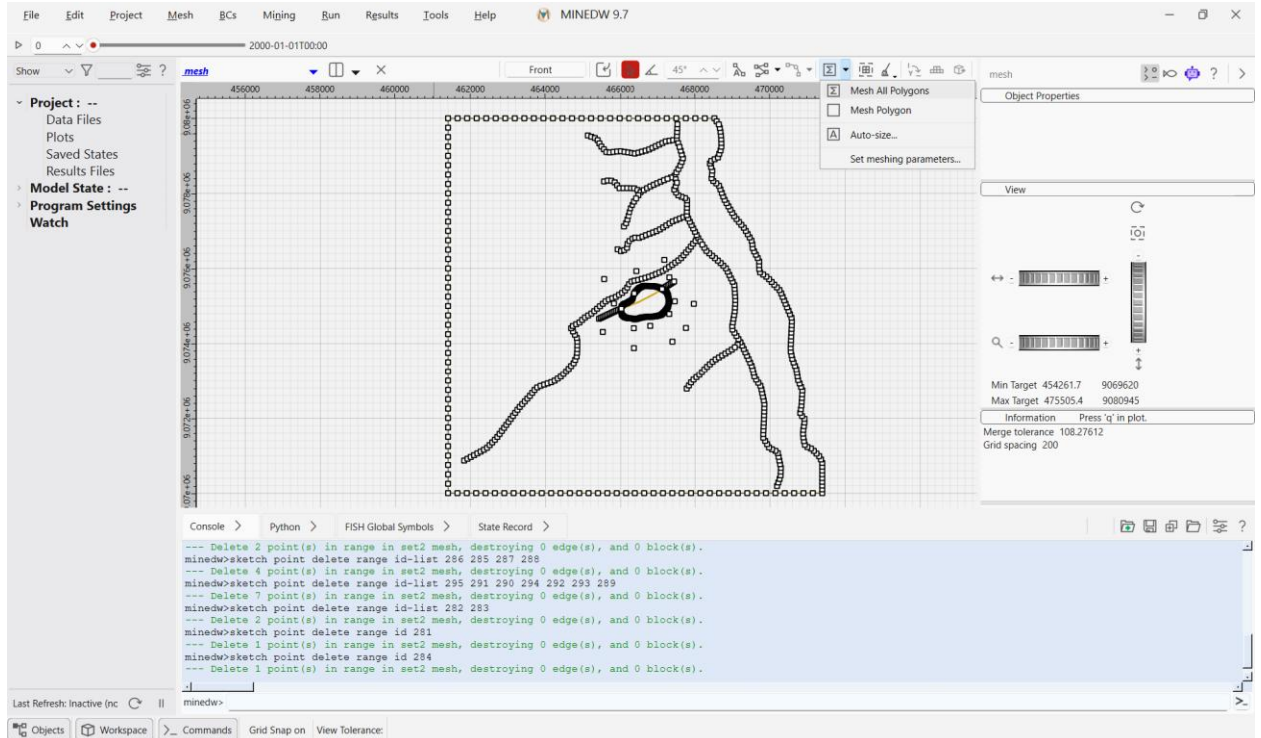


Figure 9. Mesh All Polygons.

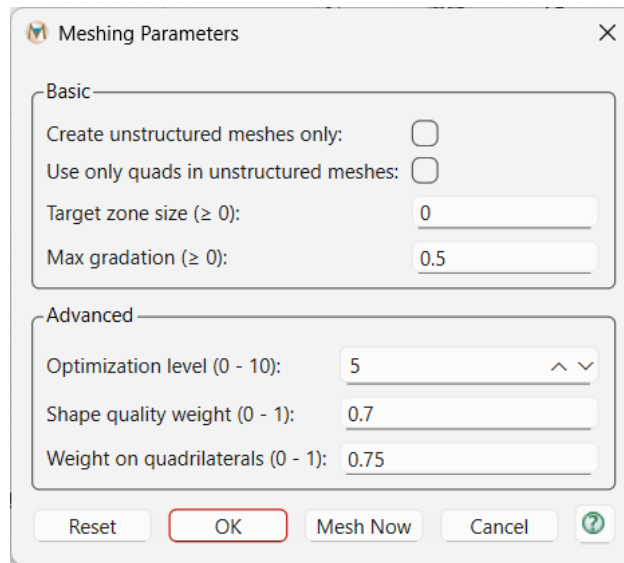


Figure 10. Meshing Parameters window.

The meshing process will end when the mesh is plotted in the Sketch Set (Figure 11). It is possible to zoom in the mesh to check if elements are well defined, if some elements are not achieving a good geometry those will be painted in orange (Figure 12) and the user should modify lines or points and regenerate the mesh to achieve a good aspect.

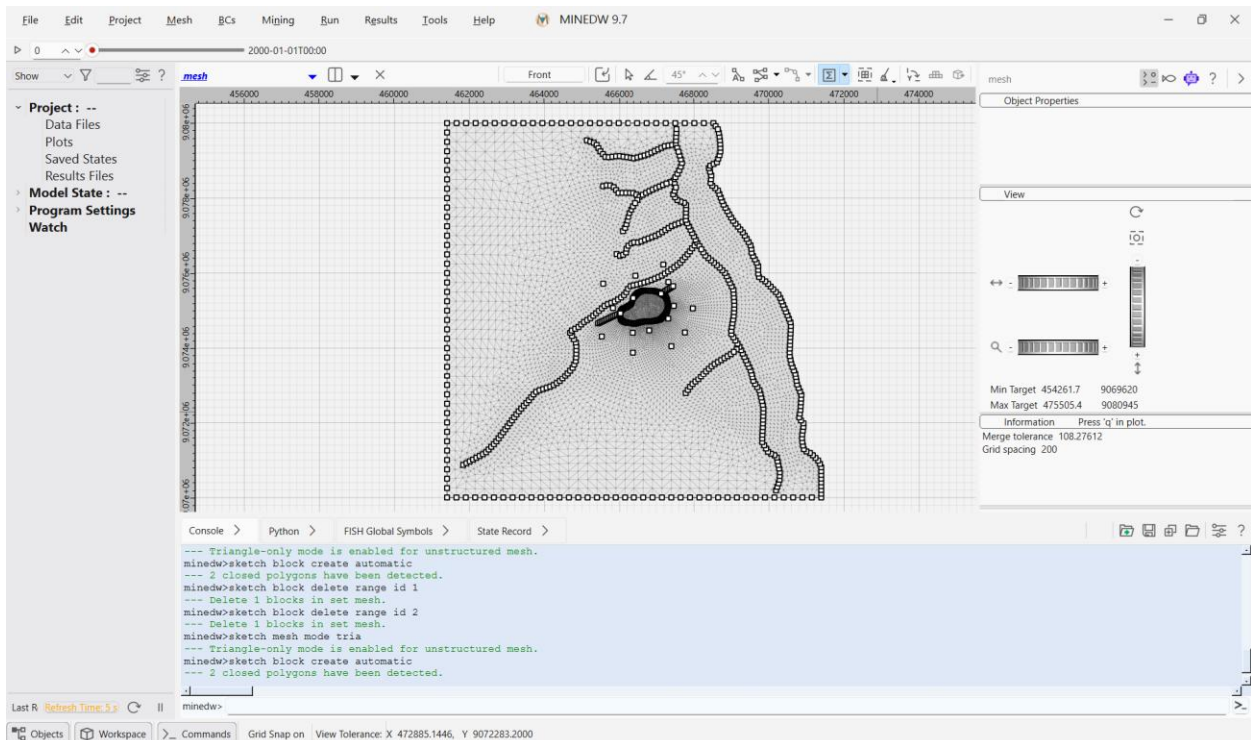


Figure 11. Mesh generated with defined parameters.

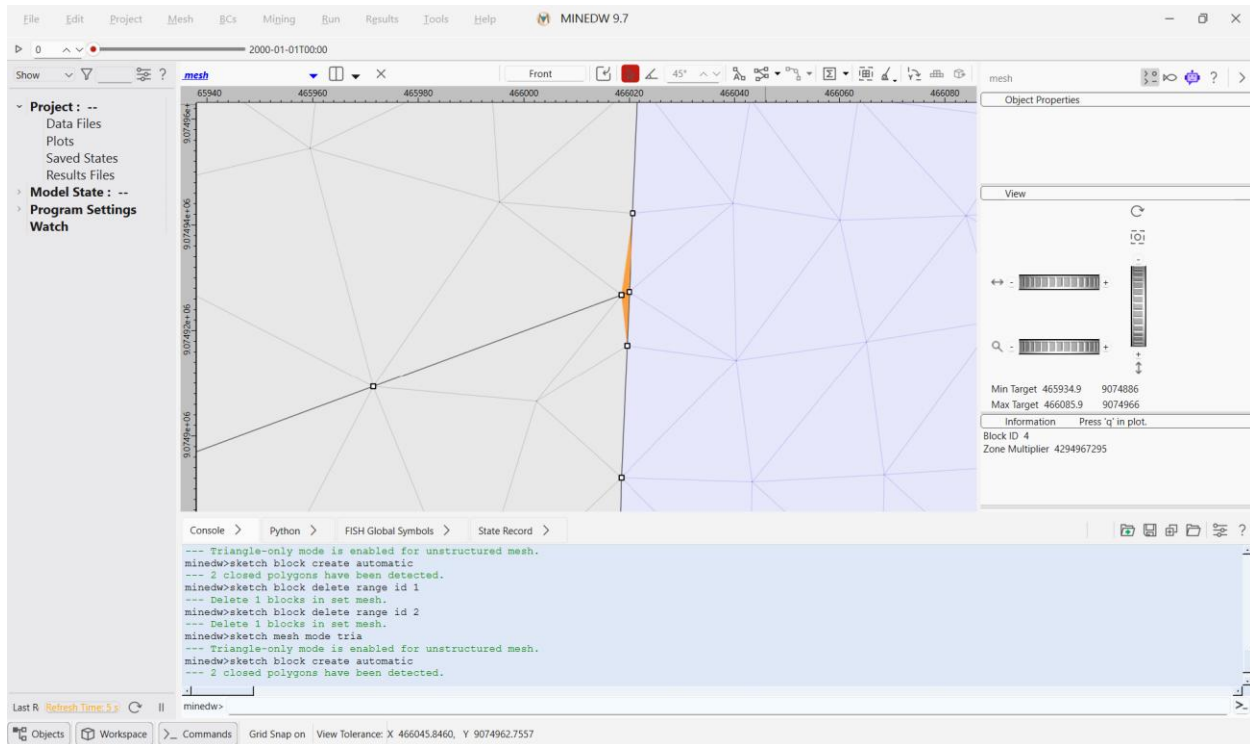


Figure 12. Bad aspect element of the mesh.

In this example, the point in the left of the element was deleted to continue with the following step.

To use user-defined points as nodes in the mesh, those should be defined as “Hard nodes”. To do this, select the points that should be considered as nodes in the mesh, and then, in the top right corner of the user interface, mark the “Hard node” checkbox (Figure 13) and assign the minimum element size that will be used around the node, for this case, 20m.

After committing these changes, the configuration is ready to be used to create the new mesh, Click on “Mesh all polygons” again and wait for the mesh to be plotted. Now the mesh is considering the points as nodes, the defined element size, and the bad aspect element is no longer in the mesh (Figure 14).

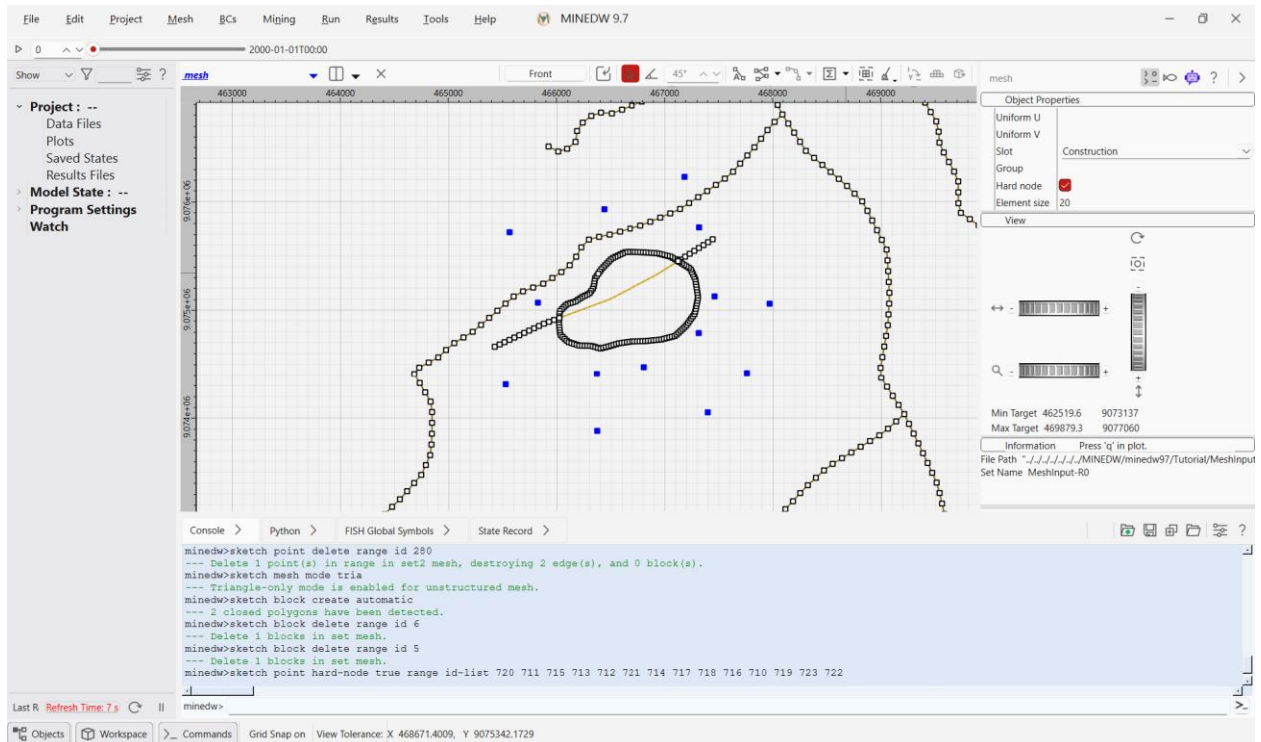


Figure 13. Selection of points to be used as mesh nodes using "Hard node".

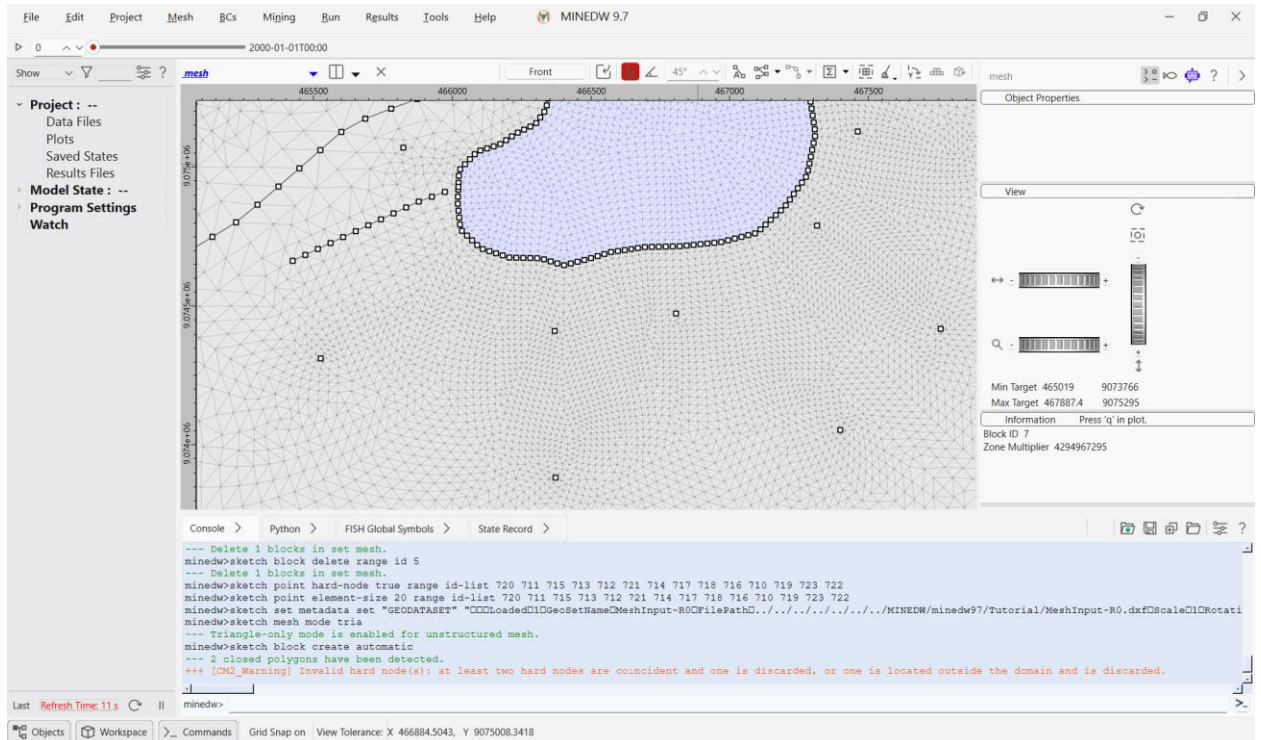


Figure 14. New mesh using updated configuration.

Now it is possible to use the generated mesh as in a MINEDW model. To do this, right click on the user interface and click on "Export as MineDW Model" (Figure 15). If the model was not saved, the software will ask to be saved before moving forward.

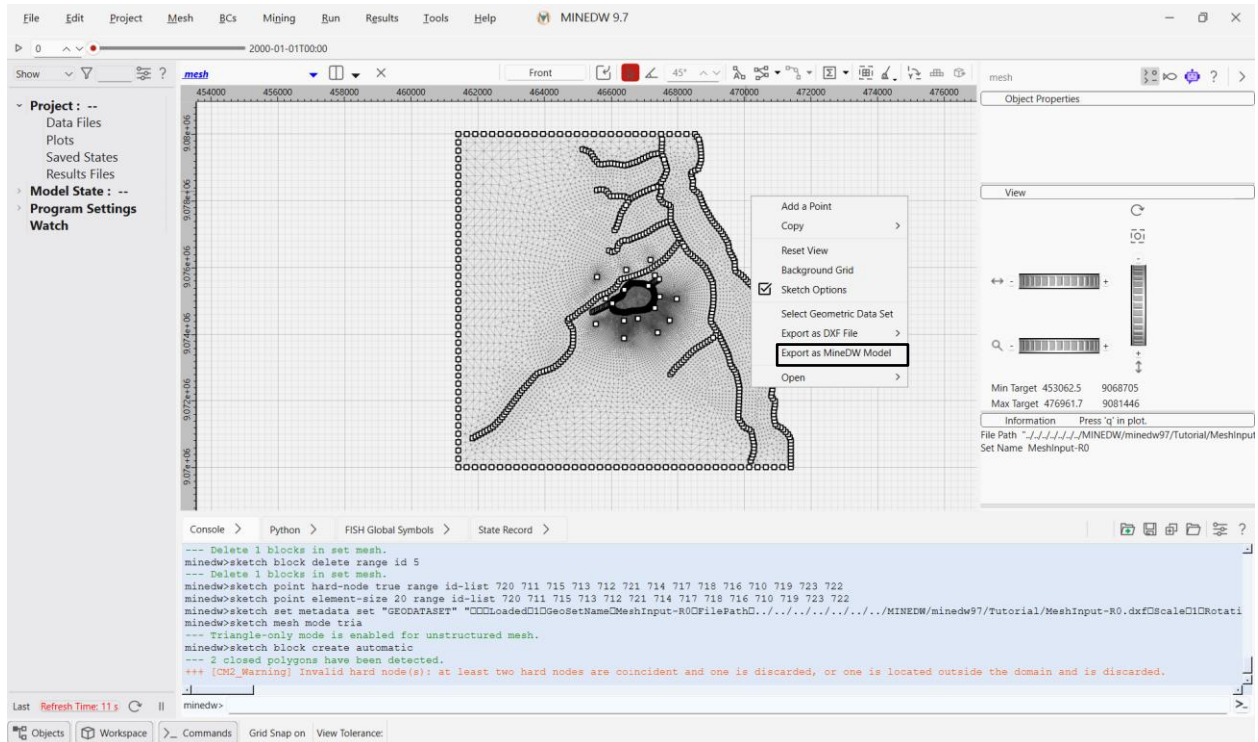


Figure 15. From Sketch Set, export mesh to MINEDW model.

Once the process is complete, the mesh is ready to be plotted as a MINEDW model, click on "File – New – Plot" and assign a name for a new plot (Figure 16). Then, in the plot window right click on the background and select "Add plot item" and add a 2D contour (Figure 17). Click OK and the model will be plotted (Figure 18).

With this process ready, the model is ready to be used in the following steps of building a MINEDW groundwater numerical model.

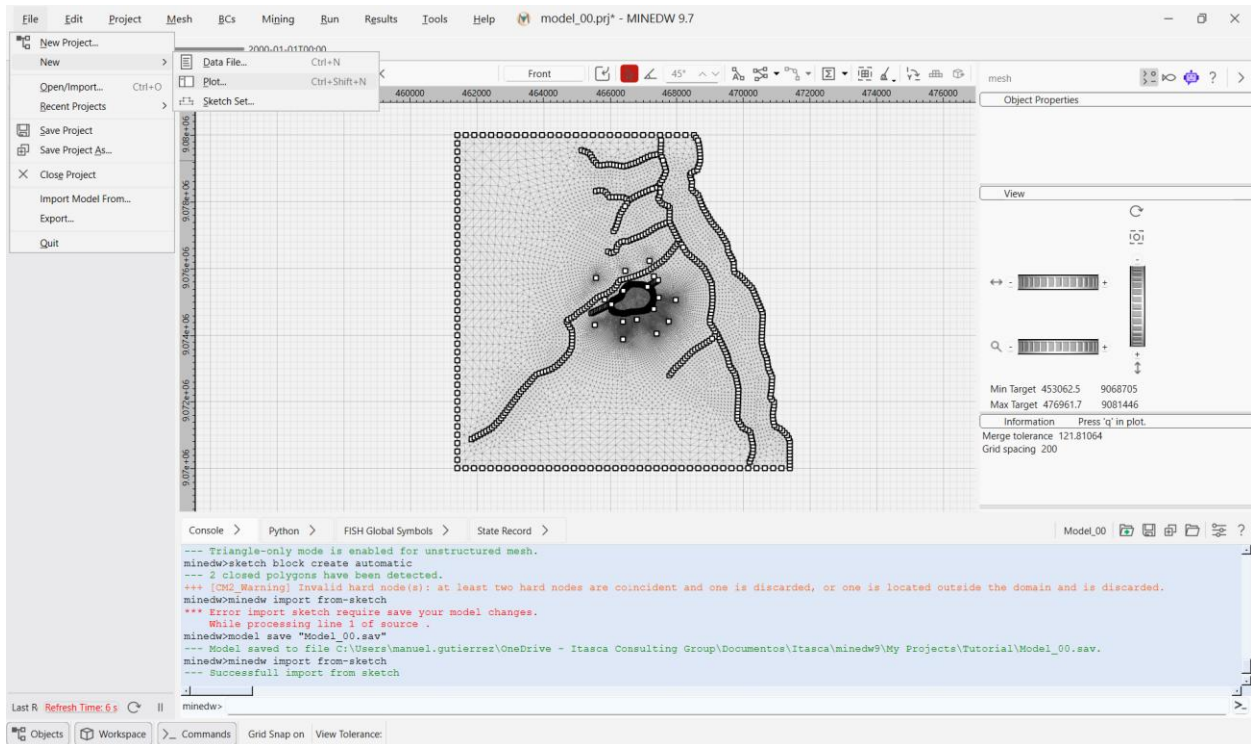


Figure 16. Creating a new plot.

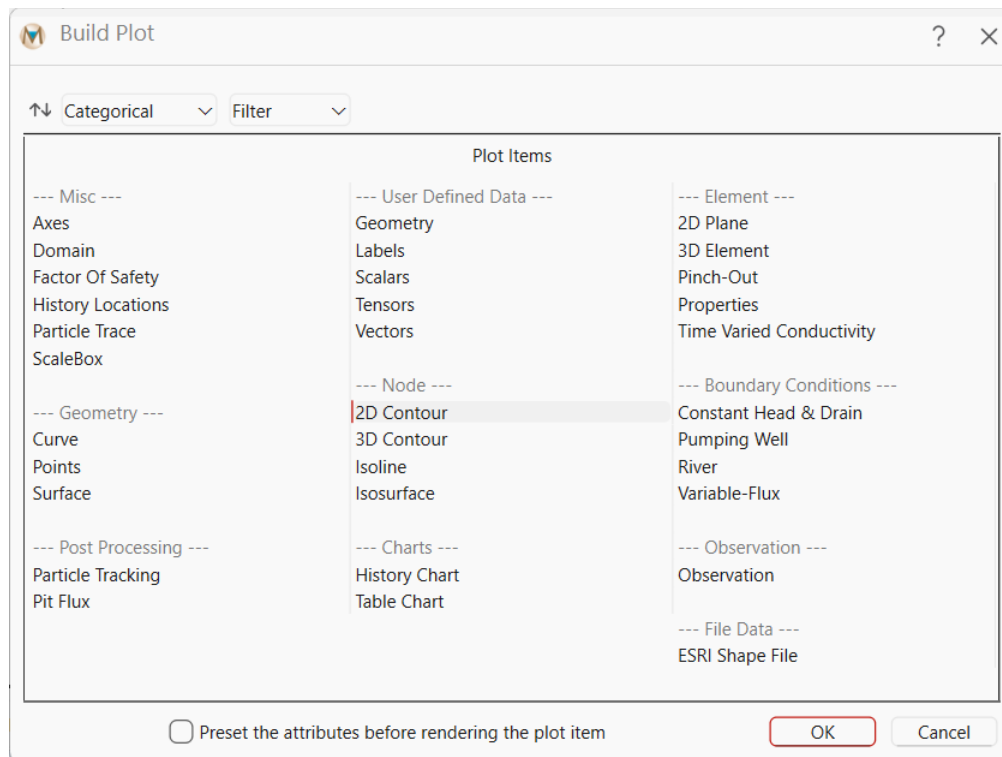


Figure 17. Add 2D Contour to the plot.

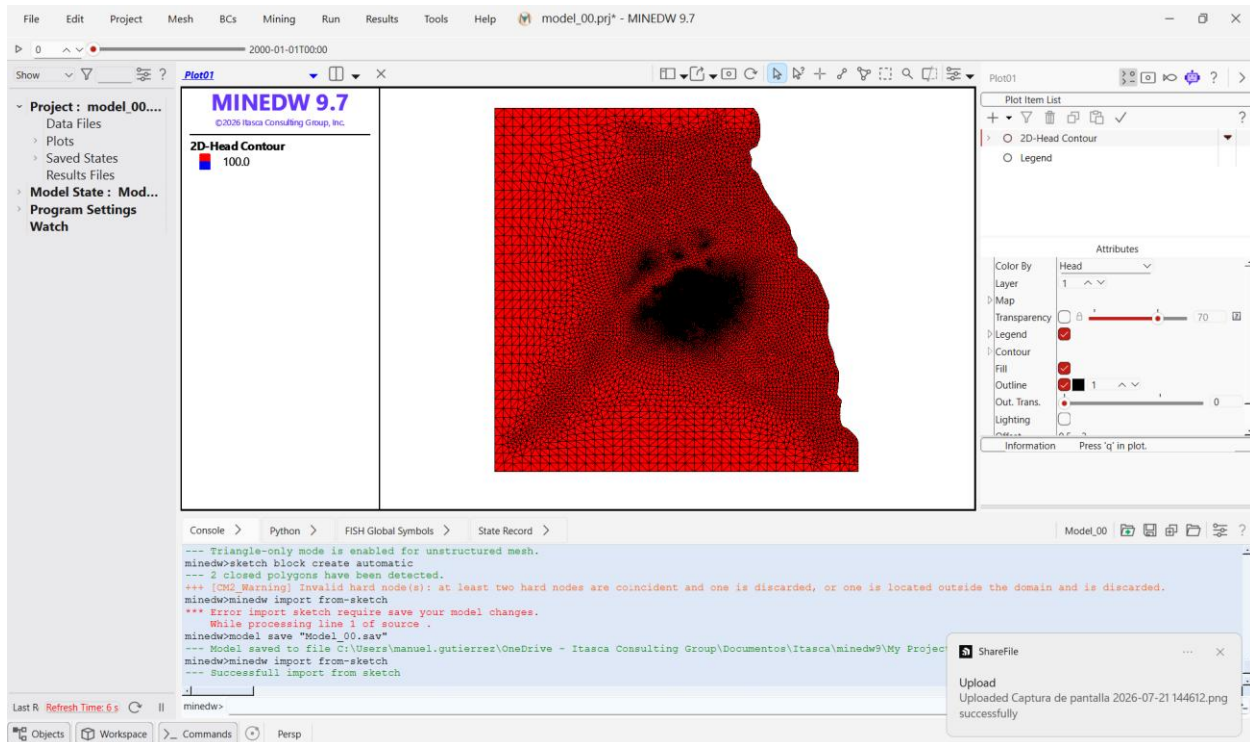


Figure 18. Visualization of the mesh as MINEDW model.

2.2.2 Step 1. Generating a Mesh Using Rhinoceros and Grasshopper

It is important to note that the finite-element mesh is constructed using a third-party program, Rhinoceros (Rhino). In order to follow the steps in this tutorial for constructing a model mesh, Rhino must be installed. Additionally, Grasshopper, an algorithmic modeling tool for Rhino, must be installed if Rhino Version 5.0 is used (Grasshopper comes pre-installed in newer versions of Rhino). Rhino is used to create a 2-D mesh that can be imported into the **MINEDW** GUI, where the remaining model setup occurs.

Open Rhino (if using Rhino 5.0, use the 64-bit option), click on the "File" menu, then select "Import", as shown in Figure 19 Using the "Import" dialog box that opens, navigate to the location where the tutorial files are stored, INPUT>MESH_INPUT>DXF, and select "BOUNDARY.dxf", then click "Open". Repeat these steps for each of the files in the INPUT>MESH_INPUT>DXF directory. If the imported objects do not appear in the view ports, select the "Zoom Extents" on the "Standard" tool tab. All imported objects should now be visible.

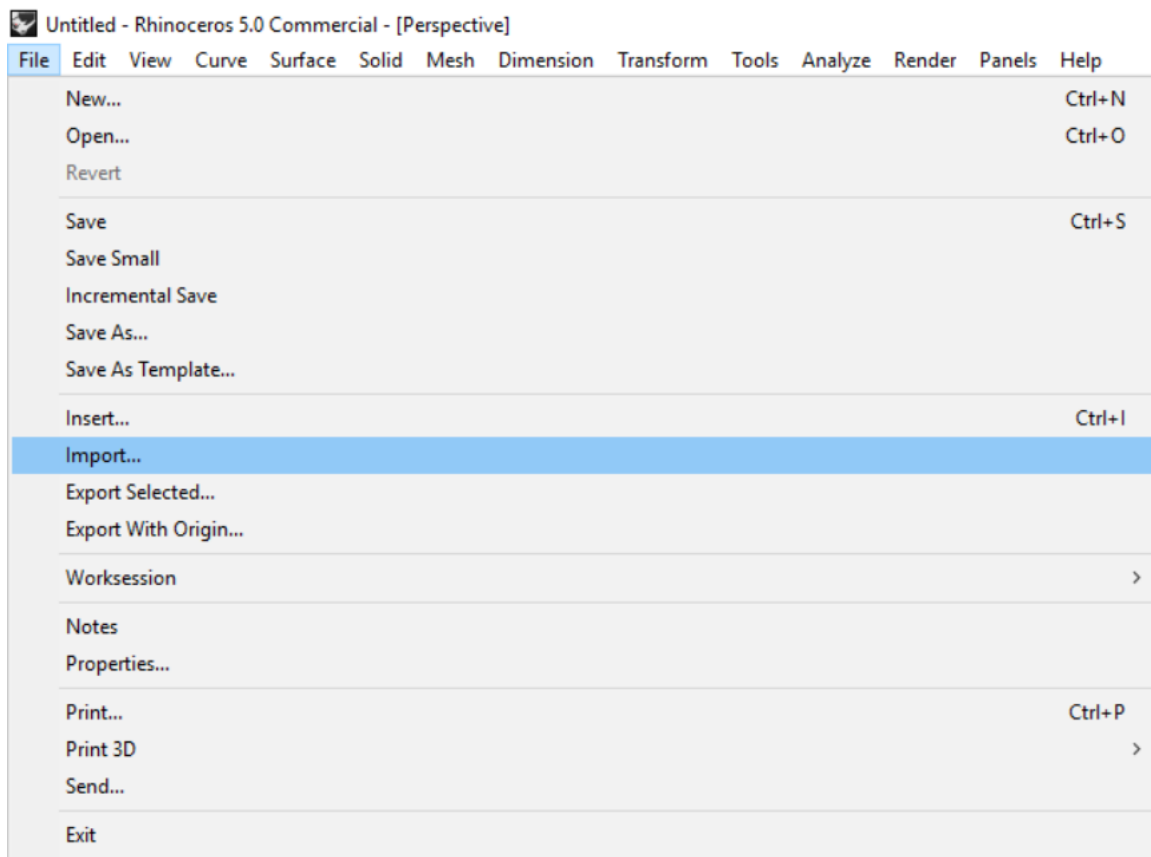


Figure 19. The “File” menu.

Rhino has four default layout view ports to view the imported data: “Perspective”, “Top”, “Front”, and “Right” (Figure 20). Because **MINEDW** requires a 2-D mesh, all work can be conducted using the “Top” layout. The other layouts are useful for checking that all mesh input items lie within the same plane, as the mesh generator in Rhino requires that all mesh items be within the same plane. Verify that all imported items are within the same plane by using the “Perspective” layout to rotate the imported data in 3-D. To do this, right-click and drag within the layout.

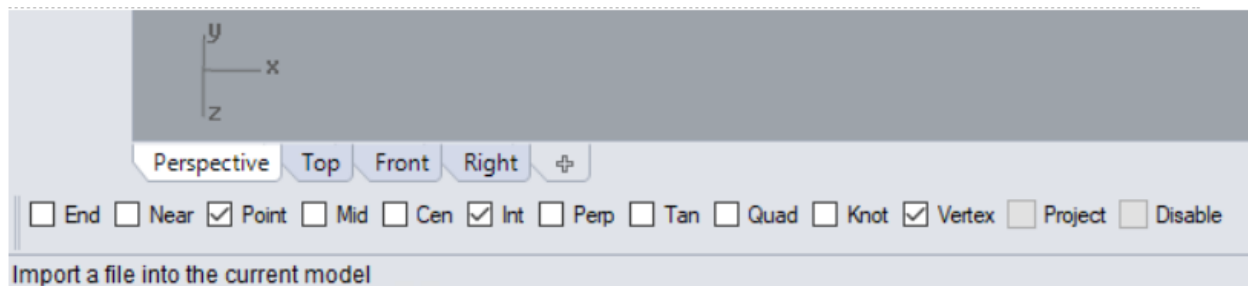


Figure 20. The default view ports in Rhino.

After importing the required files, the result should appear similar to Figure 21. The “MODEL BOUNDARY” must be a completely closed region in order to generate a mesh. The “OPEN PIT” and “TRANSITION ZONE” (Figure 21 and Figure 22) regions will contain a refined mesh and must be closed as well. The “STREAM NETWORK” represents a series of interconnected streams. The mesh generator will create a mesh that honors the stream network by placing mesh nodes along each stream segment. The “PUMPING WELL” points are the locations of dewatering wells in and around the open pit. The mesh generator will honor these locations by placing a node at each one of the points. Adding additional wells to an existing **MINEDW** model is done within **MINEDW** and does not require the mesh to be re-created. For existing models, the user can move the closest mesh node to the location of the pump that is being added.

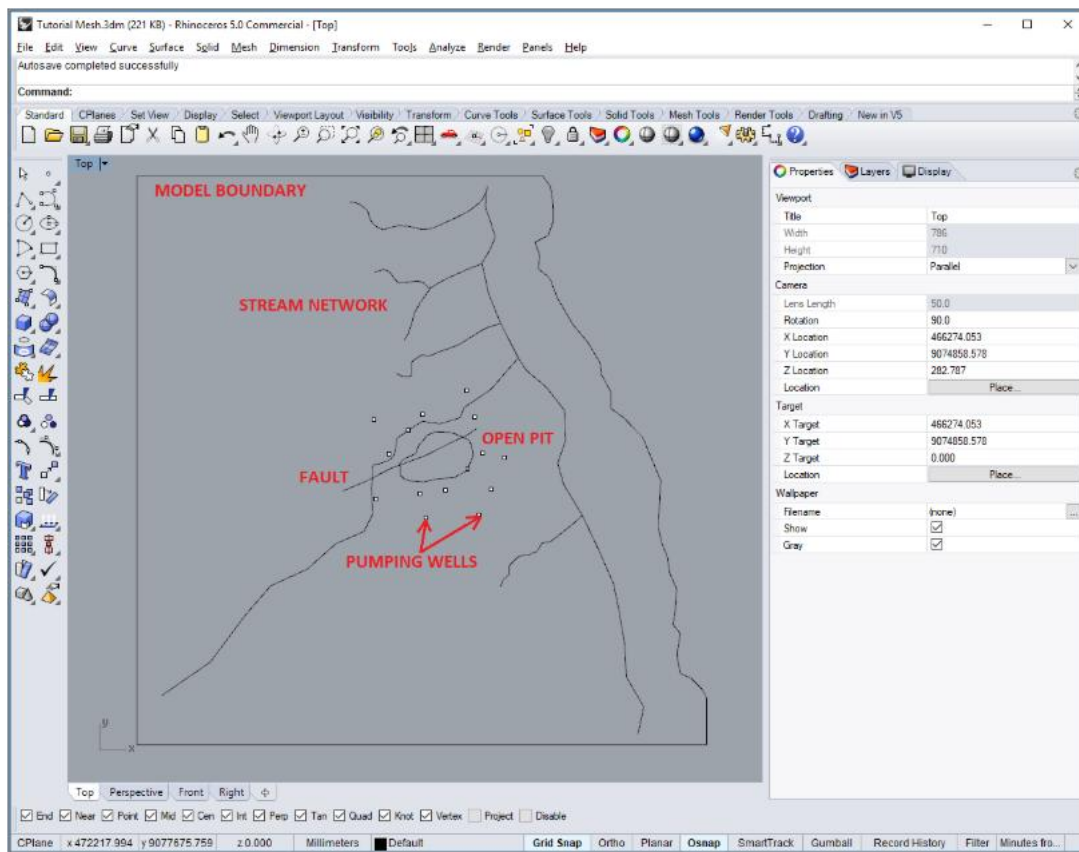


Figure 21. The overlay data used to create a mesh.

To create the transition zone, click on the "Polyline" tool on the "Standard" tab (Figure 22). Next, draw a circle around the "OPEN PIT" object. To finish and close the transition zone, click on the first point of the circle; the drawing tool will likely snap to this point once the mouse pointer gets close enough. The completed transition zone should appear similar to the one shown in Figure 22.

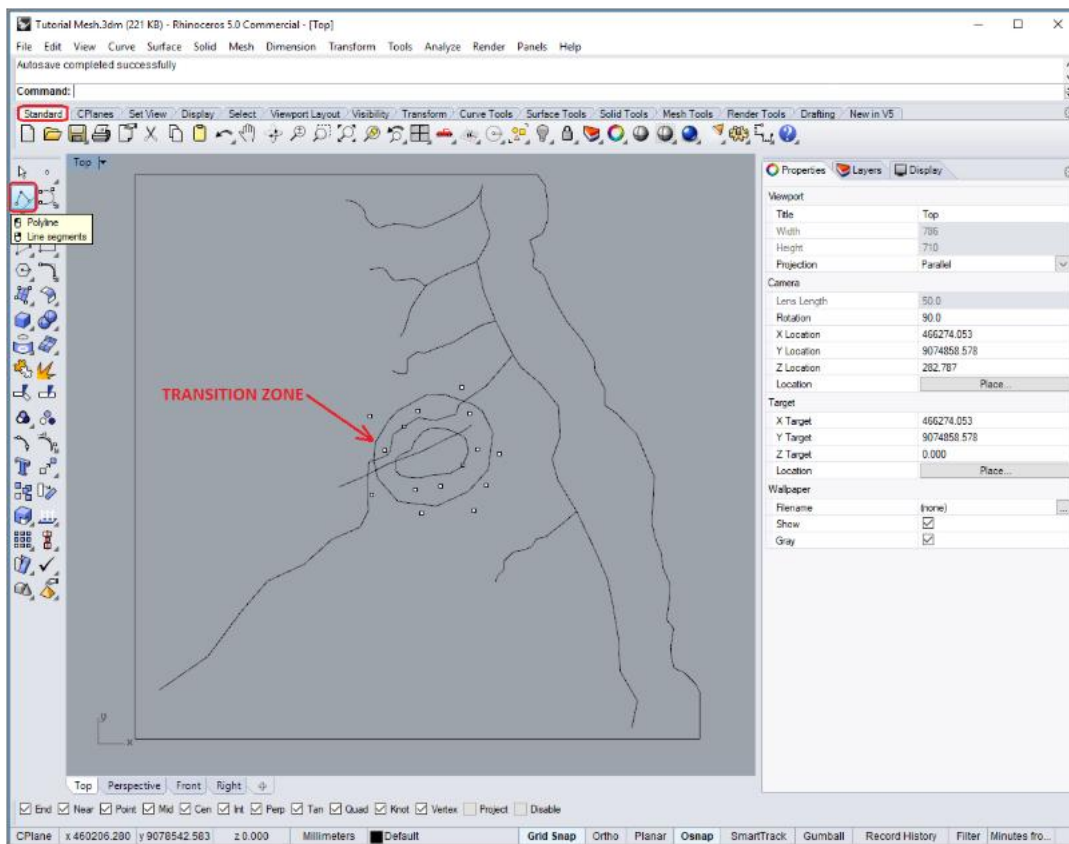


Figure 22. The completed transition zone.

After completing the transition zone, open Grasshopper by typing "grasshopper" into the command line and pressing [ENTER]. A new window will open for Grasshopper. If you are using Rhino 5 and Grasshopper is not installed yet, download and install Grasshopper for Rhino. Grasshopper does not require an additional license or fee. In Grasshopper, select "File>New Document" to create a new mesh algorithm. Next, click the "MMesh" tab. The "MMesh Util" contains three components, "Fault", "Region", and "MMesh". To create the finite-element mesh for this tutorial, one (1) "Fault" component, three (3) "Region" components, and one (1) "MMesh" component will be needed. Place these components on the canvas by clicking on them in the banner menu and then clicking on the canvas. Each component can be moved around on the canvas as necessary by clicking and holding then dragging the component to the desired location. Each component can be renamed by right-clicking on it and then entering a new name where the component's current name appears.

In order to complete the mesh, additional components are needed: one (1) "Panel", five (5) "Number Sliders", one (1) "Boolean Toggle", one (1) "Curve", one (1) "Point", and one (1) "Merge". All of these components, except for the "Merge" component, can be found on the "Params" tab under the "Geometry" or "Input" groups (Figure 23).

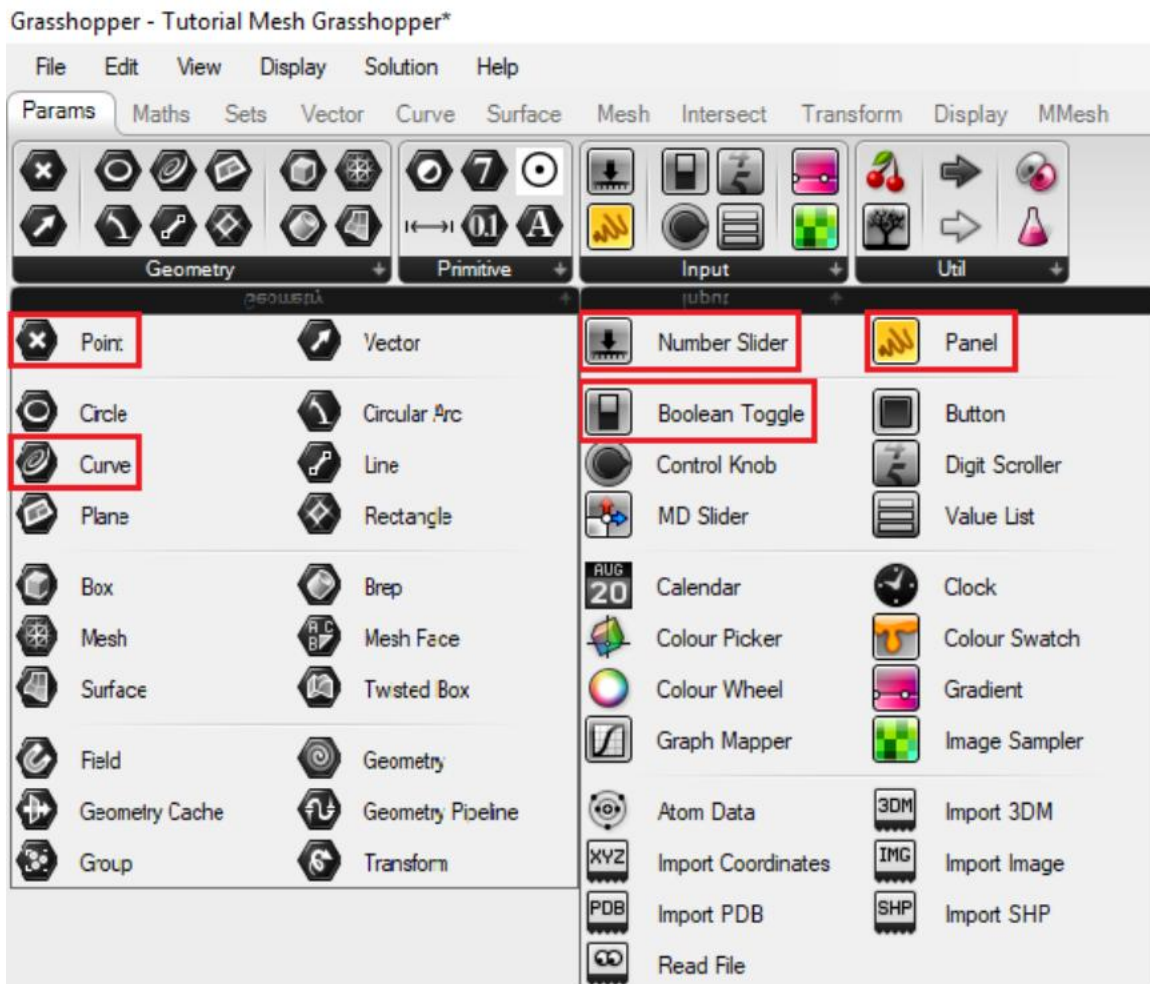


Figure 23. Grasshopper components used to create the finite-element mesh.

The “Merge” component is used to merge generic streams of data. For example, it is used in this tutorial demonstration to merge the three regions (“MODEL BOUNDARY”, “TRANSITION ZONE”, and “OPEN PIT”) into one data stream for processing. This component is found on the “Sets” tab under the “Tree” group (Figure 24).

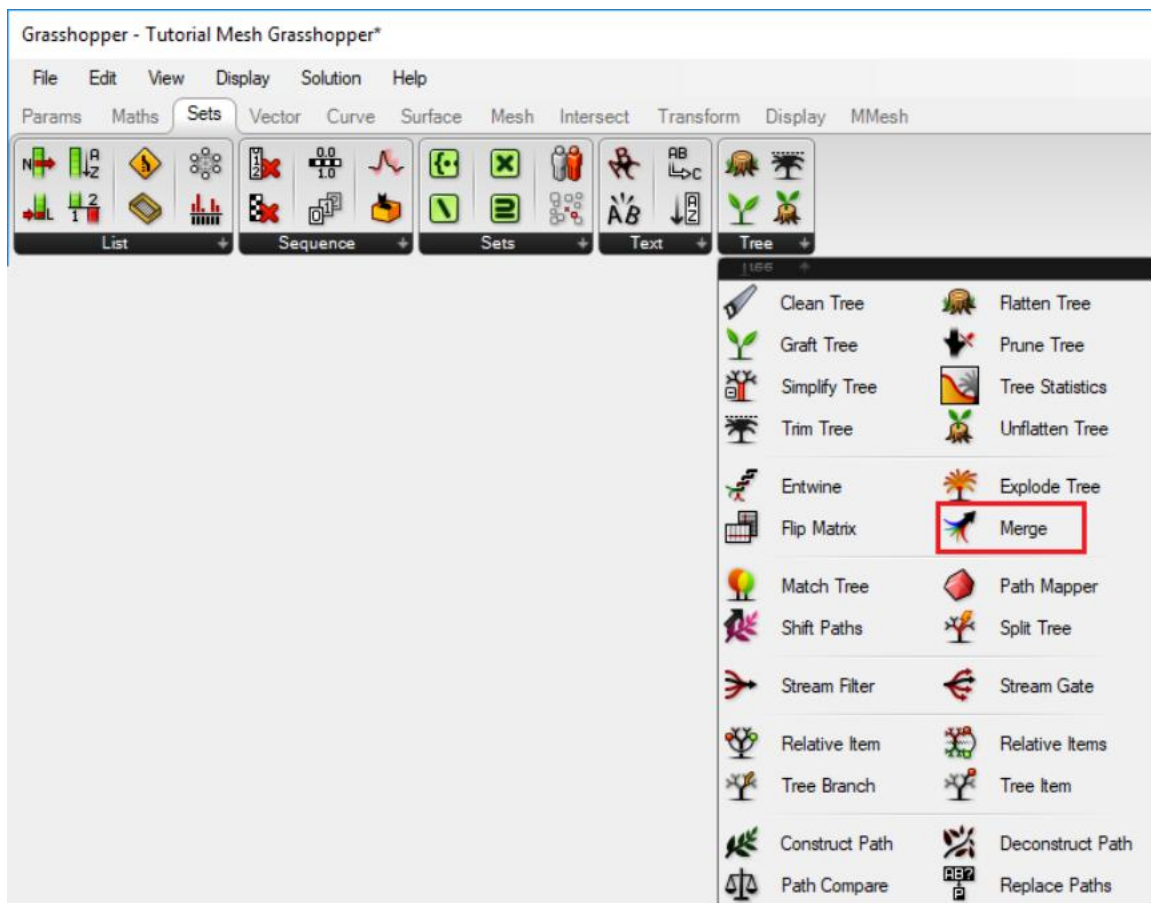


Figure 24. The “Merge” component in Grasshopper.

After adding all of the necessary components to the canvas, connect them together following the example in Figure 25. The easiest method for connecting components is to click on the half circles that appear to the left or right of the component parameter and drag them to the appropriate component. Figure 25 can be used as an example of how to arrange and connect the various components.



ITASCA Chile
Santiago, Chile

After saving the Grasshopper document, assign the "Polygon (PG)" and "Primary Region (PR)" parameters for the "Region" components. Note that the "Area (A)" parameters are defined by connecting the parameters to the number sliders. The "MODEL DOMAIN" curve will be assigned to the "PG" parameter of the "Model Domain" component, the "TRANSITION ZONE" curve will be assigned to the "Transition Zone", and the "OPEN PIT" curve will be assigned to the "Open Pit" component. To assign a component to a curve, right-click on "PG" and then select "Set One Curve" from the drop-down menu. The Grasshopper window will disappear to show the Rhino window. In the Rhino window, click on the appropriate curve and the Grasshopper window will reappear. Repeat these steps for the other regions.

By default, the "Primary Region (PR)" parameter will be assigned "False" for each of the "Region" components. For the "Model Domain" component, this will need to be changed, and the "PR" component will need to be assigned "True". To do this, right-click on the "PG" parameter of the "Model Domain" component and then select "Set Boolean>True" from the drop-down list. This parameter identifies the outermost boundary of the mesh to be created. As many regions as needed can be added to the mesh, but only one can be the primary region and must include (surround) all of the other regions.

To assign the fault, repeat the previously described steps, but instead of a "PG" parameter, "PL" is the parameter that will need to be set. The "Streams" component needs to be assigned to the "STREAM NETWORK" in the Rhino window. This assignment is done the same way as the other assignments previously described, but instead of selecting "Set One Curve", click on "Set Multiple Curves". Using this option, you can select all of the curves that represent the streams in the Rhino window. After selecting all of the curves, simply press [ENTER] to finish the assignment. The final component that needs to be assigned is the "Pumping Borehole", which requires points. Follow the same steps as previously described for the other components and select "Set Multiple Points". In the Rhino window, you can select each point representing the pumping boreholes by hand as you did with the streams, or you can use a Rhino function to do it all at once. To use the Rhino function, type "selpt" into the command line and press [ENTER]. Rhino will select the points and print out how many points were selected, in this case 15 points. After completing the selection, press [ENTER] again to complete the assignment.

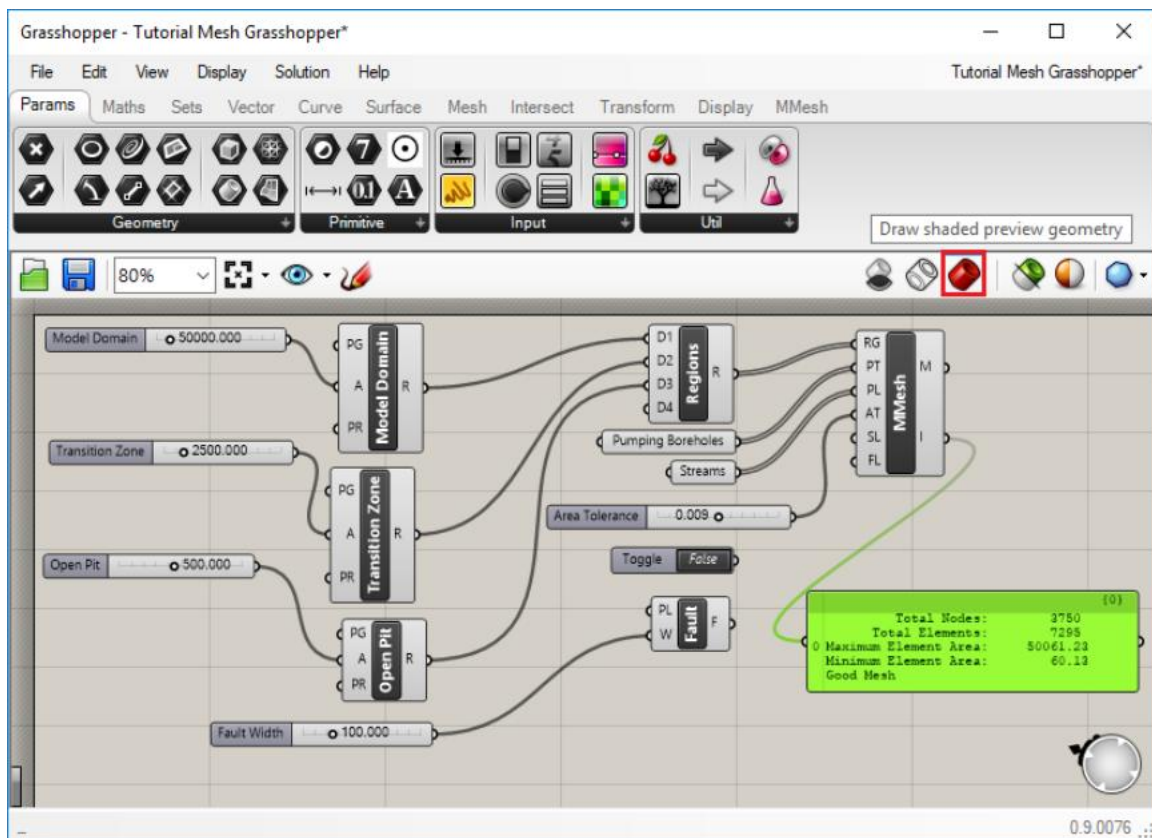


Figure 26. The output from the Grasshopper algorithm.

Once properly configured, the Grasshopper algorithm will generate a mesh that is visible in the Rhino window. If it is not visible, be sure to toggle the “Draw shaded preview geometry” as shown in Figure 26. If the mesh still does not display, check that “Preview Mesh Edges” is turned on by selecting it under the “Display” menu or pressing the [Ctrl] and [M] keys together. The “Panel” item that was connected to the “MMesh” region will display details about the mesh, such as the number of nodes and elements and the minimum and maximum size of the elements. Figure 27 is an example of what the mesh created by the Grasshopper algorithm should look like. Note, if you close the Grasshopper window, this mesh will disappear, but as long as you have saved the changes to the Grasshopper document, opening the Grasshopper document will regenerate this mesh.

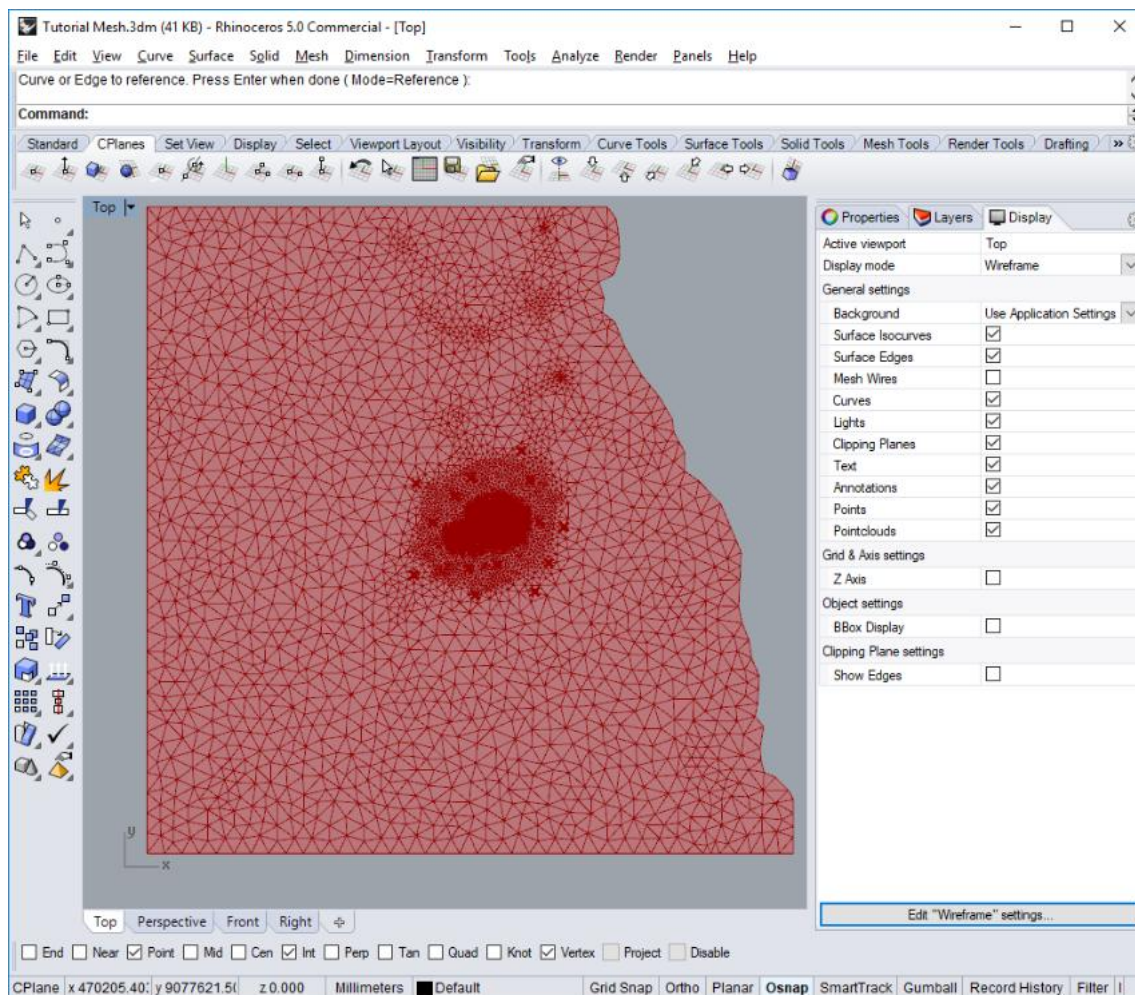


Figure 27. The Grasshopper-generated mesh.

In order to finalize the mesh and export it in a format that is useable by **MINEDW**, the Grasshopper-generated mesh needs to be "Bake[d]". To do this, right-click on the "MMesh" component in Grasshopper and select "Bake" from the drop-down menu. An "Attributes" window will open where you can select the layer where you wish to "Bake" the mesh. Any layer is acceptable. The result of this operation should look similar to that in Figure 28.

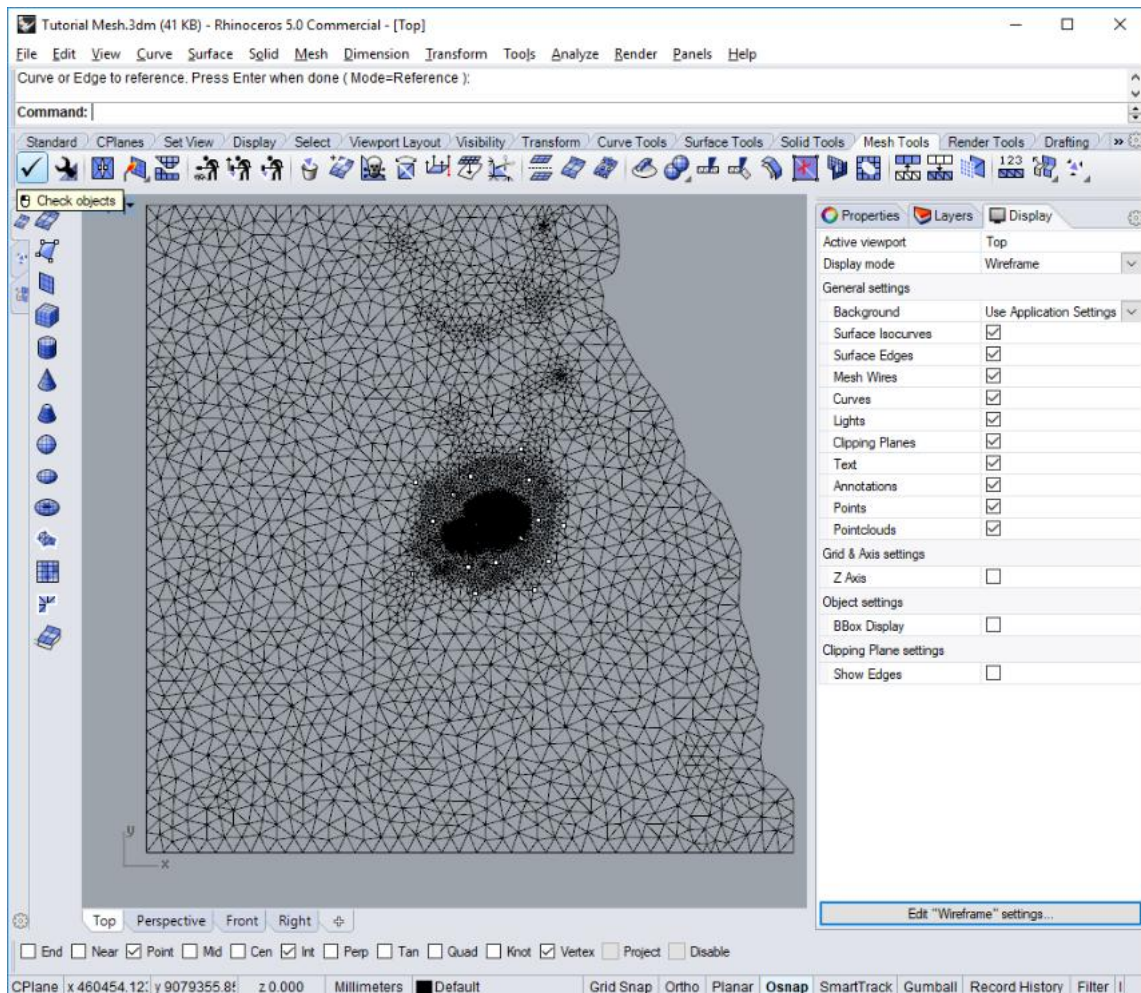


Figure 28. The completed mesh in Rhino.

After the mesh has been “Bake[d]” to Rhino, it should be checked to ensure there are no problems with the mesh. To check the mesh, click on the “Mesh Tools” tab and then click “Check objects” as shown in Figure 28. If there are problems with the mesh, these can be fixed using the mesh tools available in Rhino. Refer to Rhino documentation for instructions on using these functions.

Once the mesh is finished in Rhino, it needs to be exported as an .STL file for import into **MINEDW**. Select the mesh in Rhino by clicking on it, then select “File>Export Selected...” Using the “Export” dialog box, navigate to where you wish to save the mesh file, then enter a name and select “STL (Stereolithography)(*.stl)” from the “Save as type” drop-down box and click “Save”. Next, in the “STL Export Options”, select the option to export the .STL file in ASCII format, then click “OK”.

2.3 Building the model in MINEDW

Upon opening **MINEDW** the “**MINEDW** Startup” dialog box appears (Figure 29), click “OK” to create a new project, navigate to the location where you wish to save the model files. In the “File

Name” box, enter the name of the project, for example, “SSTUTORIAL”, and then click on “Save” to create the project file and close the dialog box.

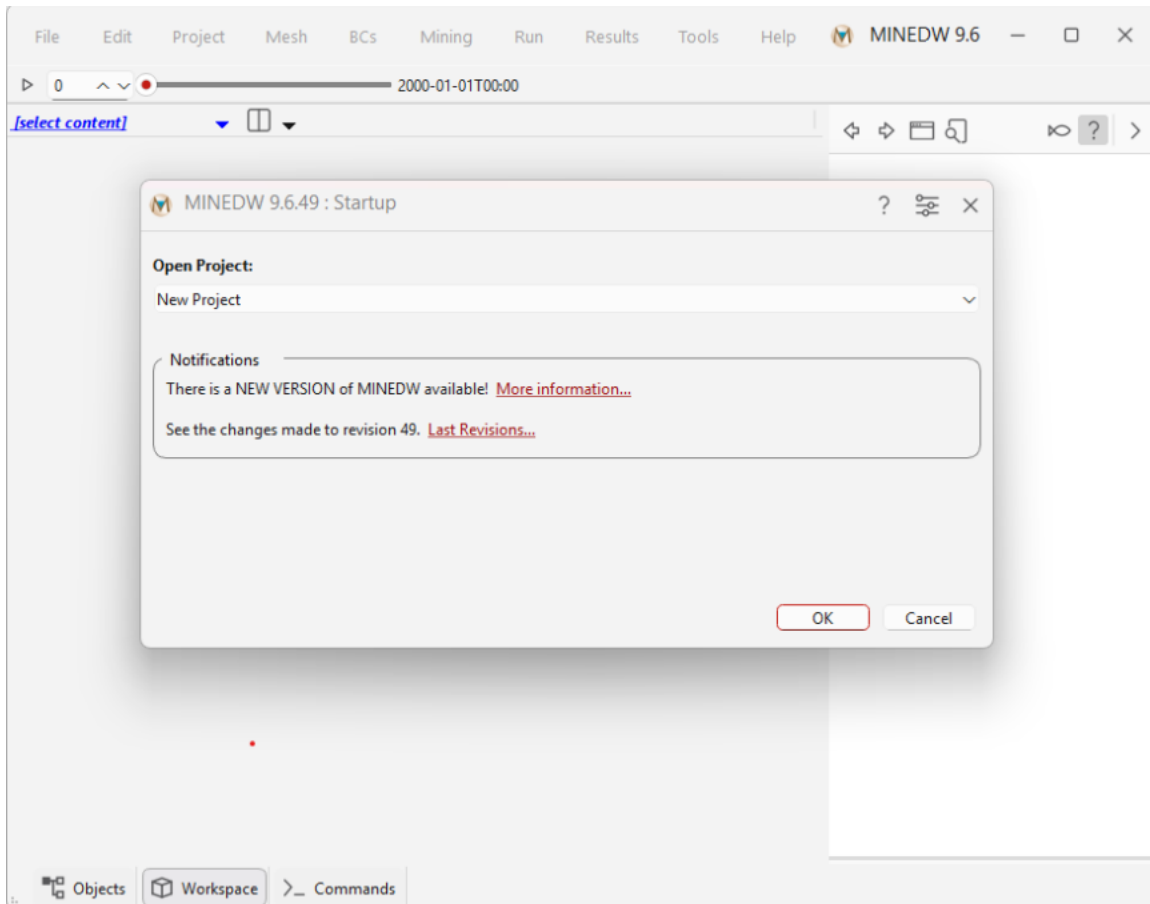


Figure 29. MINEDW Startup window.

Now you can import the created mesh into **MINEDW**. Click “File” on the Main Menu banner, and then select “Import Model From”. The “Import File” dialog box appears (Figure 30). Select “STL (Stereolithograph) Ascii Files (*.STL)” from the list of options under the “Files of Type” drop-down list. Navigate to the location where you created the mesh file, select it, and click “Open”. IMPORTANT: No mesh will appear in **MINEDW** at this time; a few more steps must be taken before the mesh is displayed, but you will see the message “STL mesh was imported successfully” and click “OK”.

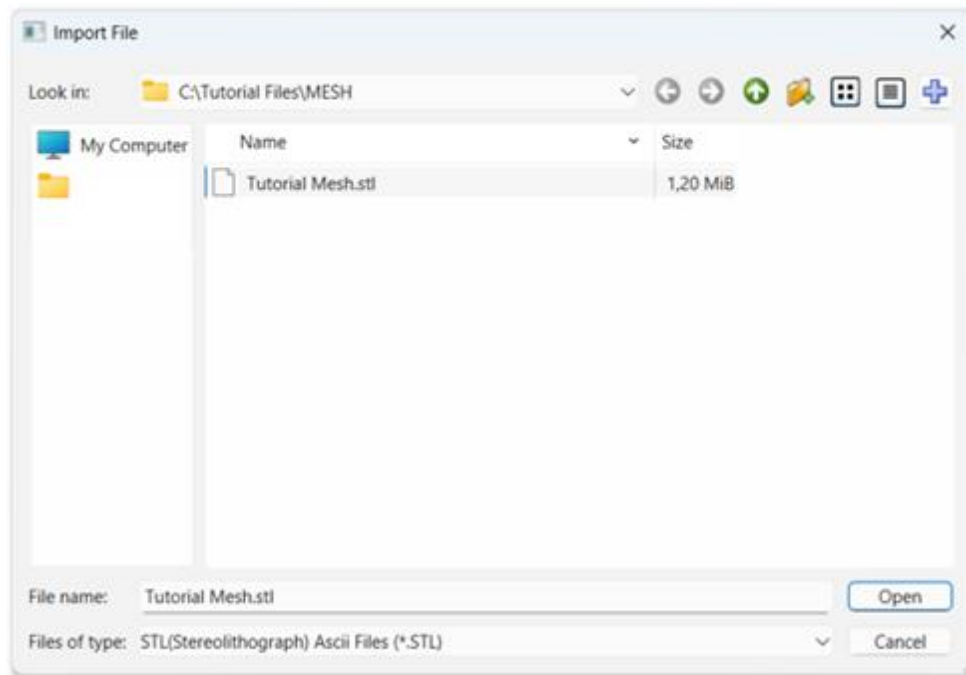
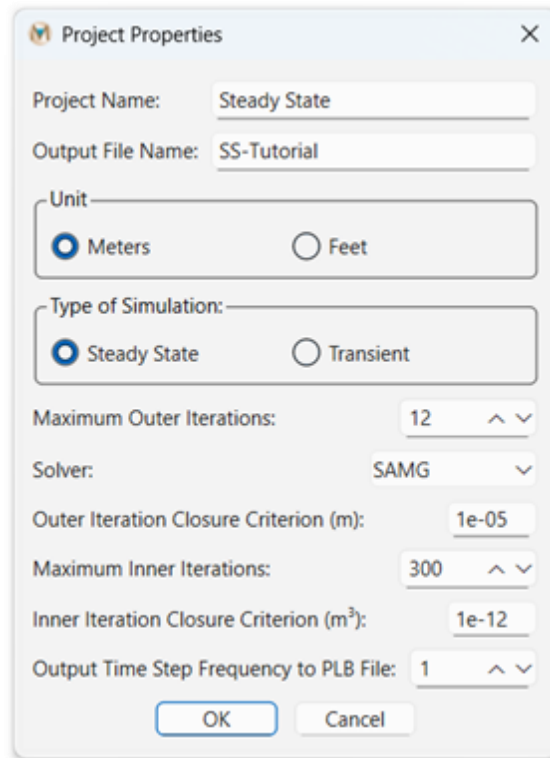


Figure 30. The "Import File" dialog box.

2.3.1 Step 2. Defining Project Properties

To define the project properties such as units (meters or feet), type of simulation, closure criteria, and type of solver, click "Project" and then select "Project Properties". The "Project Properties" dialog box shown in Figure 27 appears. Make sure that all the fields contain the values shown in Figure 31. For steady-state simulations, the "Steady State" option should be selected.

For this simulation, the SAMG solver will be used. The outer closure criterion is based on head, while the inner closure criterion is based on mass. Because of this, the inner closure criterion needs be less than 1×10^{-10} , while the outer closure criterion is likely to be greater than 1×10^{-3} . Differences in the closure criterion selection will affect the residual error in the model. Users should check the budget (.BUD) and output (.OUT) files to evaluate the accuracy of the model run and whether more stringent closure criteria are necessary. Maximum inner iterations should be large (i.e., greater than 250); however, the user can evaluate the optimal amount by consulting the **MINEDW** output file.



Project Properties

Project Name: Steady State

Output File Name: SS-Tutorial

Unit

☒ Meters ☐ Feet

Type of Simulation:

☒ Steady State ☐ Transient

Maximum Outer Iterations: 12

Solver: SAMG

Outer Iteration Closure Criterion (m): 1e-05

Maximum Inner Iterations: 300

Inner Iteration Closure Criterion (m³): 1e-12

Output Time Step Frequency to PLB File: 1

OK Cancel

Figure 31. The “Project Properties” dialog box.

2.3.2 Step 3. Defining Time Steps

To define the time steps for the project, click “Project” on the Main Menu banner, then select “Time Steps”. The “Set Up Time Step” dialog box (Figure 32) appears. Set the “Maximum # of Time Steps” to 100 or greater. Leave “30.875” as the “Initial Time Step Length”. For steady-state simulations, only the length of the first time step will be used. Each subsequent time step will be 1.2 times longer than the previous time step. For transient simulations, the multiplication factor is 1 and the time-step length is defined in the “Set Up Time Step” dialog box for transient model runs. Ensure that the “# of Time Steps for This Simulation” slider is set to 100 or greater and that the “Start Time Step” combo box is set to 1. Click “OK” to save the changes and close the “Set Up Time Step” dialog box.

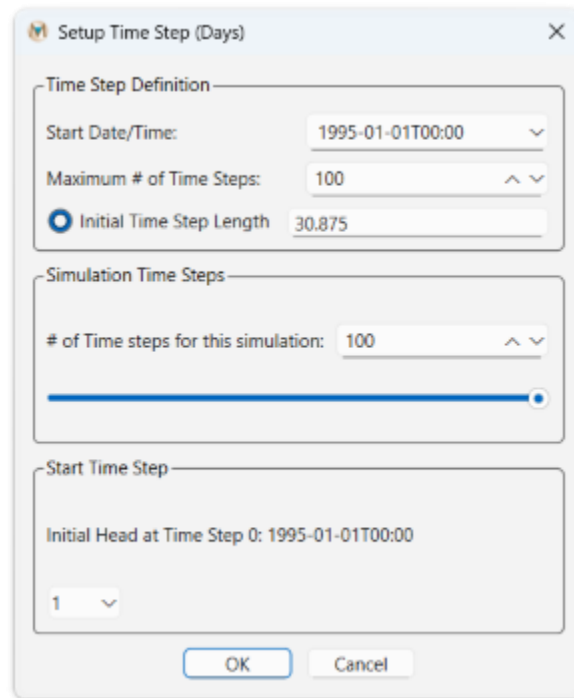


Figure 32. The "Set Up Time Step" dialog box.

2.3.3 Step 4. Importing Ground-Surface Elevation

To visualize the mesh, first create a plot pane by right-clicking the **MINEDW** View Pane, then selecting "New" > "Plot" > "OK". After that, right click the plot pane area and select "Add Plot Item" (Figure 33). The "Build Plot" dialog box appears (Figure 34). Under the "Node" category, select "2D Contour" and click "OK". The mesh will be displayed in the plot pane. You can rotate it in space freely by holding left click and dragging your mouse, or you can select a standard view angle by clicking the "Plot Views and Controls" button in the right side of the screen, and then "Apply a standard view" button within the "View Stats" section.

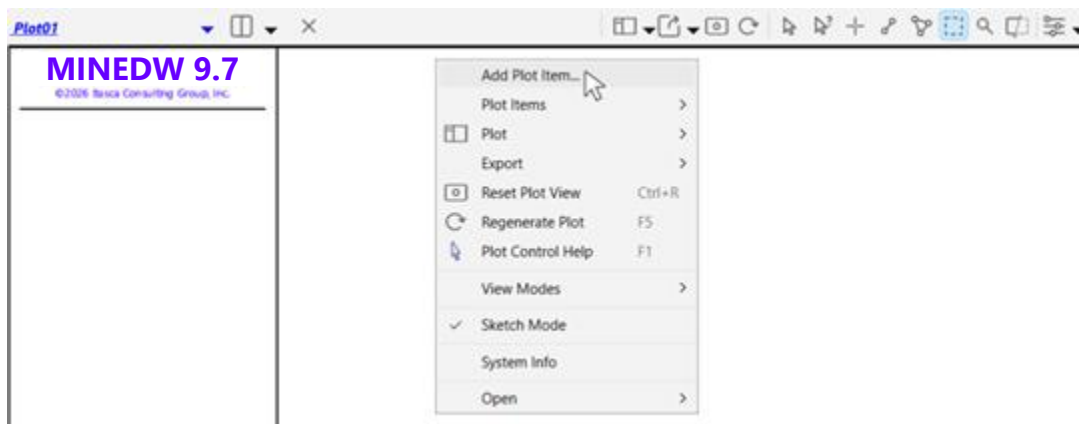


Figure 33. The "Add Plot Item" button.

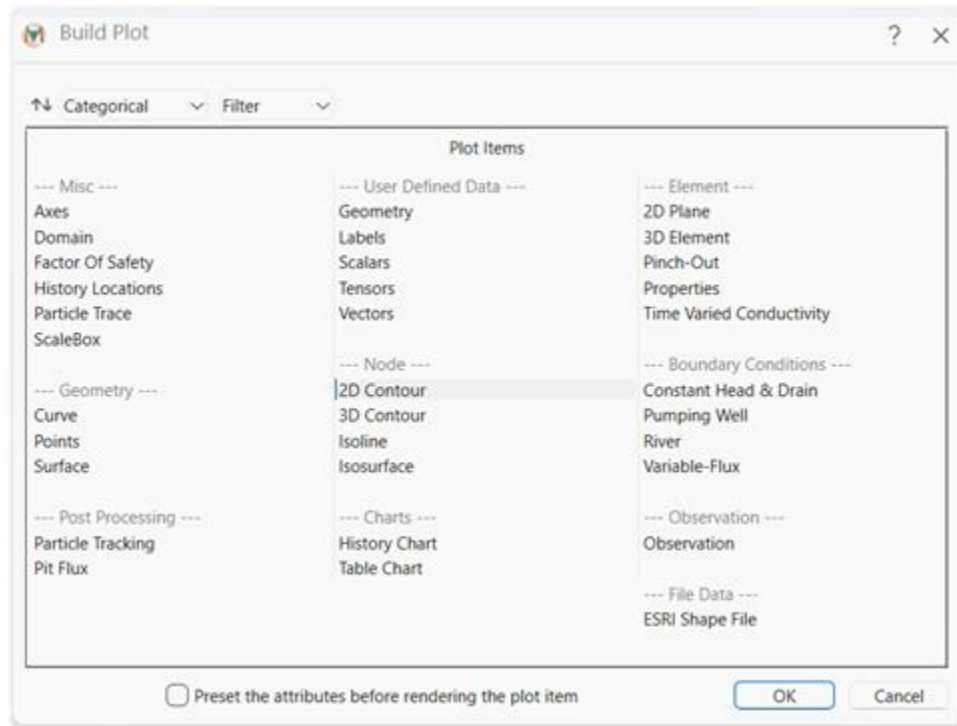


Figure 34. The “Build Plot” dialog box.

From the “Plot Objects Panel” pane on the right-hand side of the **MINEDW** interface, under the “Attributes” tab, make sure that the “Color By” attribute is set to “Elevation” and that the “Layer” attribute is set to 1 (Figure 35).

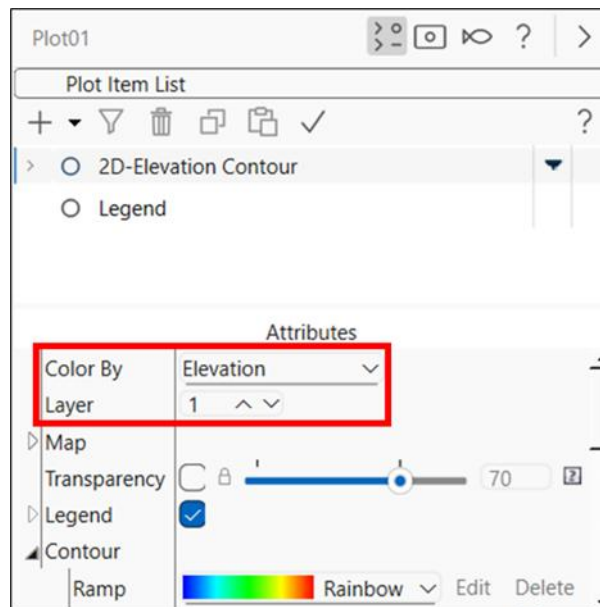


Figure 35. The “Control Panel” pane showing the “Attributes” tab with “Layer” set to “1”.

In the **MINEDW** View Pane, click on the “Select with Rectangle” tool (Figure 36) and use the cursor to click and drag a box around the whole domain to select all the nodes in Layer 1.

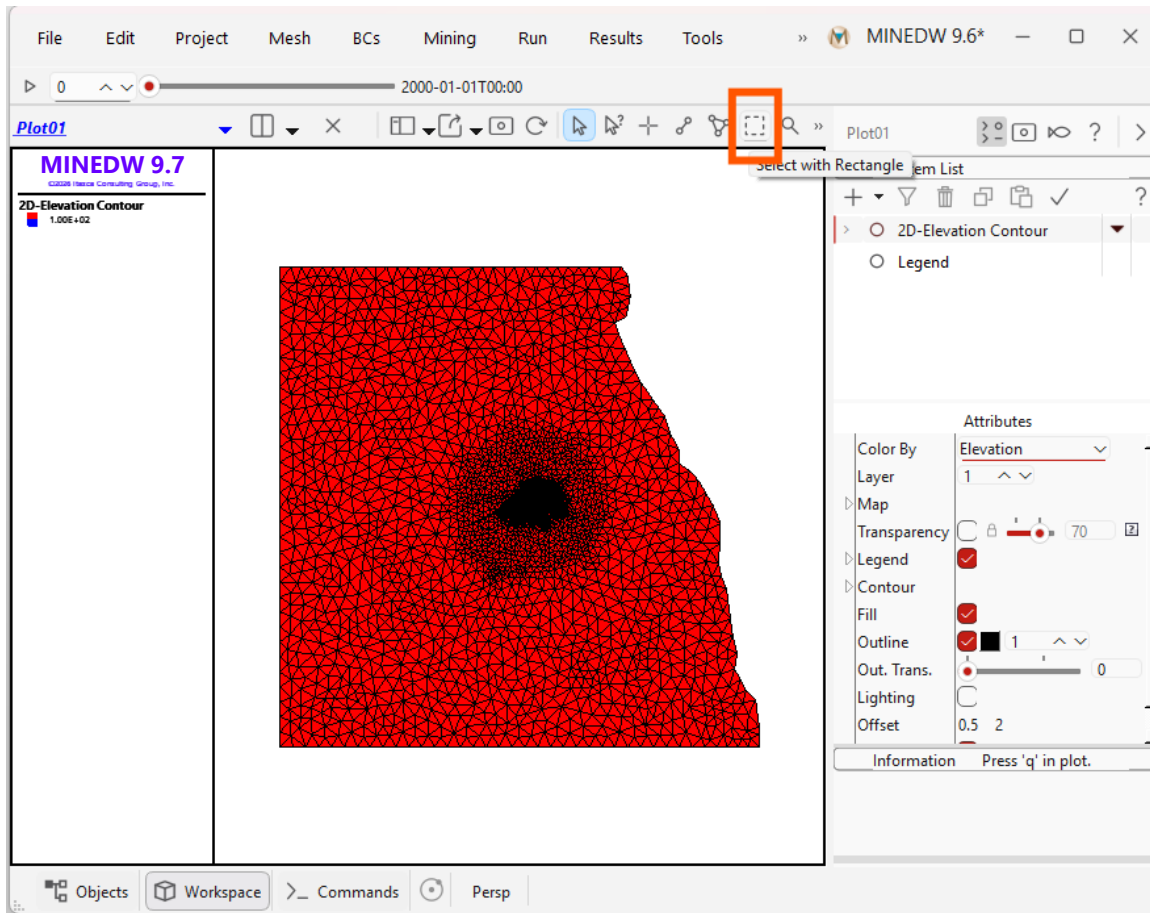


Figure 36. Select with rectangle tool.

Press the [Enter] key to open the “Assign Properties for Nodes” dialog box shown in Figure 37. Click “Interpolate From File” and the “Open Data File” dialog box appears. Select the data file in the directory “/INPUT/SURFACE” named “SURFACEELEV.DAT” and click “Open”. The “Grid” dialog box shown in Figure 38 appears. Define the interpolation method (Inverse Distance or Kriging) and the required parameters and then click “OK”. The interpolation procedure assigns the ground-surface elevation contained in the “SURFACEELEV.DAT” file to the first layer of the model.

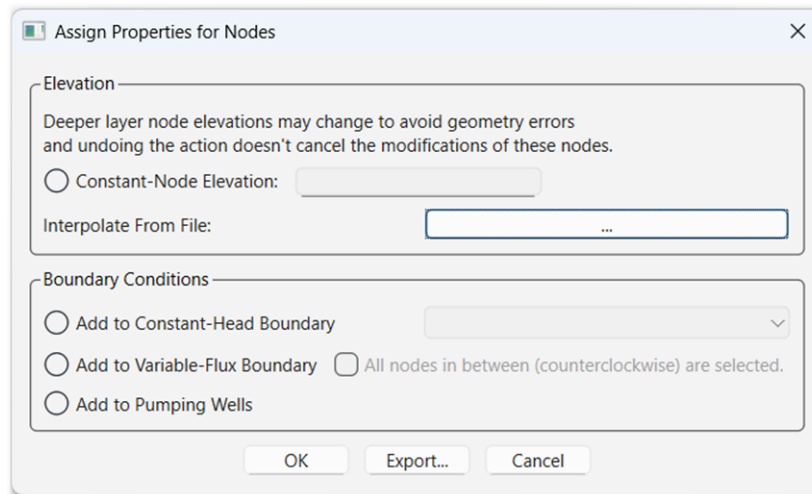


Figure 37. The “Assign Properties for Nodes” operations dialog box.

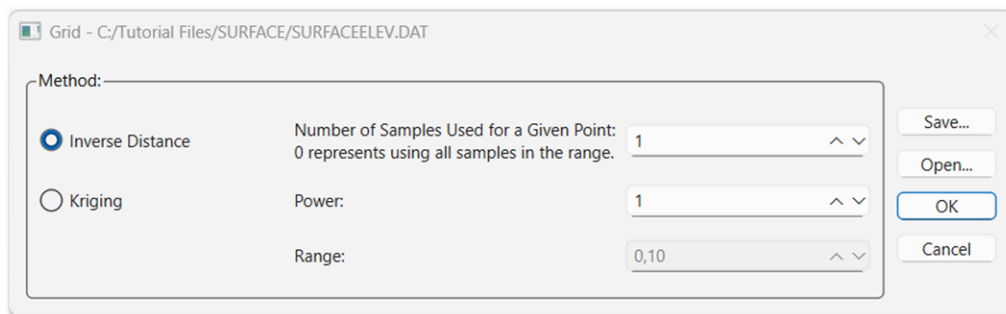


Figure 38. The “Grid” dialog box and example values for surface interpolation.

View the ground-surface elevation as contours by going to the “Attributes” tab in the “Plot Objects Panel” pane. Next, click on the small triangle next to “Contour”. Deselect the “Auto” box next to “Minimum” and replace the default value with 800. You may also wish to change the contour interval; smaller numbers produce a smoother color flood. Press [Enter] and the model will appear similar to what is displayed in Figure 39.

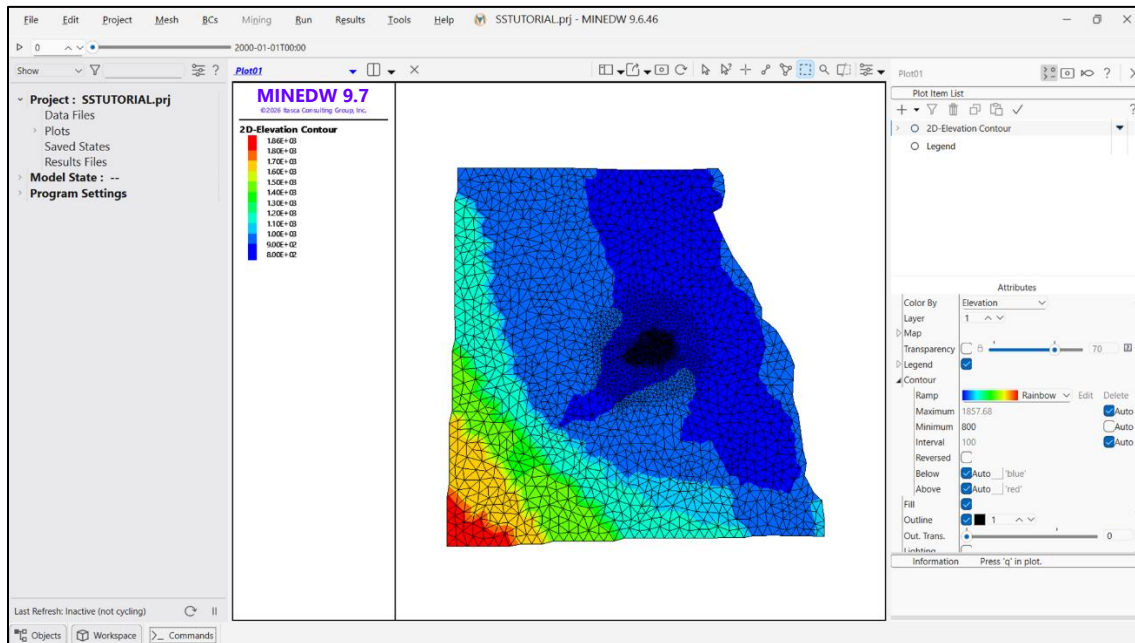


Figure 39. MINEDW color flood of topography.

2.3.4 Step 5. Adding Layers

To add layers, select "Mesh" from the Main Menu banner, then select "Define Main Layer". The "Define Main Nodal Layers" dialog box (Figure 40) appears. Select the bottom layer. To add a node layer above the selected node layer, click "Insert". For this model, three layers will be created using the "Depth" method with thicknesses of 20, 30, and 50 m (Figure 41). Next, add three more layers but choose the "Average" method. Finally, add 11 more layers for a total of 19 layers and choose "Constant" from the "Method" drop-down box. For the layers with the "Constant" method selected, enter the values displayed in Figure 1.20. **Note:** Do not change the elevation of the first node layer because its elevation was assigned by interpolation. Once you have finished entering the required information, click "OK" to add the layers. **MINEDW** will produce a warning stating that the ZOR may need to be updated. Disregard this message, as no ZOR has been defined. For more information on adding layers, refer to the User Manual section 7.2.2.

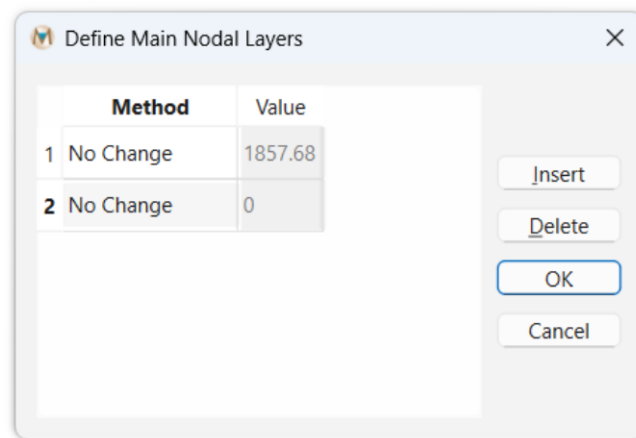


Figure 40. The "Define Main Nodal Layers" dialog box.

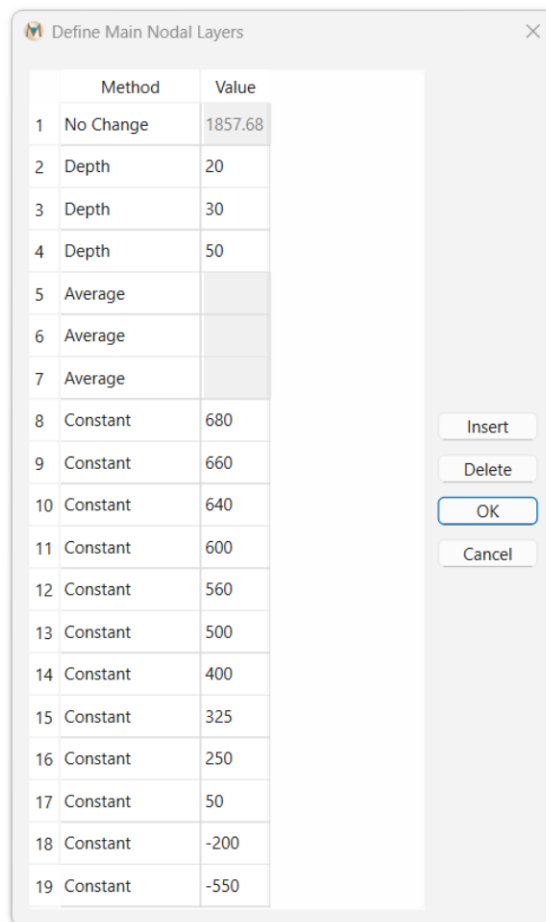


Figure 41. The "Define Main Nodal Layers" dialog box showing different methods available.

2.3.5 Step 6. Defining Hydrogeological Units

To define the properties of a hydraulic zone, click "Project" on the Main Menu banner and then select "Hydrogeological Units". The "Hydrogeological Unit Properties" dialog box (Figure 42) appears. To add a new unit, click the "Add" button. Enter the units and related hydraulic properties as listed in Figure 42 and then click "OK".

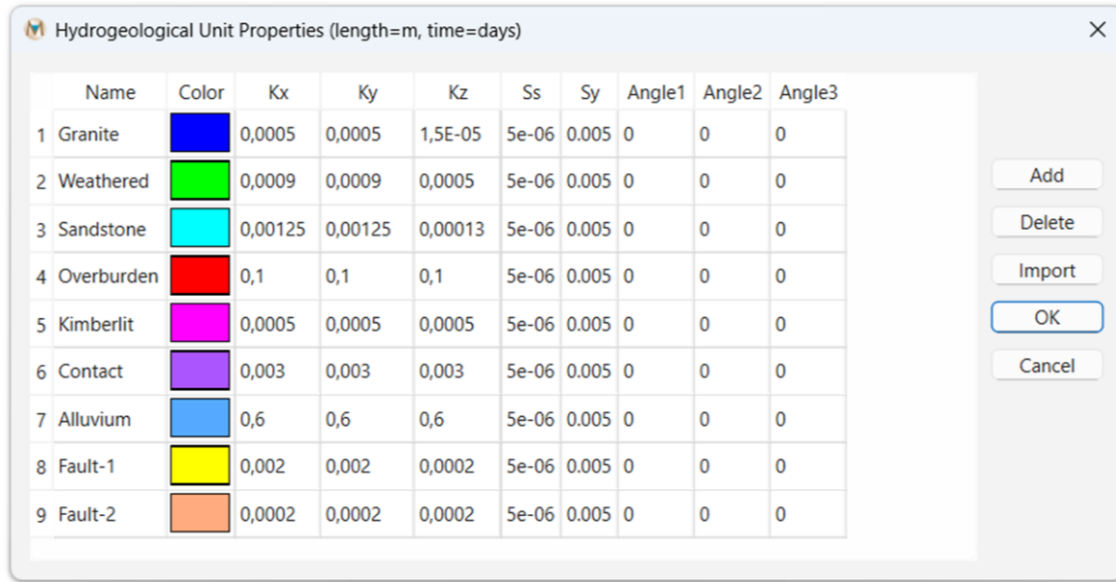


Figure 42. The "Hydrogeological Unit Properties" dialog box.

2.3.6 Step 7. Assigning Hydraulic Zones to Elements

Right click the **MINEDW** plot pane area and select "Add Plot Item". Under the "Element" category, select "3D Element", as shown in Figure 43 and click "OK". While working on element-related operations, such as assigning hydraulic zones, ensure that all other plot items related to "Node", such as a "Contour" plot, are deactivated or deleted from the View Pane (**MINEDW** Manual 3.4.1).

In **MINEDW**, the assignment of hydraulic zones can be done using a 3-D .DXF with a wireframe format of a geological unit (Figure 44). In this tutorial, hydraulic zones will be assigned in two steps. First, regional geology will be assigned by layers and then a kimberlite and a contact zone will be assigned using a 3-D wireframe .DXF file. The regional geology consists of overburden, weathered bedrock, sandstone, and granite. To assign the regional hydraulic zones, click the "Select" tool. Make sure that "Layer" under the "Attributes" tab shown in Figure 45 within the "Plot Objects Panel" pane is set to 1.

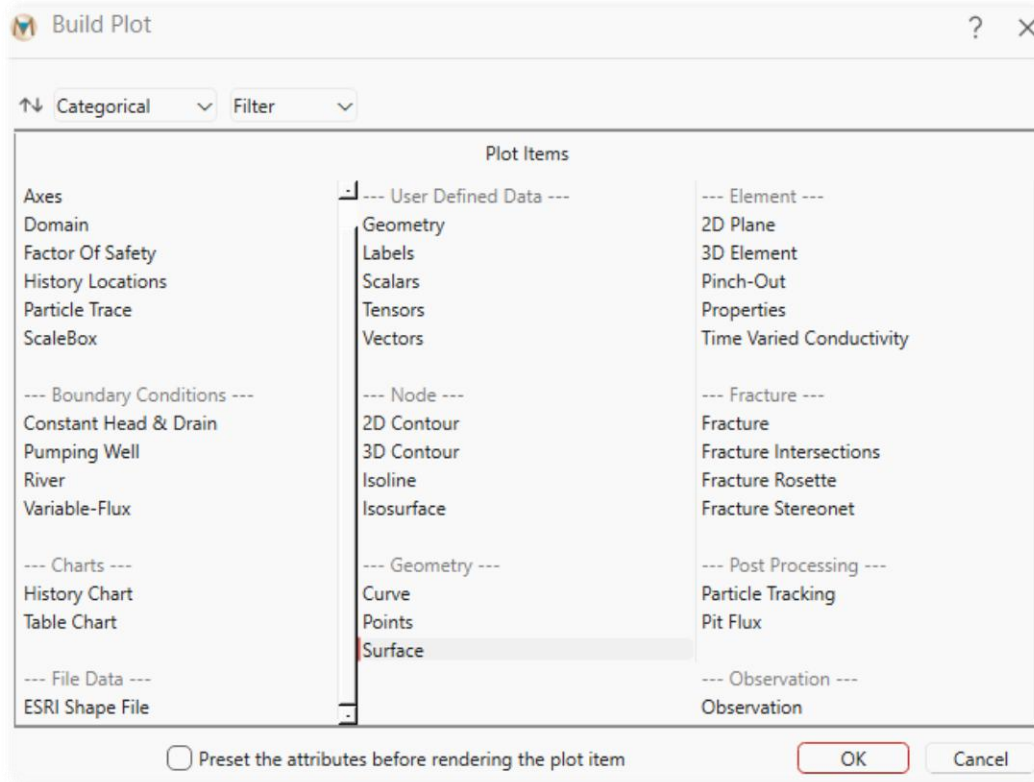


Figure 43. The "Build plot" Dialog box.

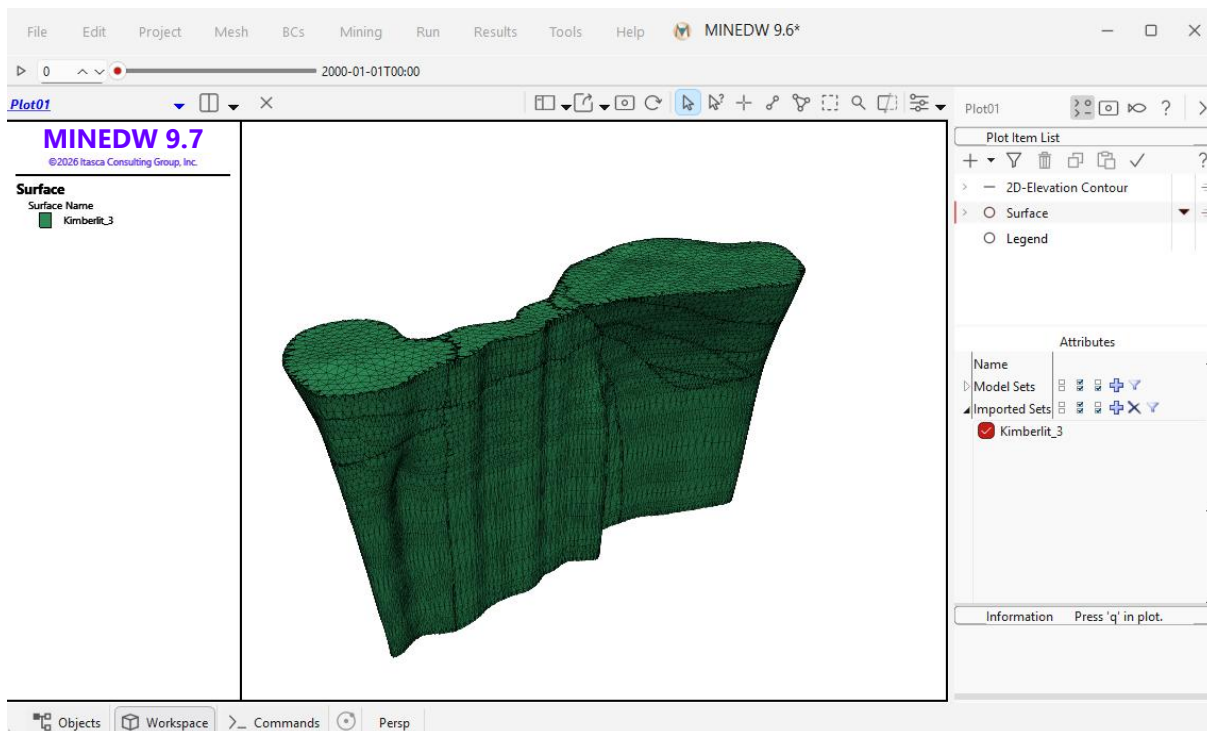


Figure 44. A wireframe format of a geological unit.

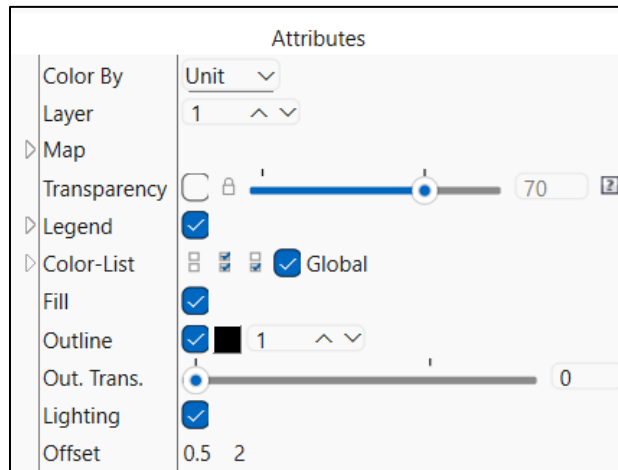


Figure 45. The "Attributes" tab with "Layer" set to 1.

Select all the elements on the first layer by clicking on the "Select with Rectangle" tool and using the cursor to click and drag a selection box around the whole domain (Figure 46), then press the [Enter] key.

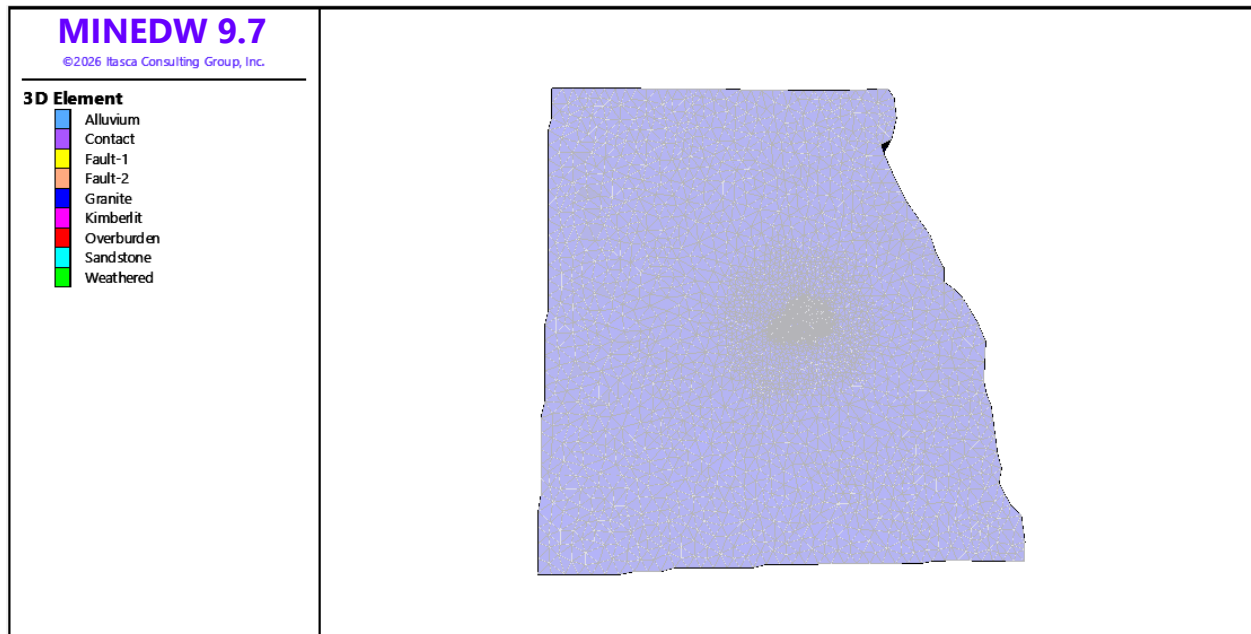


Figure 46. Selection of first layer of elements.

The "Select Hydrogeological Unit:" dialog box shown in Figure 47 appears. Change the "Zone" to "Overburden", change the value in the box next to "From Current Layer #1 To" from 1 to 2, and then click "OK".

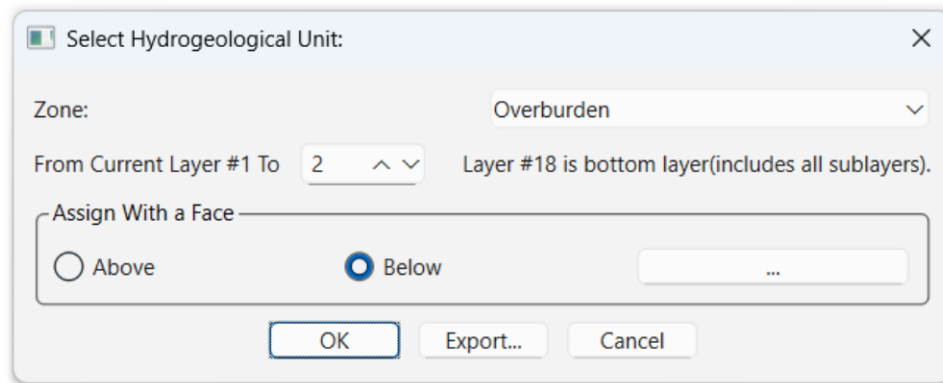


Figure 47. The “Select Hydrogeological Unit:” operations dialog box.

To assign the weathered unit, change the value for the “Layer” attribute on the “Attributes” tab to 3. Select all the elements on the third layer by using the cursor to click and drag a selection box, then press the [Enter] key. The “Select Hydrogeological Unit:” operations dialog box appears. Change the “Zone” to “Weathered”, leave the value in the box next to “From Current Layer #3 To” as 3, and then click “OK”.

The “Sandstone” unit is below the “Weathered” unit. To assign it to the model, click the “Select” button. Make sure that the “Layer” attribute on the “Attributes” tab under the “Control Panel” is set to 4. Select all the elements in the fourth layer by using the cursor to click and drag a selection box and then press the [Enter] key. The “Select Hydrogeological Unit:” operations dialog box appears. Change the “Zone” to “Sandstone” and the value in the “From Current Layer #4 To” box to 6 and then click “OK”. Finally, assign “Granite” using the same steps as above, from “Layer 7” to “Layer 18”, if it has not already been done.

This model also includes an alluvial zone in a river channel, two fault zones, and a contact zone between the kimberlite pipe and surrounding geology. The alluvial zone representing the river channel can be selected using a .DXF file. To select elements in the river channel, open the “Build Plot” dialog box by right clicking the plot pane area and selecting “Add Plot Item”. Under the “Geometry” category, double-click “Surface” and then, on the “Attributes” tab, click the “+” next to “Imported Sets”. Browse to “/MESH INPUT/DXF” and open the “STREAM_ALLUVIUM_Polyline.dxf” file. The polygon is automatically imported to elevation=0, so it will be hidden inside the model; to visualize it, click on the triangle next to the “Map” section, under the “Attributes” tab, and translate its Z coordinate by changing the third zero next to “Translate” to a number higher than the topography (Figure 48).

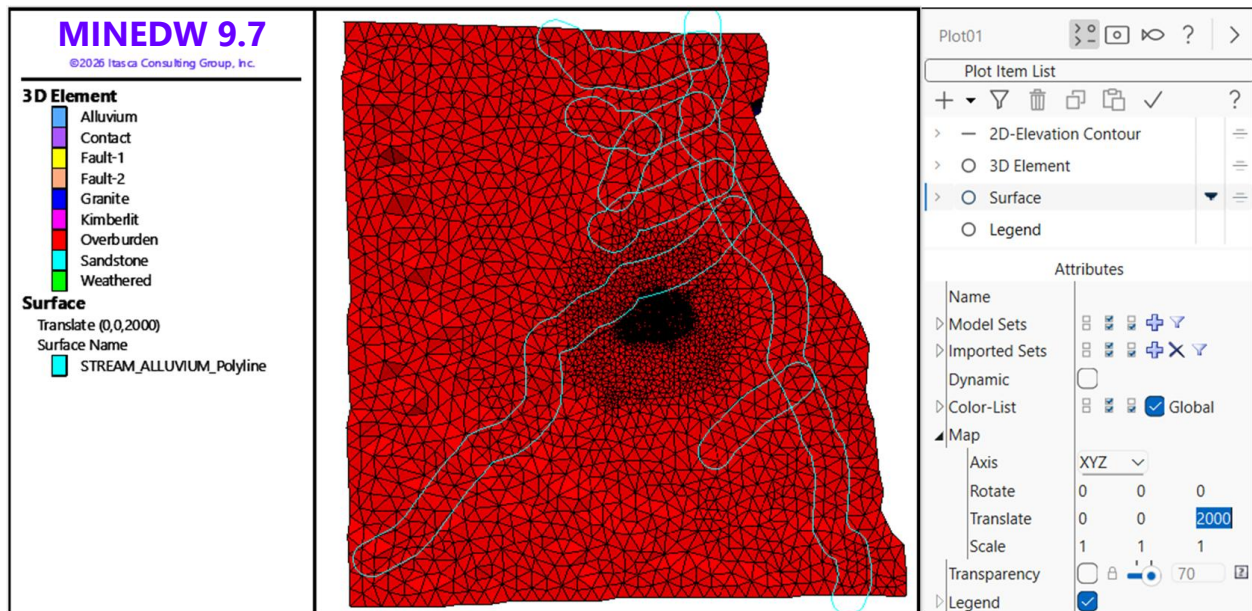


Figure 48. Alluvium polyline translated 2000 m upward along the Z-axis.

Next, click on the triangle next to "Imported Sets" to expand it, right click on the alluvium set and then click on "geometry-space DV(1,0,0)" (Figure 49) to create a virtual space of selection along the vertical axis. Then, in the "Objects" pane on the left-hand side of the **MINEDW** interface, expand the "named ranges" category within "Model State", right-click on "STREAM_ALLUVIUM_Polyline-oddX" and then click "Apply to Elements" to create a group. Next, expand the "groups" category, expand "elements" under it, right-click on the alluvium element and click "Assign Property" (Figure 50).

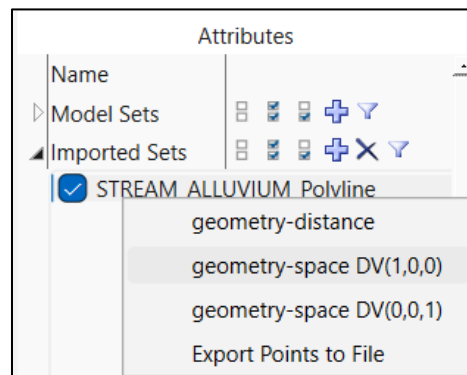


Figure 49. Creation of a virtual geometry-space along the vertical axis for the alluvium polyline.

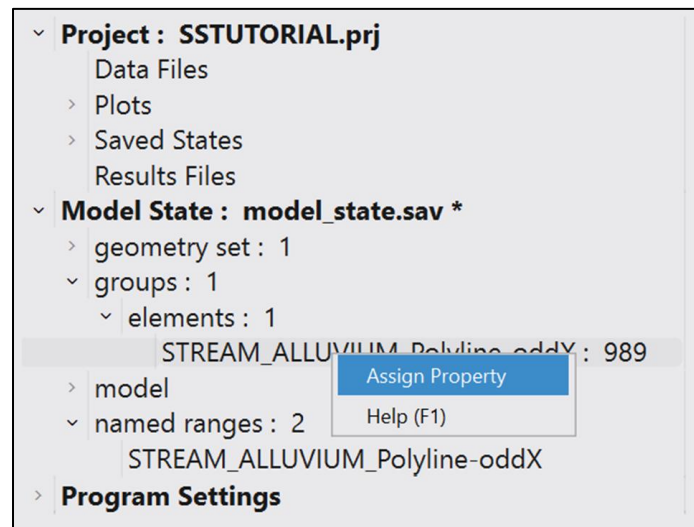


Figure 50. Place to assign properties for the alluvium unit, within the “Objects” pane.

The “Assign Properties for Elements” dialog box appears (Figure 51), select “Alluvium” as the Hydrogeological Unit and then click “OK”.

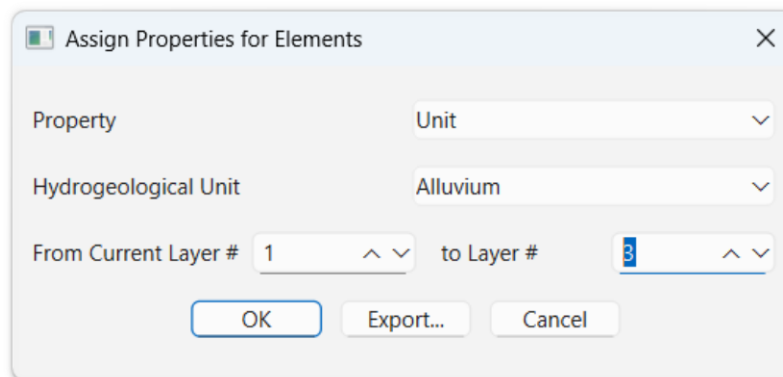


Figure 51. The “Assign Properties for Elements” dialog box.

To assign the “Kimberlite” and “Contact” zones using a 3-D .DXF repeat the same procedure: select the existing “Surface” plot item, under “Attributes” click on the “+” next to “Imported Sets”, browse to “/GEOLOGY” and open the “Kimberlit.dxf” file.

Next, click on the triangle next to “Imported Sets” to expand it, right click on the Kimberlit set and then click on “geometry-distance” (Figure 52) to create a virtual space of selection around the geometry faces. Then, in the “Objects” pane on the left-hand side of the **MINEDW** interface, expand the “named ranges” category within “Model State”, right-click on “Kimberlit-distance” and then click “Apply to Elements” to create a group. Next, expand the “groups” category, expand “elements” under it, right-click on the alluvium element and click “Assign Property”. The “Assign

Properties for Elements" dialog box appears, select "Contact" as the Hydrogeological Unit and then click "OK".

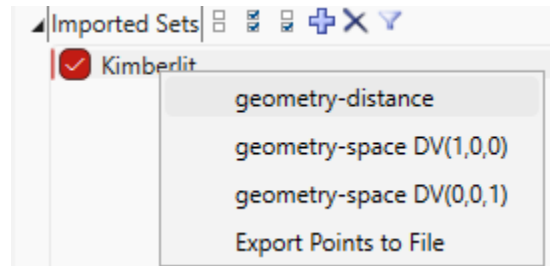


Figure 52. *Creation of a virtual geometry-space at certain distance from the geometry faces.*

Then, right click again on the Kimberlit set and then click on "geometry-space DV(0,0,1), in the "Objects" pane expand the "named ranges" category within "Model State", right-click on "Kimberlit-oddZ" and then click "Apply to Elements" to create a group. Next, expand the "groups" category, expand "elements" under it, right-click on the alluvium element and click "Assign Property". The "Assign Properties for Elements" dialog box appears, select "Contact" as the Hydrogeological Unit and then click "OK".

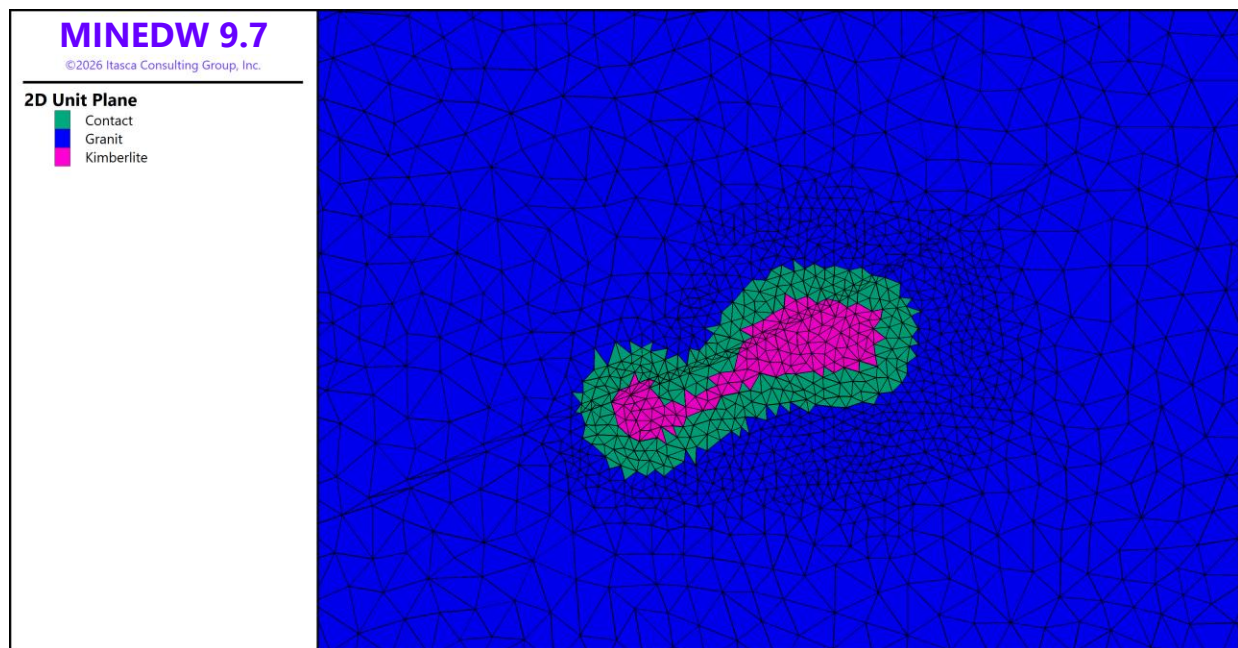


Figure 53. *View of "Contact" and "Kimberlite" hydrogeological units after assignation from DXF.*

2.3.7 Step 8. Defining “Pinch-Outs”

MINEDW is able to simulate locally refined vertical discretization in areas of interest. The enhanced discretization is referred to as a pinch-out since layers pinch out where the vertical discretization is not needed. Defining pinch-outs in the steady-state model is necessary because hydraulic heads from the steady-state model will be used as initial conditions for the transient model run, which requires that both models have identical meshes. To define the pinch-out types, select “Mesh” from the Main Menu banner and then select “Define Pinch-Out...” The “Define Pinch-Outs” dialog box appears (Figure 54).

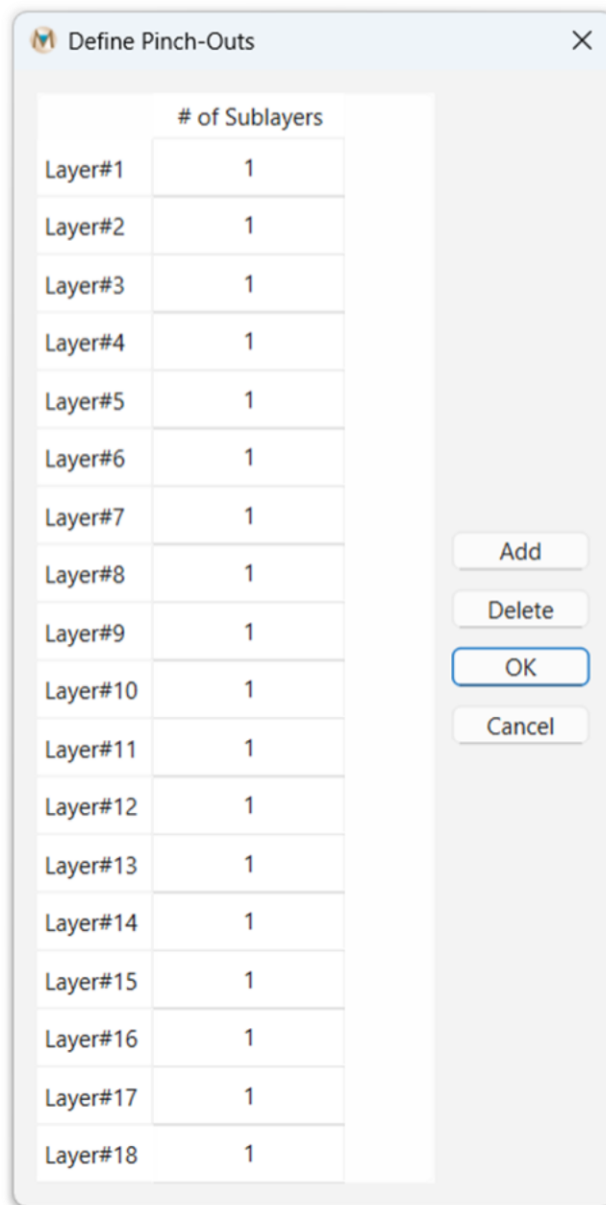


Figure 54. The “Define Pinch-Outs” dialog box.

In **MINEDW**, various pinch-out configurations can be defined and simulated. Each configuration is defined as one type of pinch-out. Based on modeling needs, the user can define as many pinch-out configurations as necessary.

Figure 55 illustrates the working principle of the pinch-out method. For example, suppose a model has four regional layers that are each represented (sequentially, Layer 1 to Layer 4, from top to bottom) by a black square. The column labeled “# of Layer after Pinch-out” shows the number of pinch-out layers (shown in blue) allowed for each model layer. For illustration purposes, pinch-outs have been created in two layers, and using these pinch-outs, three different pinch-out types have been created (Figure 55). Type 1 consists of three pinch-outs in model layer 2 (depicted by red lines). Type 2 consists of three pinch-outs in model layer 2 and two pinch-outs in model Layer 3 (depicted by a green line). Type 3 consists of two pinch-outs in model Layer 3. When assigning pinch-outs to the elements, only one type can be assigned to a selected location. It should be noted that, for each model layer, **MINEDW** only allows one value for the number of pinch-outs (e.g., Layer 3 can only have two pinch-outs. It cannot have two pinch-outs in one area and three pinch-outs in another area).

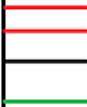
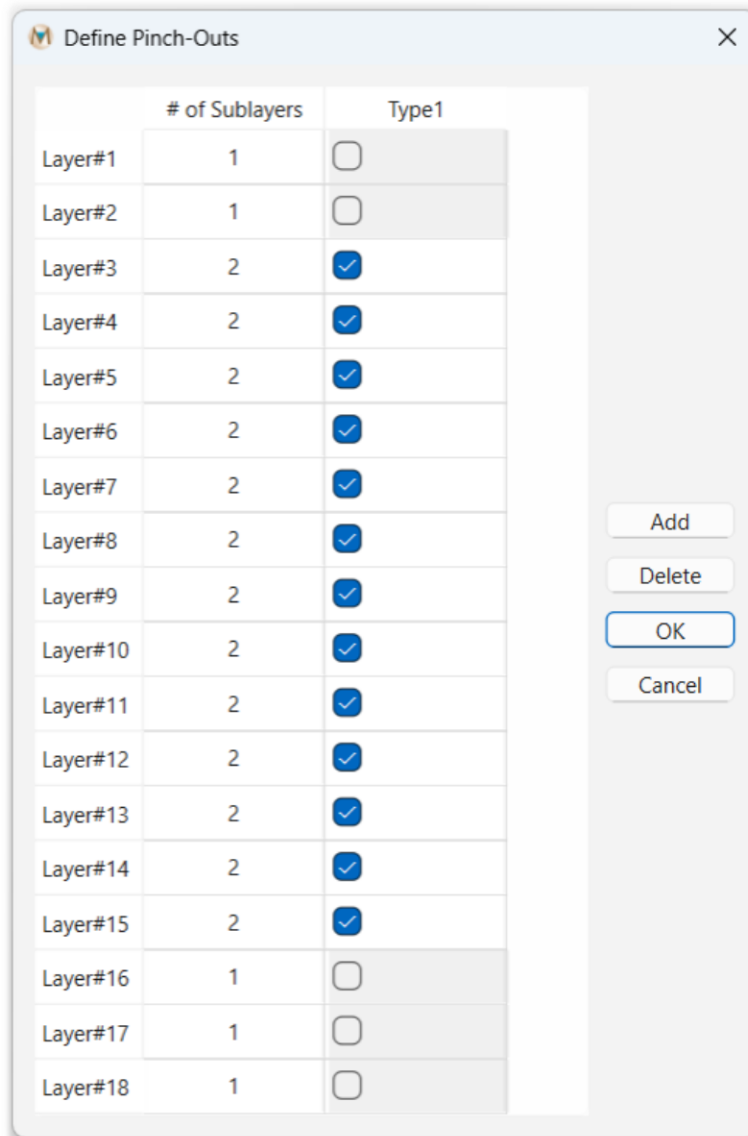
Layer #	# of Layer after Pinch-out	Type 1	Type 2	Type 3
1	1			
2	3	TRUE	TRUE	FALSE
3	2	FALSE	TRUE	TRUE
4	1	FALSE	FALSE	FALSE

Figure 55. Schematic explanation of pinch-outs.

To define pinch-outs, click the “Add” button on the right side of the “Define Pinch-Outs” dialog box to add pinch-out types.

In this tutorial model, there is one type of pinch-out. This pinch-out type (“Type 1”) has two pinch-outs in Layers 3 through 15. Type in the number of pinch-outs in the “# of Sublayers” column and create the pinch-out type by checking the box next to the appropriate layer under the “Type #” field (Figure 56), then click “OK”.



Layer#	# of Sublayers	Type1
Layer#1	1	☐
Layer#2	1	☐
Layer#3	2	☒
Layer#4	2	☒
Layer#5	2	☒
Layer#6	2	☒
Layer#7	2	☒
Layer#8	2	☒
Layer#9	2	☒
Layer#10	2	☒
Layer#11	2	☒
Layer#12	2	☒
Layer#13	2	☒
Layer#14	2	☒
Layer#15	2	☒
Layer#16	1	☐
Layer#17	1	☐
Layer#18	1	☐

Buttons: Add, Delete, OK, Cancel

Figure 56. The “Define Pinch-Outs” dialog box.

To add pinch-outs to a selected area, open the “Build Plot” dialog box by right clicking the plot pane area and selecting “Add Plot Item”. Under the “Element” category, double-click “Pinch-Out”, as shown in Figure 57.

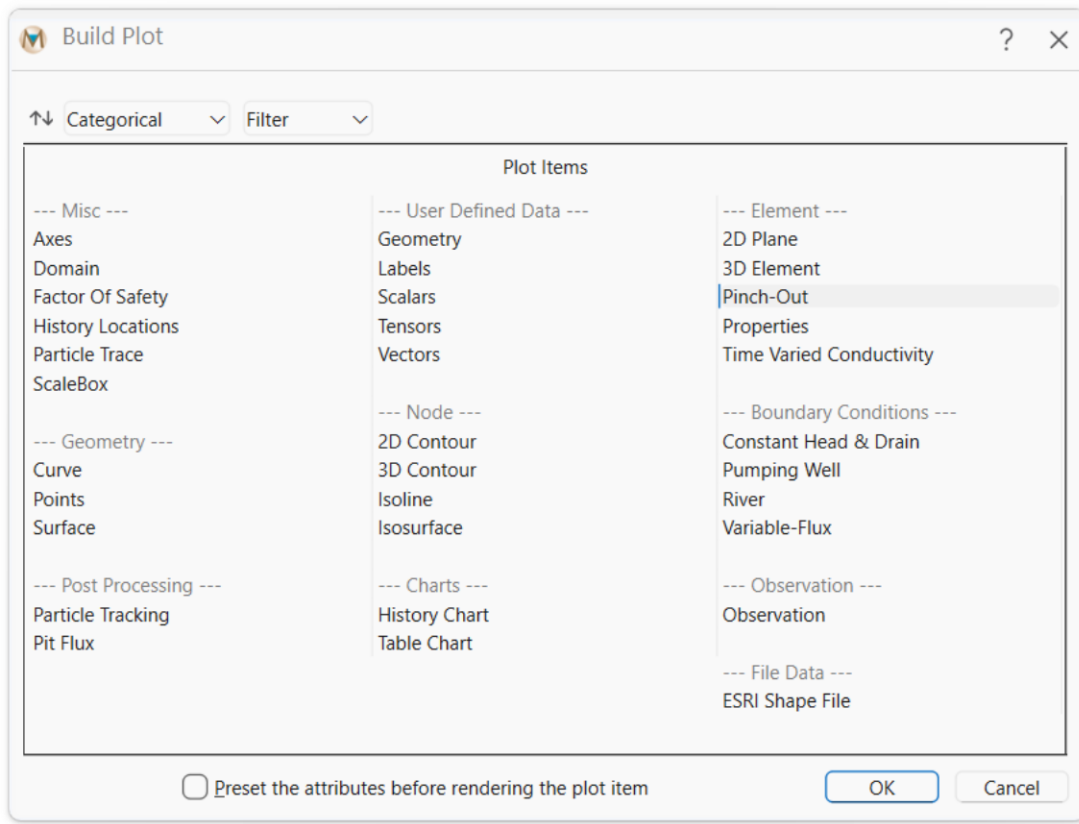


Figure 57. The “Build Plot” pane with the “Pinch-Out” plot item selected.

From the “Build Plot” window, click the “Geometry” plot item. Load the file “PIT-BOUNDARY.dxf”, then, select the nodes that define the elements where the “Type 1” pinch-out will be created by right clicking “PIT-BOUNDARY” on the imported sets, and select “geometry-space DV(1,0,0)”, the software will detect that the geometry is a polyline and will ask to create a extrusion, click “Yes” (Figure 58).

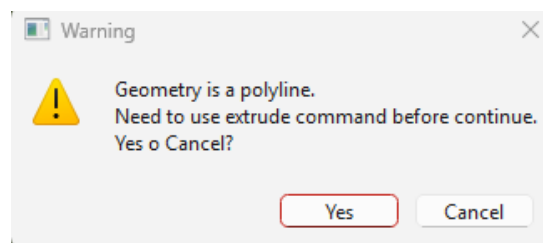


Figure 58. Creating an extrusion from polyline to select pinch-out nodes.

Then, in the “Objects” pane on the left-hand side of the **MINEDW** interface, expand the “named ranges” category within “Model State”, right-click on “PIT-BOUNDARY-oddX” and then click “Apply to Nodes” to create a group. Next, expand the “groups” category, expand “nodes” under it, right-click on the pit boundary element and click “Assign Pinch-out”. The “Select Pinch-Out Type” dialog box appears. Select “Type 1” (Figure 59) and click “OK”. Then, the Pinch-out plot item will be updated showing the new area as Figure 60.

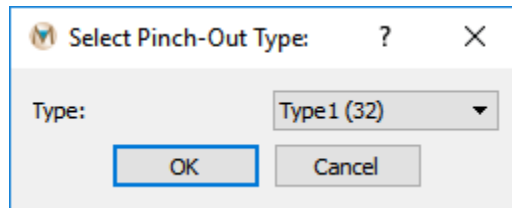


Figure 59. The “Select Pinch-Out Type” dialog box showing “Type 1” selected.

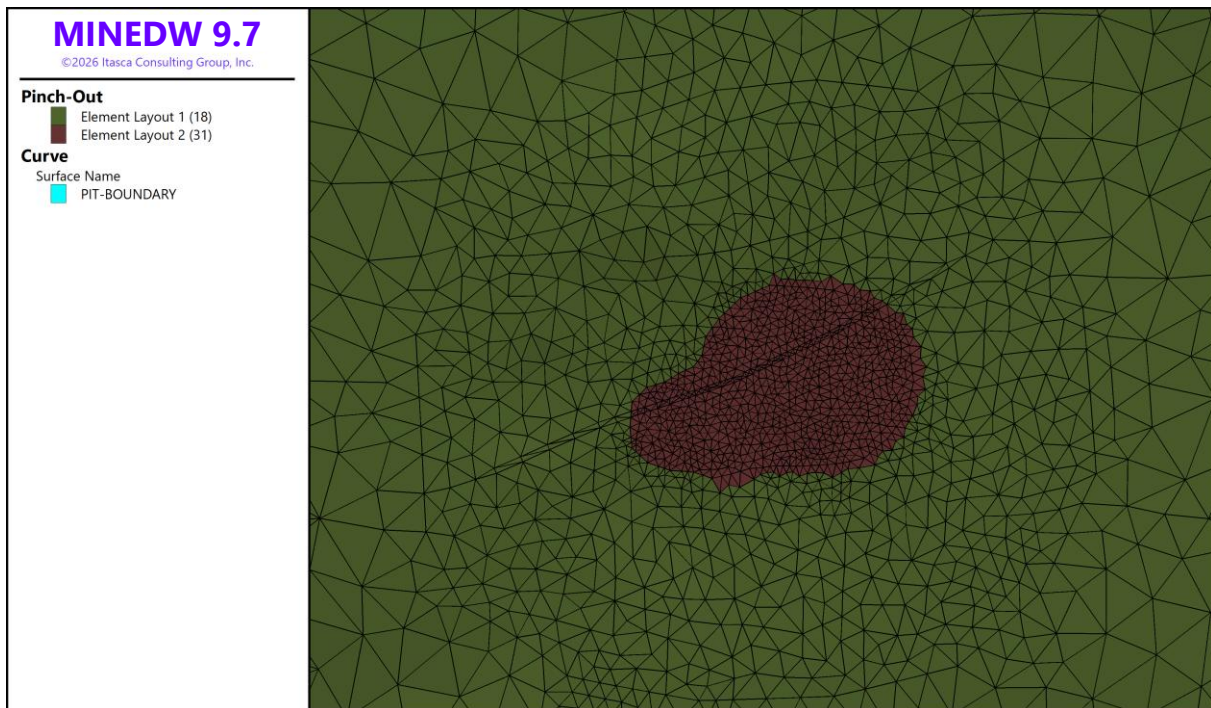


Figure 60. Plan view of Pinch-out selection.

To view the pinch-out in a cross-section view, delete or turn off the “Pinch-Out” plot item by right-clicking on it and choosing “Delete plot item” or “Deactivate plot item”. Next, add a “3D Element” plot item to the View Pane by clicking the arrow next to “Element” in the “Control Panel” pane and double-clicking on “3D Element”. Next, select the “Create Plane...” tool and select two points on either side of the area where the pinch-out was added. The result should look similar to Figure 61.

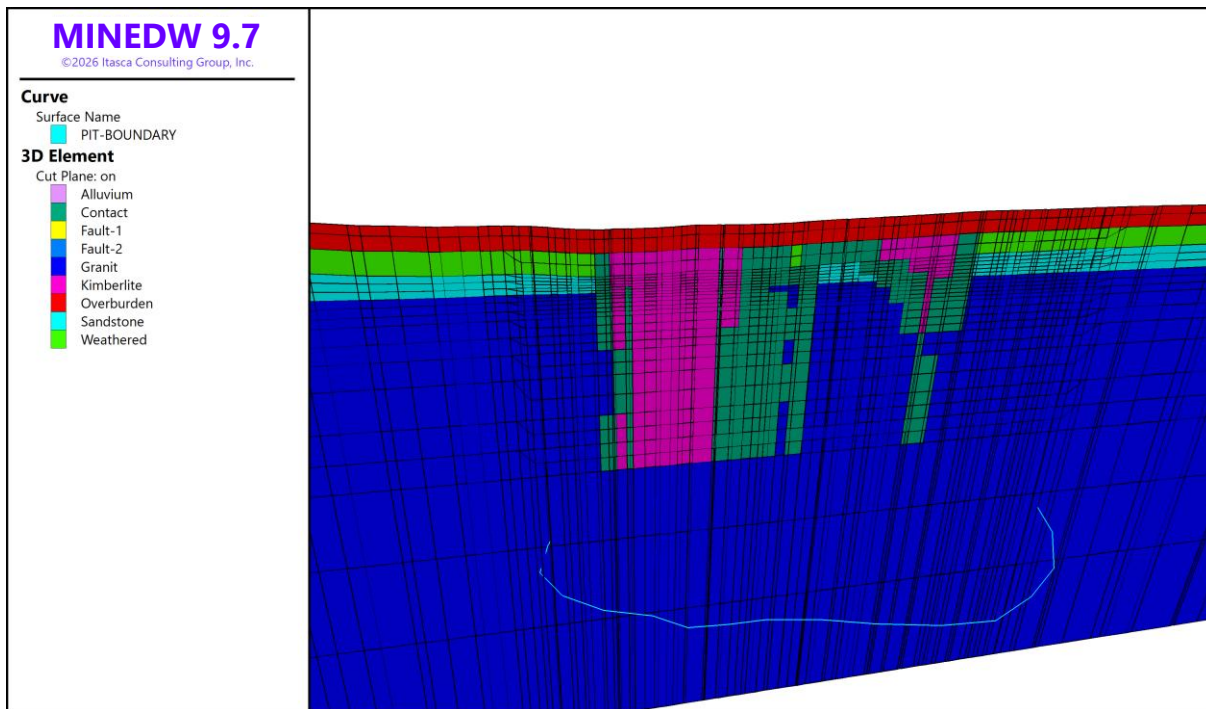


Figure 61. The View Pane with the cross-sectional view with pinch-out Type 1.

2.3.8 Step 9. Defining Initial Heads

To define the initial heads, click "Project" on the Main Menu banner, then select "Initial Heads". The "Set Up Initial Head" dialog box appears. Choose "Constant" with a level of 800 m as shown in Figure 62.

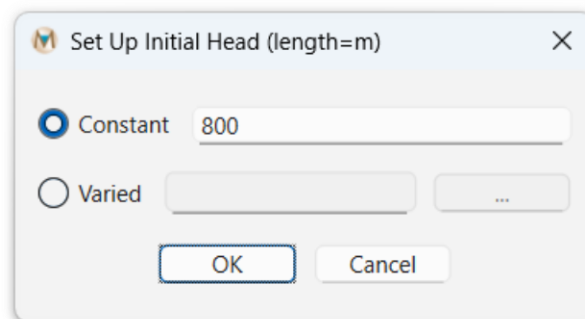


Figure 62. The "Set Up Initial Head" dialog box.

2.3.9 Step 10. Assigning the Boundary Conditions

To assign boundary conditions, open the "Build Plot" dialog box by right clicking the plot pane area and selecting "Add Plot Item". Under the "Node" category, double-click "3D Contour", as shown in Figure 63.

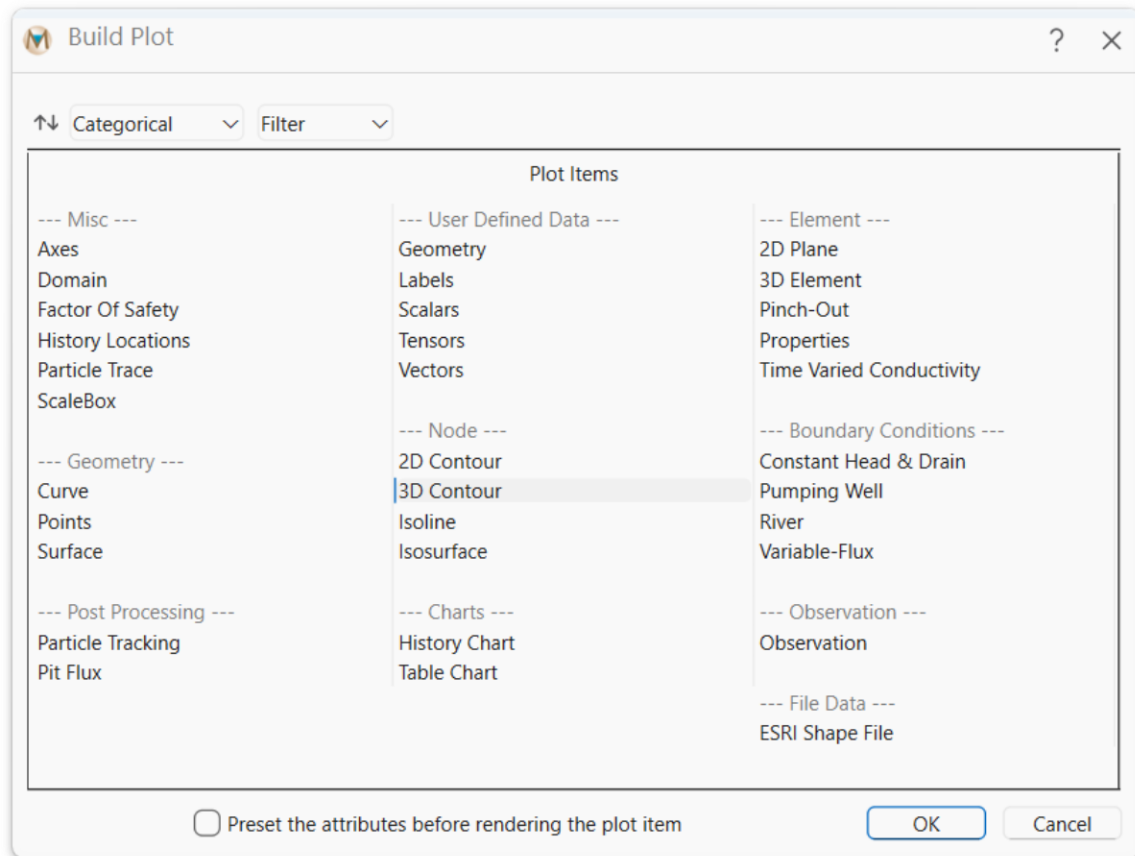


Figure 63. The "Build Plot" pane with the "3D Contour" plot item selected.

In this tutorial, three types of boundary conditions are used: constant-head, river, and no-flow boundaries. Assignment of these boundary conditions is discussed in the following sections.

2.4 Constant-Head Boundary

In order to add constant-head boundary conditions to the model, begin by defining groups for the constant-head boundary conditions. To do this, select the "Constant Head..." menu item under "BCs" on the Main Menu banner. In the "Constant-Head Boundary" dialog box that opens, click on the "Add" button (at the top of the dialog box) to add a blank group (Figure 64). To change the name of the group, click in the drop-down box where the group name appears and change the text (Figure 64). Three constant-head groups will be used for this tutorial simulation, "Pond", "LowLand", and "MtBlock". Create these groups as described above and then click "OK" in the "Constant-Head Boundary" dialog box to close the dialog box and save the changes.

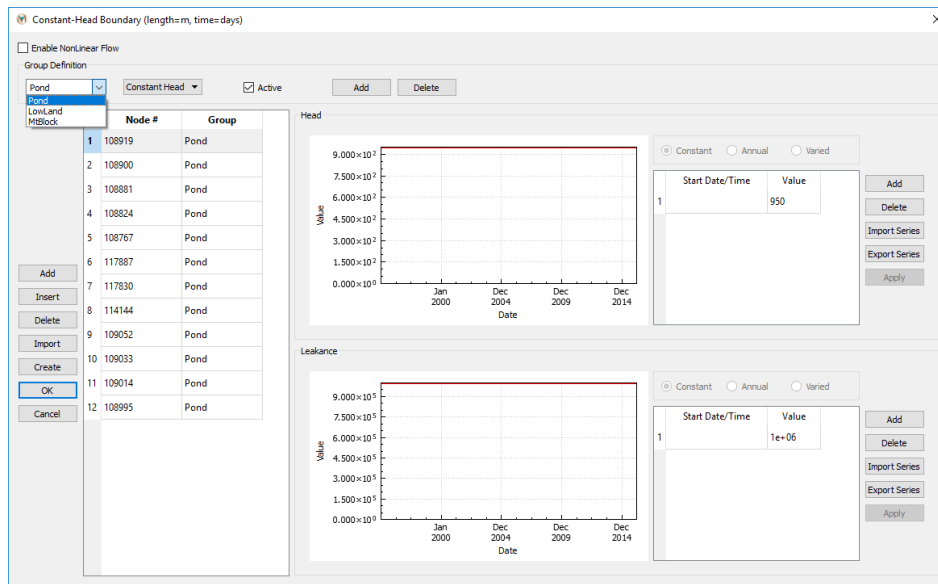


Figure 64. The “Define Group for Constant Heads & Drain Nodes” dialog box.

To add constant-head boundary conditions to the mesh, select the “3D Contour” plot item from the “Build plot” pane. Ensure that the “Layer” attribute under the “Attributes” tab is set to 5, then click the “Select with rectangle” button on the Main Menu banner. Using this tool, select the nodes along the southwest perimeter as shown in Figure 65, and then press [Enter]. In the “Assign Properties for Nodes” dialog box that appears, select “Add to Constant Head Boundary” (Figure 66). Next, select the “MtBlock” group from the drop-down menu that becomes active and then enter the bottommost layer in the box to the right next to “From Current Layer”. This will assign the constant-head boundary from Layer 5 to the bottom of the model domain.

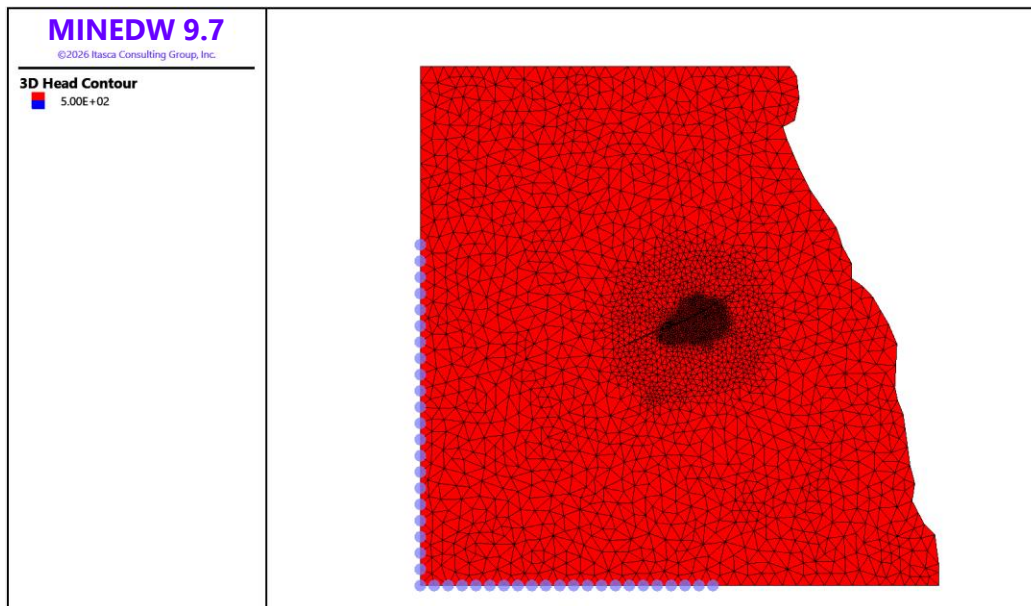


Figure 65. The View Pane with the selected constant head boundary nodes.

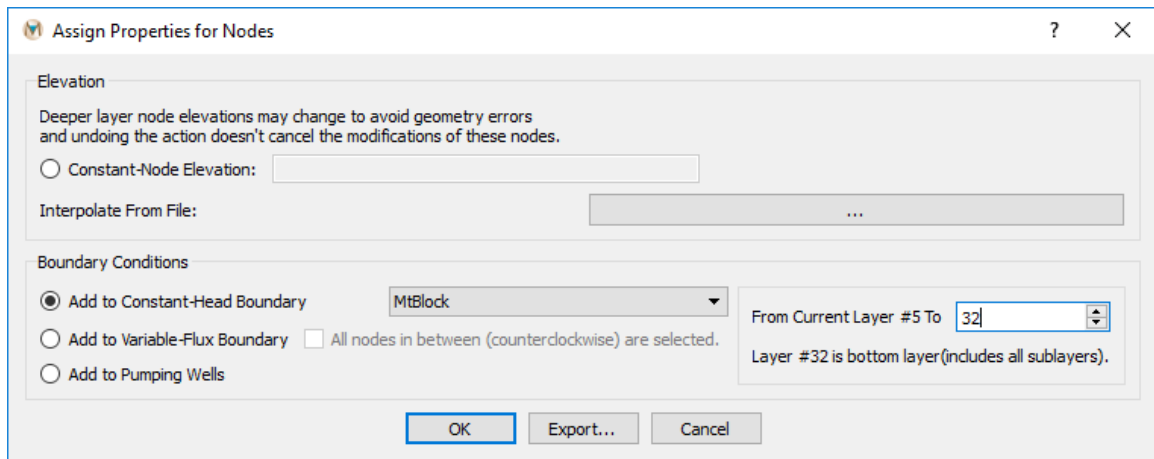


Figure 66. The “Assign Properties for Nodes” dialog box.

Repeat the steps described above for the remaining perimeter nodes that have not been assigned a boundary condition. The constant-head boundary group to be used for the remaining nodes that have not been assigned a boundary condition should be “*LowLand*”. Now that the constant-head nodes have been added to the boundary of the model domain, the constant-head values need to be modified. The default values assigned to the constant-head nodes by **MINEDW** are the elevations of the nodes. Using these default values would result in a downward hydraulic gradient. The constant-head values for nodes beneath the fifth layer should be the constant-head value of the fifth-layer node directly above. To assign these values, the constant-head value for each of the nodes needs to be recorded then copied to each of the nodes below. This can be done using the “*Constant-Head Boundary*” dialog box but would be very tedious. The following describes a method to do this using a provided Excel document.

Begin by executing the “*Create Data Set*” function under the “*Run*” menu on the Main Menu banner. This function will create several text files in a directory that the user selects. Two of the text files, “*chead.dat*” and “*node.fem*”, are needed to update the constant-head values. After running the “*Create Data Set*” function, locate the Excel file named “*UpdateCHead.xlsm*” in the “*INPUT*” directory containing the other input files for this tutorial. Copy the file and paste it in the directory where the “*Create Data Set*” function was executed. Open the Excel document and, if not already active, select the sheet labeled “*Input-chead.dat*”. Click on the button “*Import CHead.dat*” to import the chead file that was just created by **MINEDW**. Using the “*Import Text File*” dialog box that opens, navigate to the directory where the **MINEDW** files were created. The files may not be visible in the “*Import Text File*” dialog box because Excel by default looks for files that have a “.TXT” extension. To view all of the files in the directory through the “*Import Text File*” dialog box, select “*All Files (*.*)*” from the drop-down menu on the lower right-hand corner of the dialog box. Now that all of the files that **MINEDW** created are visible, select the file named “*chead.dat*”, then click “*Open*”. Repeat these steps on the “*Input-node.FEM*” tab, but this time select the file labeled “*node.fem*”. Now that the input data are imported, create the new constant-head file. Select the “*Output-chead.dat*” tab and click “*Run*”. Copy the results from the run function beginning with row 1. Paste the contents into a text editor and save the file as “*newChead.dat*”. Now that the new

thead file has been created, use the “Constant-Head Boundary” dialog box in **MINEDW** to import the file. Click the “Import” button on the left side of the “Constant-Head Boundary” dialog box and navigate to the location where “newChead.dat” was saved, select it, and click “Open”. The new constant-head values will be imported into the model.

The remaining constant-head group that should be added is the “Pond” group. This group is used to simulate an open-water body located some distance to the northeast of the open pit. Using a “3D Contour” plot item, with the “Layer” attribute set to 1 on the “Attributes” tab, select a few nodes somewhere to the northeast of the pit (Figure 67). For this tutorial, it does not matter where the nodes are selected so long as they are not adjacent to the pit. With the nodes selected, press [Enter] to open the “Add Properties for Nodes” dialog box. Click on “Assign to Constant-Head Boundary” and ensure that “Pond” is selected. Leave the “From Current Layer #1 To” as “1” and then click “OK”. The nodes that compose the “Pond” group should all be assigned with the same head value to simulate the water body. To view and edit the assigned constant-head nodes, go to “BCs” on the Main Menu banner and select “Constant Head”.

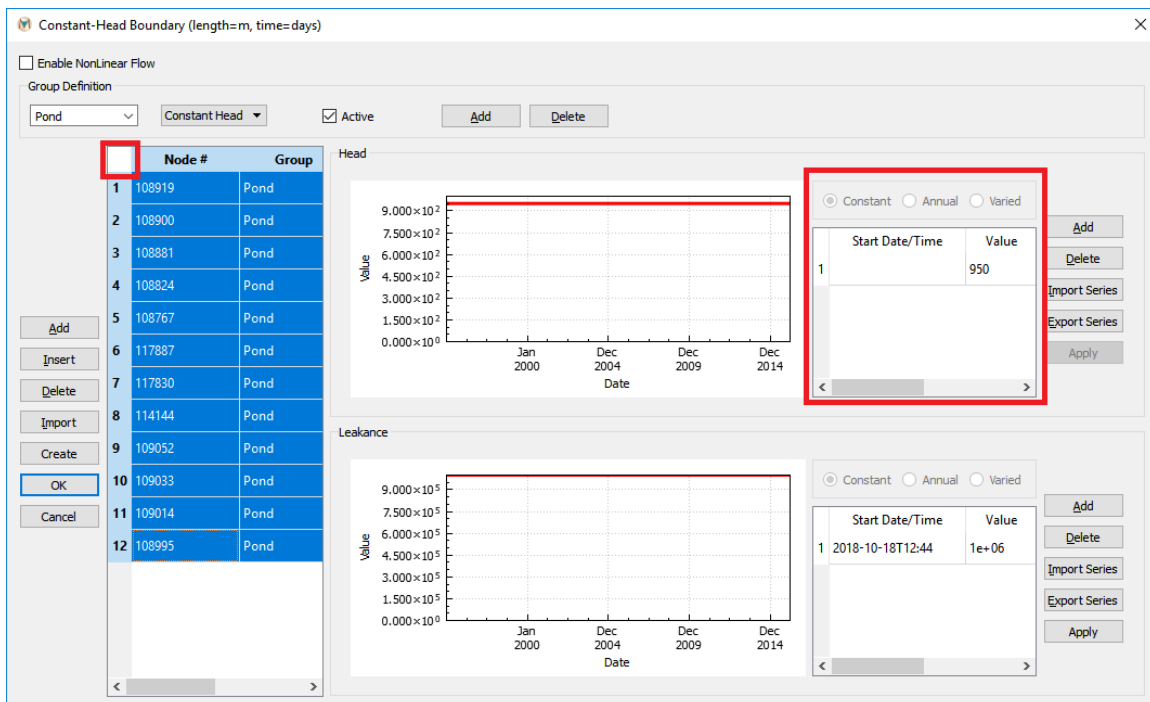


Figure 67. The “Constant-Head Boundary” dialog box.

Ensure the “Pond” group is the currently selected group in the “Group Definition” drop-down box. Select all of the nodes in the “Pond” constant-head group by clicking in the top left square of the constant-head node box, highlighted in red in Figure 67. After selecting the constant-head nodes that will compose the pond feature, change the constant-head value to the desired value, e.g., 950 m. This will update the constant-head value for each of the nodes to 950 m (Figure 65). Ensure that the leakance value is also set to a high value to ensure flow to or from the constant-head nodes is not restricted.

Figure 68 shows the completed constant-head boundary conditions that have been assigned to the model. The "LowLand" and "MtBlock" constant-head values were modified using an Excel document to assign the nodes below the fifth layer a constant-head elevation equal to the elevation of the fifth-layer node directly above. The "Pond" group was used to simulate a body of open water. The constant-head elevation for the "Pond" group was modified in the "Constant-Head Boundary" dialog box.

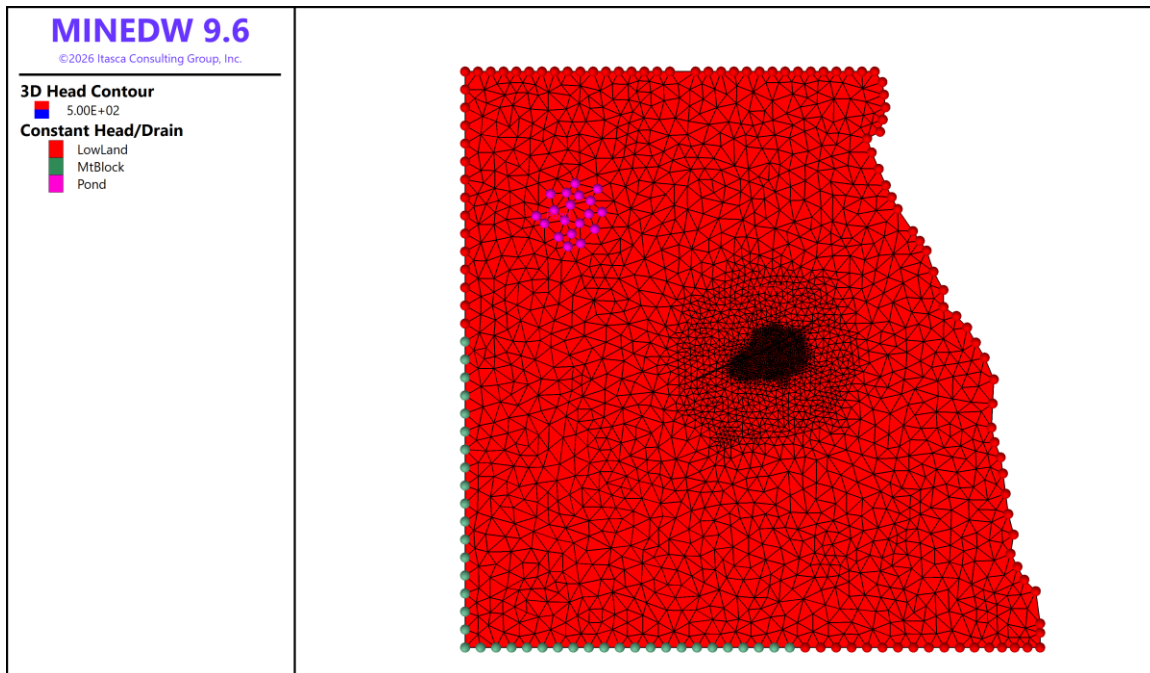


Figure 68. The completed constant-head boundary conditions.

2.5 River Boundary Condition

The "River" boundary condition is used to simulate interactions between a routed river and an aquifer. There are several methods that can be used to add a river boundary condition to a model. This tutorial describes a method using .BLN files to create the river network boundary condition.

To begin, open the "River" dialog box by clicking "BCs" on the Main Menu banner and selecting the "River" menu item. On the "River" tab in the "River" dialog box, click "Add" to add a river segment. Click "Create" in the lower right-hand corner of the "River" tab to open the "Select BLN file" dialog box. Navigate to the directory containing the .BLN files representing the river. These are the same files that were used to create the mesh in Rhino but in individual files for each of the reaches of the river network. Begin by selecting "STREAM_1.BLN" and clicking "Open". **MINEDW** will select the appropriate nodes to simulate the river reach and assign a bed elevation at each of the nodes based on the node elevation. The slope of the riverbed will be variable between nodes depending on the nodes' elevations. Because of this, the "Bed Elev". of some nodes will need to be adjusted to ensure there is a downward slope from the highest to lowest node. **MINEDW** will

produce a warning if the slope at any of the nodes is incorrect. If a constant riverbed slope is desired, click the "Constant Slope" button to calculate a constant slope for the entire river reach. The slope will be calculated based on the highest and lowest node elevations. Change the default values for the "Manning Coefficient", "Width", and "River Bed Thickness" as necessary. The "River" tab is shown in Figure 69 with the completed fields.

The screenshot shows the 'River' dialog box with the 'River' tab selected. The title bar reads 'River (length=m, time=seconds)'. Below the tabs, there is a dropdown menu for 'River1' and buttons for 'Add' and 'Delete'. The main area contains several input fields: 'Manning Coefficient' (0.02), 'Width' (10), 'First (Lowest) Node: Number' (3079), 'Bed Ele.' (805.543), 'River Bed Elevation', 'River Bed Thickness' (2), and 'Elevation Difference' (56.12). There are also buttons for 'Follow Topo' and 'Constant Slope'. A table with 5 columns (Node #, Bed Elev., Bed Conductivity, Output) and 6 rows of data is shown. To the right of the table are buttons for 'Add', 'Insert', 'Delete', and 'Create...'. At the bottom are 'OK', 'Import...', and 'Cancel' buttons.

Node #	Bed Elev.	Bed Conductivity	Output	
1	3060	806	0.002	☐
2	3041	806.5	0.002	☐
3	3022	807	0.002	☐
4	3003	808	0.002	☐
5	2984	809	0.002	☐
6	2965	810.191	0.002	☐

Figure 69. Completed river segment 1 and simulation parameters.

Repeat the steps above for river reaches 2 through 7. Once all of the river reaches have been created, ensure the connections between reaches are properly defined. To do this, click the "Connection" tab in the "River" dialog box. Click the "Refresh" button at the bottom of the tab. **MINEDW** will automatically create the connections between connected river reaches. If the bed elevations of the nodes where the river reaches connect do not match, **MINEDW** will produce a warning indicating where the problem is located. The easiest way to resolve the issue is to adjust the elevation of the upstream reach rather than the downstream reach. Figure 70 shows the "Connection" tab with the correctly defined connections for the tutorial.

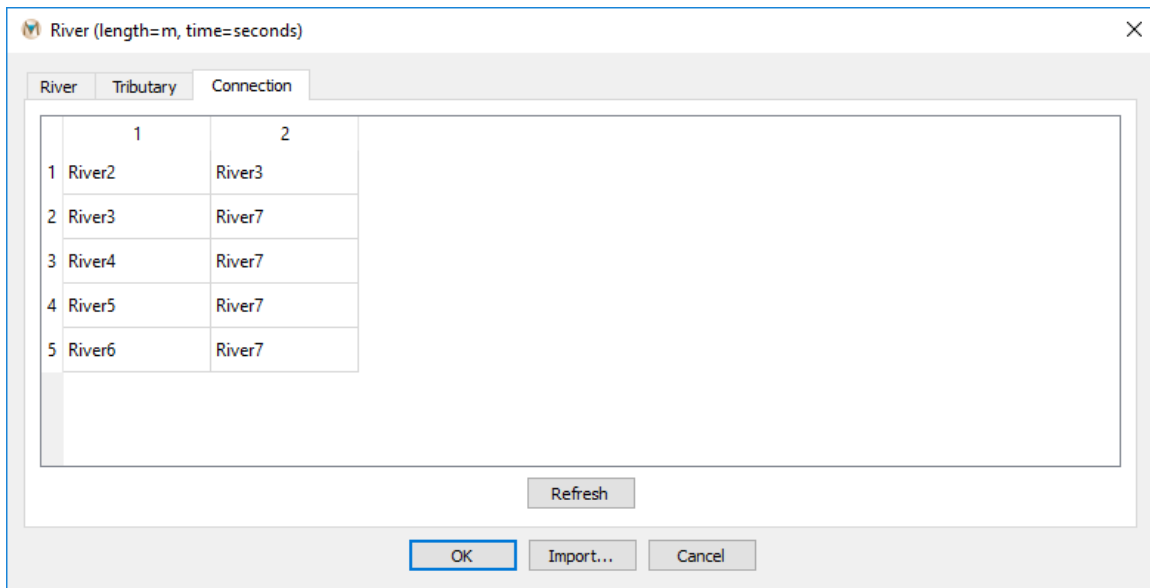


Figure 70. The “Connection” tab with connections between river reaches defined.

The “Tributary” tab is used to define inflows to river reaches that come from outside of the model domain or from some other external source, such as a pipeline that discharges flow to a river. In this tutorial, two tributaries are simulated, one for River 5 and one for River 7. Although the tributary function can except temporally varying inputs for a transient simulation, this tutorial will use constant inflow for both the steady-state and transient simulations. Figure 71 shows the “Tributary” tab with the tributary for River 5 defined. To correctly define a tributary, it must be on one of the nodes that make up a river reach. In this example, the uppermost node of River 5 was chosen as the tributary node. If a node that is not connected to any river is used, **MINEDW** will produce a warning indicating that the tributary is not used. If the tributary is correctly defined, the tributary connection to the river will appear on the “Connection” tab when “Refresh” is clicked.

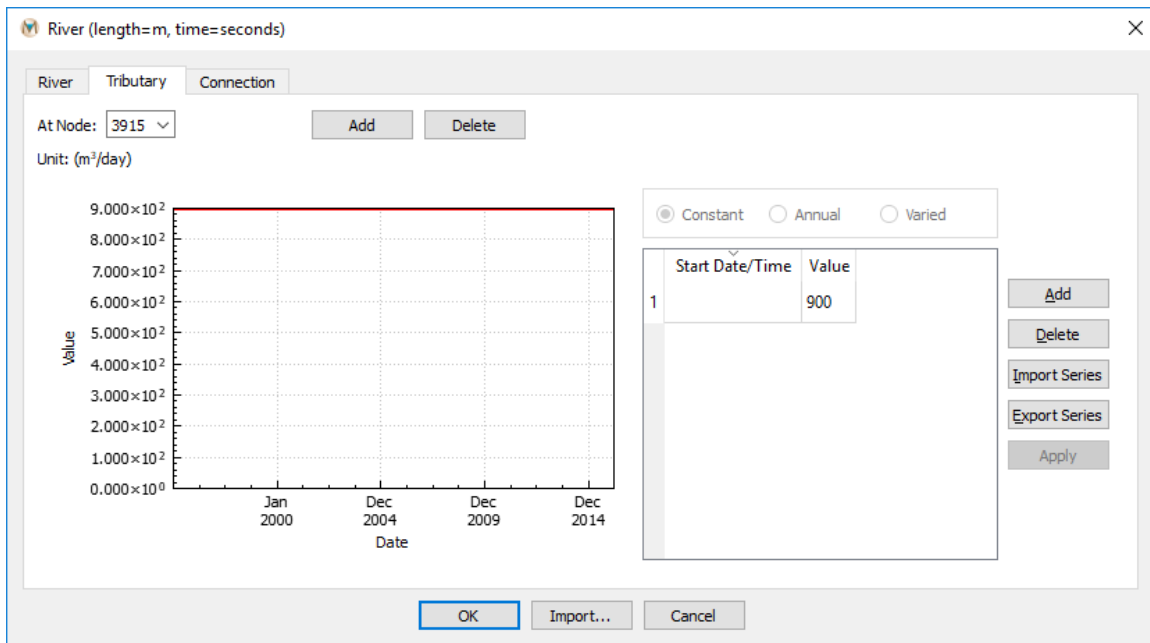


Figure 71. The "Tributary" tab showing the tributary for River 5.

The completed river boundary condition can be displayed in the view pane by selecting the "River" plot item under the "Boundary Conditions" group in the "Control Panel" pane. Figure 72 shows the completed river network and two tributaries that were added to the model.

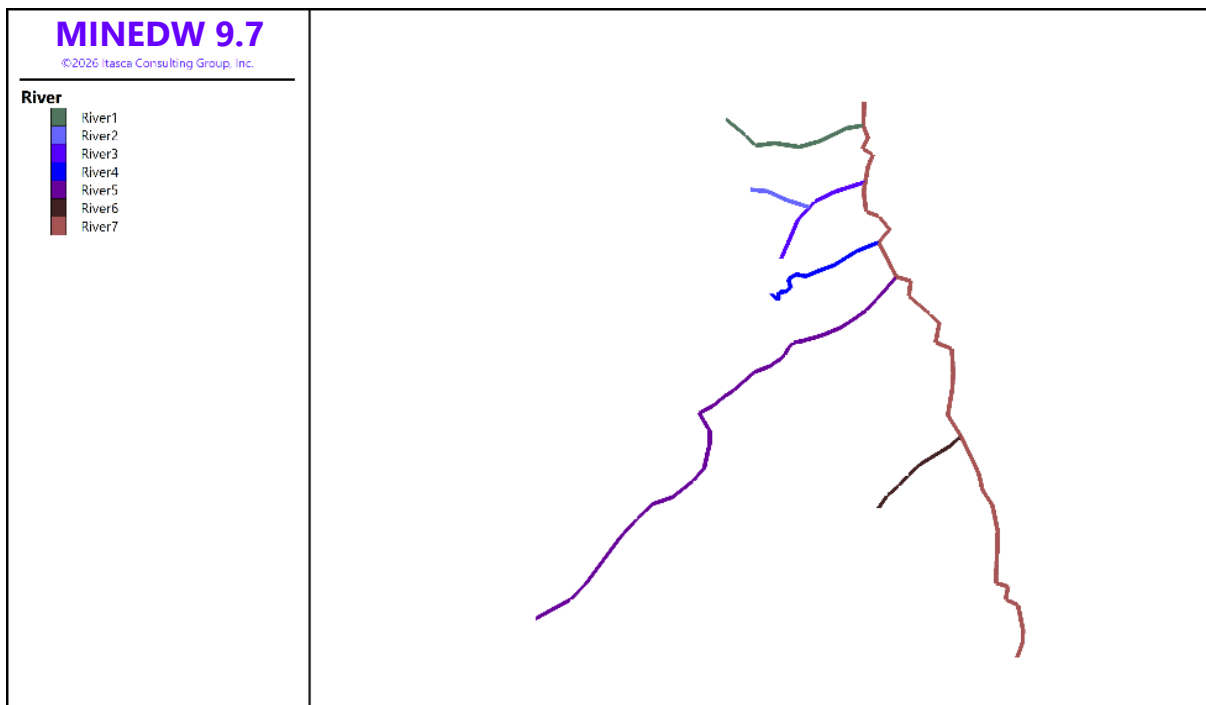


Figure 72. The completed river network.

2.6 No-Flow Boundary

No-flow boundary conditions are assumed by **MINEDW** for any location along the model domain where other boundary conditions have not been assigned. In this tutorial, the bottom of the model is a no-flow boundary condition.

2.6.1 Step 12. Assigning Recharge

In this tutorial, recharge is applied to the first wet layer of the model. The default option in **MINEDW** is to apply recharge to the first wet layer, but this can be changed if needed (Figure 73). To create recharge zones, click "BCs" on the Main Menu banner, then select "Recharge". The "Recharge" dialog box (Figure 71) appears. Click the "Add Zone" button at the bottom of the dialog box to add a recharge zone. Complete the fields for Zone #1 (Figure 73).

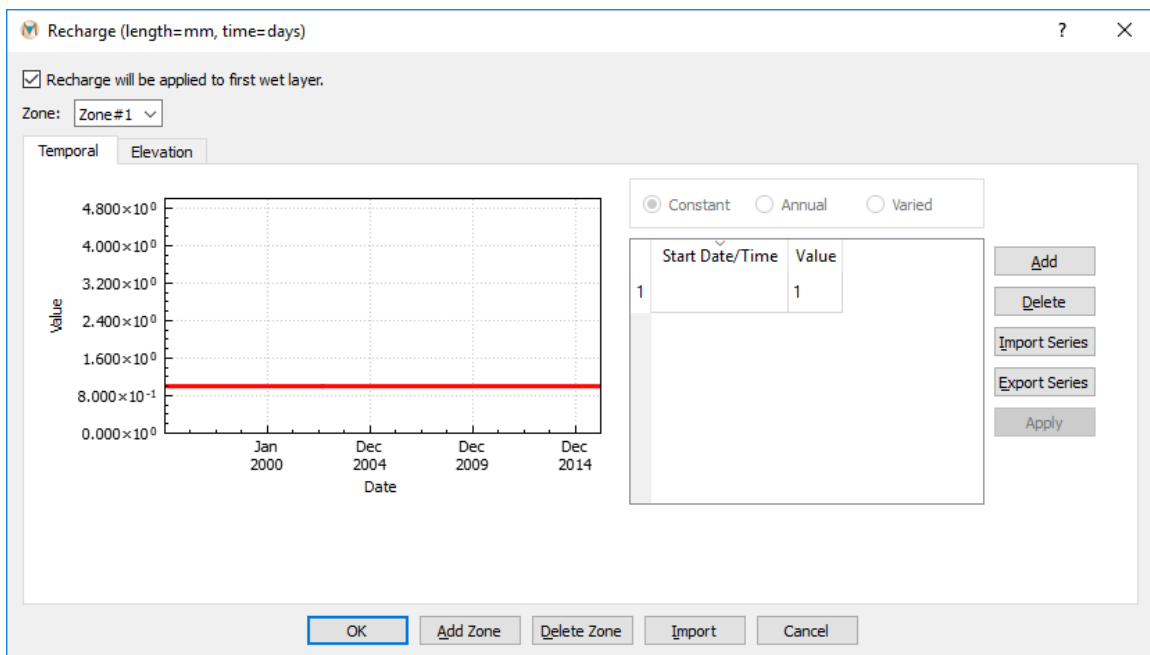


Figure 73. The "Recharge" dialog box showing Zone #1.

Next, click on the "Elevation" tab and define the parameters for the elevation-based recharge equation (Figure 74). This zone will be used to simulate orographically controlled recharge in the model. The parameter values for parameters "A", "B", and "C" can be calculated by fitting the equation (Figure 74) to a precipitation versus elevation data set. For this tutorial, the parameter values for "A", "B", and "C" are provided. The "Factor" that is used is a scaling factor and is explained in further detail in section 7.4.6 of the **MINEDW** manual.

After entering the parameters required to define the first recharge zone (Figure 73 and Figure 74), return to the "Temporal" tab. Click "Add Zone" at the bottom of the "Recharge" dialog box to add a second recharge zone. This zone will be used to simulate recharge in the lower-elevation region of the model domain and will not use the elevation equation for orographically controlled

precipitation. The recharge value that is used for this zone is 0.25 millimeters per day (mm/day) and should be entered on the *Temporal* tab. Once the recharge value for Zone #2 has been entered, click *Apply* on the right side of the *Recharge* dialog box, then click *OK*.

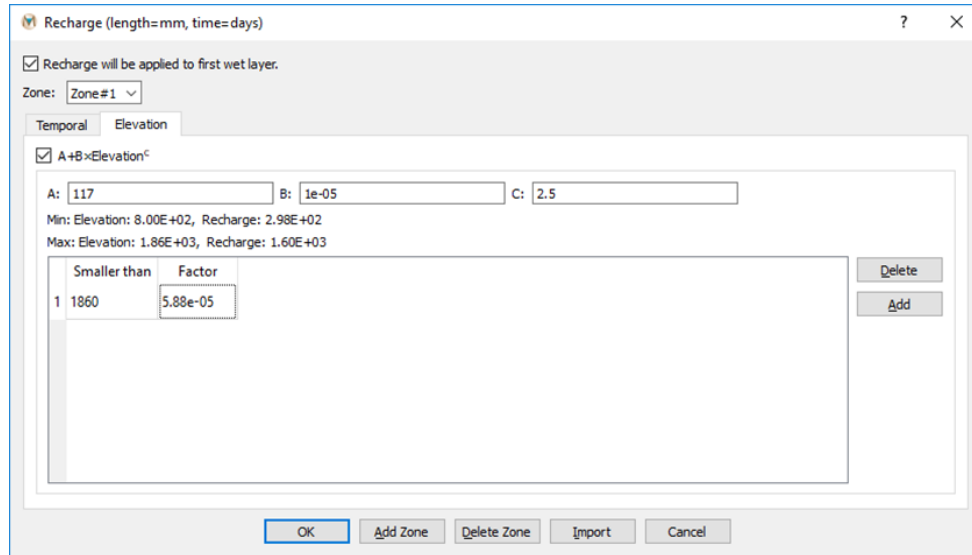


Figure 74. Parameters for orographically controlled recharge.

To assign recharge zones to the model domain, open the *Build Plot* dialog box by right clicking the plot pane area and selecting *Add Plot Item*. Under the *Element* category, double-click *2D Plane*, as shown in Figure 75.

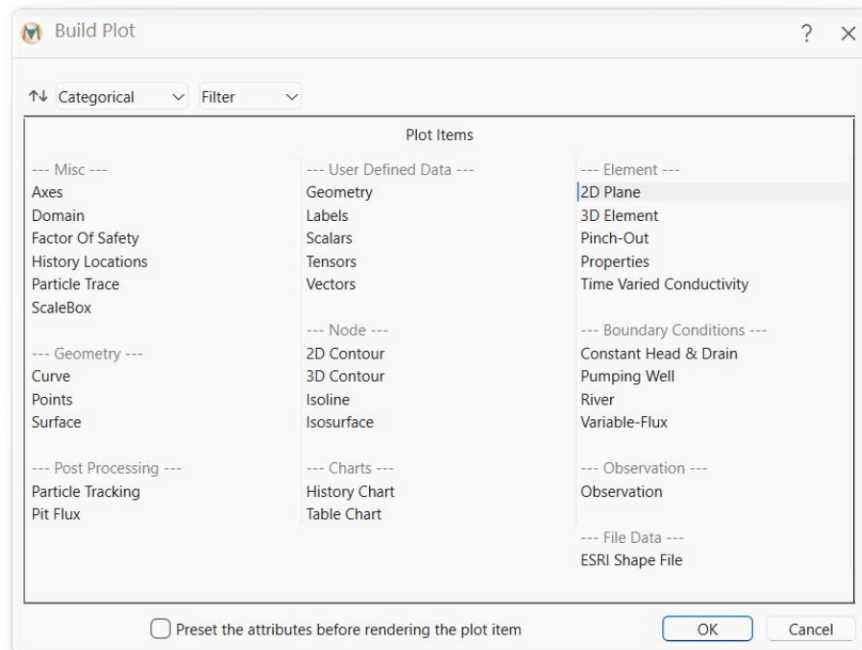


Figure 75. The "Build Plot" pane with the "2D Plane" plot item selected.

While the 2-D plane is selected in the "Control Panel" pane, under the "Attributes" tab, select "Recharge" from the "Color By" drop-down menu (Figure 76).

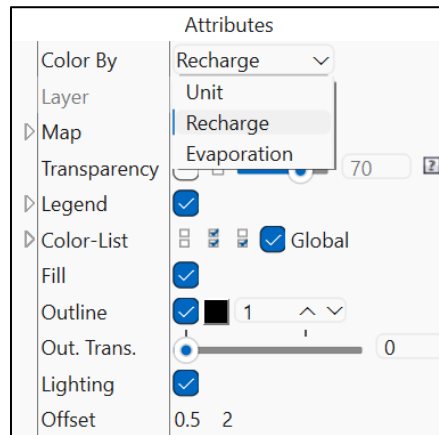


Figure 76. The "Attributes" of the "2D Plane" plot item.

Click the "Select" tool and drag a box around the mountainous region of the model domain to assign Recharge Zone #1 to the model. After selecting the elements, press the [Enter] key, and the "Select Recharge Zone" dialog box appears (Figure 77). Make sure that Zone #1 is selected (Figure 77) and then click "OK".

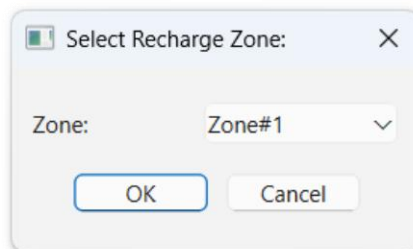


Figure 77. The "Select Recharge Zone" dialog box.

Repeat the steps above, but this time select the elements that form the lowland area of the model domain and select "Zone#2" from the "Select Recharge Zone" dialog box. When assignment of the recharge zones is complete, the result should appear similar to what is shown in Figure 78.

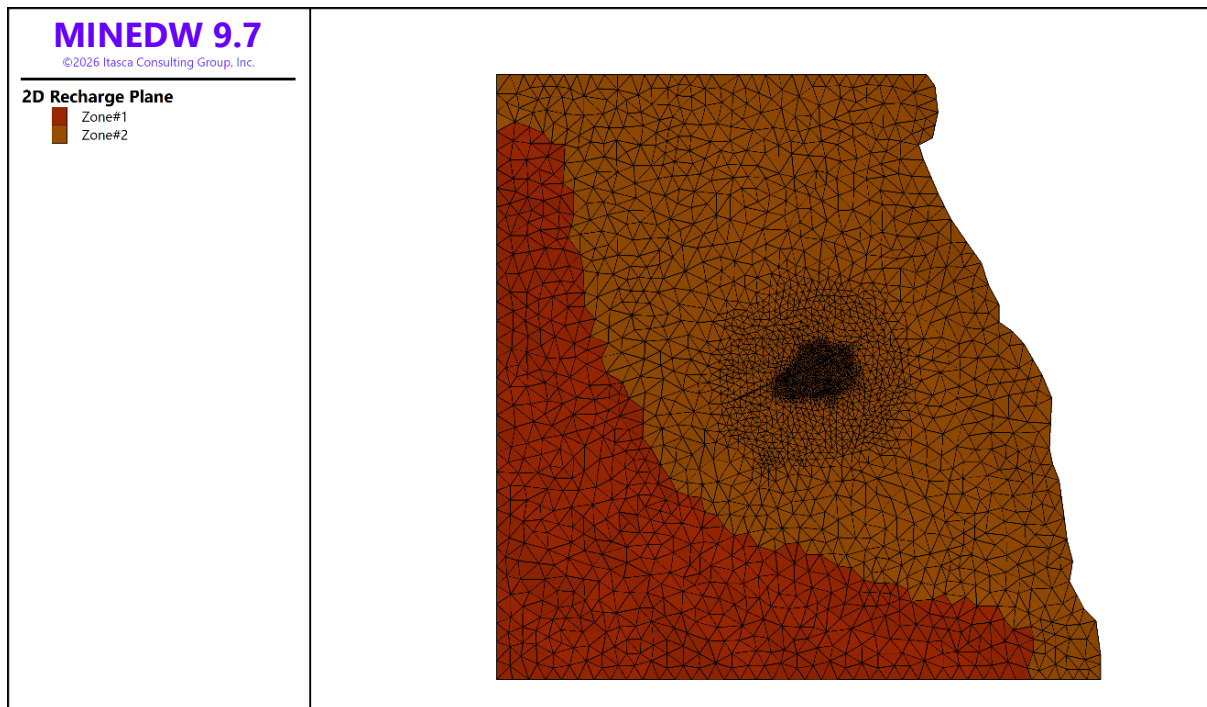


Figure 78. Assignment of recharge zones to the model.

2.6.2 Step 13. Adding Evaporation

To finalize the boundary condition definitions, evaporation will need to be added to the model. Unlike recharge, only one evaporation zone can be created. Evaporation rates can vary over time or be constant for a transient simulation. For the steady-state simulation, a constant-rate evaporation zone will be used.

To create an evaporation zone, select "Evaporation" under the "BCs" menu. The "Define Evaporation Zones" dialog box appears. Enter the evaporation values in the appropriate dialog boxes (Figure 79). For this tutorial, evaporation will be simulated as constant over time. Once the appropriate evaporation values have been entered in the "Define Evaporation Zones" dialog box, click "OK" to finalize the evaporation zone definition.

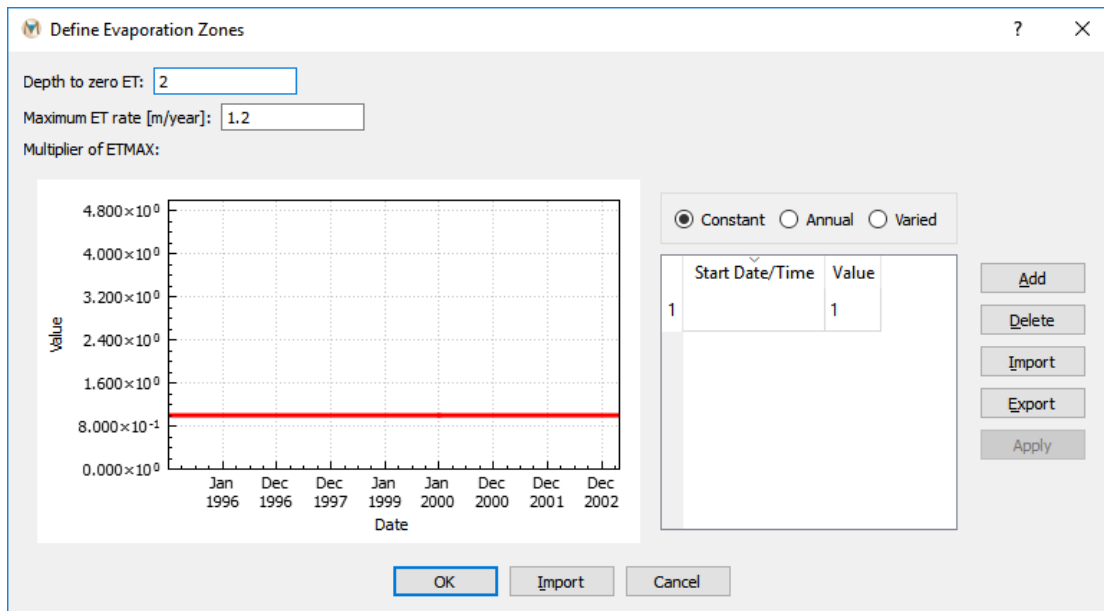


Figure 79. The “Define Evaporation Zones” dialog box.

Now that the evaporation zone has been created, it needs to be assigned to the mesh. To assign evaporation to the mesh, the alluvial zone will be used again, to select elements in the river channel, open the “Build Plot” dialog box by right clicking the plot pane area and selecting “Add Plot Item”. Under the “Geometry” category, double-click “Surface” and then, on the “Attributes” tab, click the “+” next to “Imported Sets”. Browse to “/MESH INPUT/DXF” and open the “STREAM_ALLUVIUM_Polyline.dxf” file.

Next, click on the triangle next to “Imported Sets” to expand it, right click on the alluvium set and then click on “geometry-space DV(1,0,0)” (Figure 49). Then, in the “Objects” pane on the left-hand side of the **MINEDW** interface, expand the “named ranges” category within “Model State”, right-click on “STREAM_ALLUVIUM_Polyline-oddX” and then click “Apply to Elements” to create a group. Next, expand the “groups” category, expand “elements” under it, right-click on the alluvium element and click “Assign Property”. The “Assign Properties for Elements” dialog box appears, select “Evapotranspiration” and then click “OK”. See the results by plotting a 2D plane with color by Evapotranspiration (Figure 80).

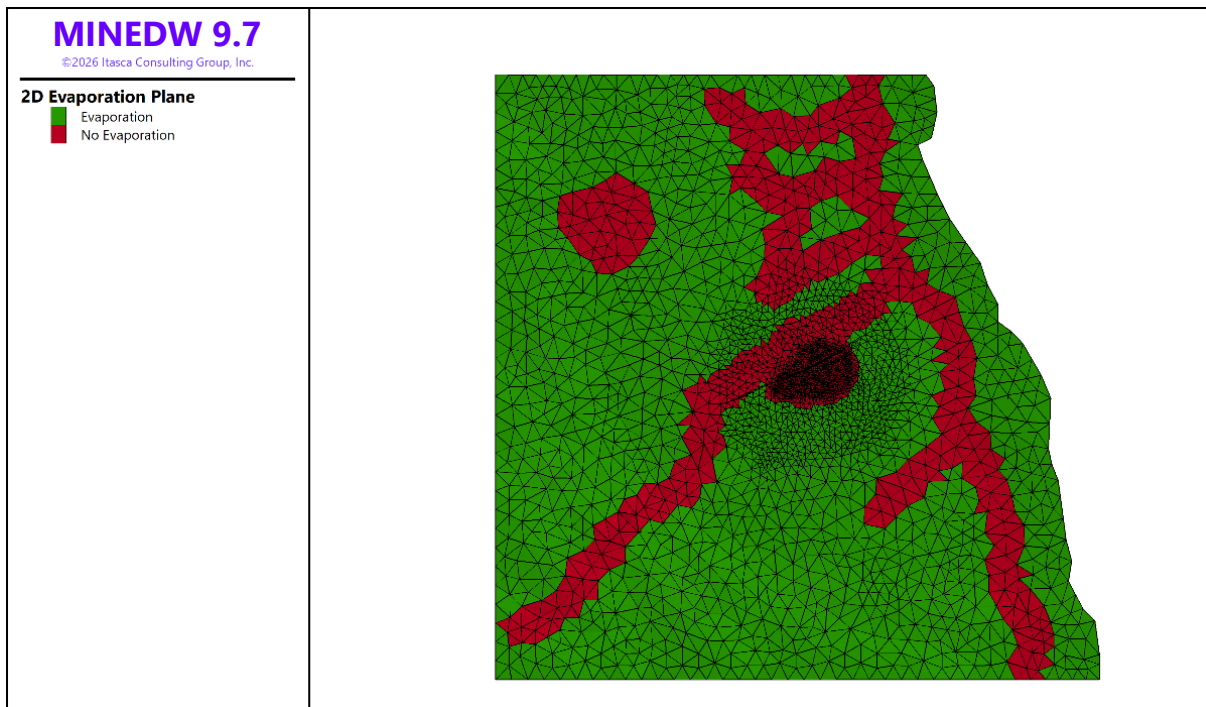


Figure 80. The assigned evaporation zones.

2.6.3 Step 14. Saving the Project

After completing the model setup, click "File" on the Main Menu banner. Select "Save Project as" to save the changes to the project. Save the project (.prj file) and the state (.sav file).

2.7 Running the Model

To run the model, click "Run" on the Main Menu banner, then select "Validate". The "Validate" function checks for errors, such as defining a constant-head level for a constant-head node that is lower than the node itself, and for multiple assignments to a node, such as assigning two constant-head boundary conditions to one node. The function also summarizes the model construction, such as the number of nodes, elements, pumps, etc. After reviewing the model, click "Validate", and **MINEDW** will check for errors in the model. Any errors will be highlighted in the window. If errors are found, review the relevant definitions by following the steps outlined in the previous sections. If no errors are found, click "OK" to complete the validation process.

If the model passes the validation process, then click "Run" again and select "Create Data Set". Select the folder where you want to create your input data set. All input files for **MINEDW** model runs will be created in the chosen directory. NOTE: FILES MUST BE CREATED BEFORE RUNNING THE MODEL. Next, click "Run" on the Main Menu banner and select "Execute" to run the model. A window will appear showing the simulation progress.

2.8 Results

MINEDW has several functions that allow the user to do most of the post-processing they will need.

Under the “*Element*” plot item category on the “*List*” tab, the “*3D Element*” plot item can be used with the “*Isoline*” or “*Isosurface*” plot items under “*Node*” to plot geology and head, pore-pressure, or water-table contours. The “*2D Contour*” and “*3D Contour*” plot items can also be used to plot contours or color floods of head, pore pressure, or the water table.

To visualize the simulation results, click “*Results*” on the Main Menu banner and then choose “*Read Results*”. Select the .PLB file from the folder where **MINEDW** was executed. (Note: The .PLB file is a binary file that contains the calculated head over time.) **MINEDW** will read the entire file.

2.9 Water-Table Contour

To plot the water-table contours, select the “Add Plot Item” option from the right click on the **MINEDW** GUI, find the “*Node*” items, and click “*2D Contour*” (Figure 81).

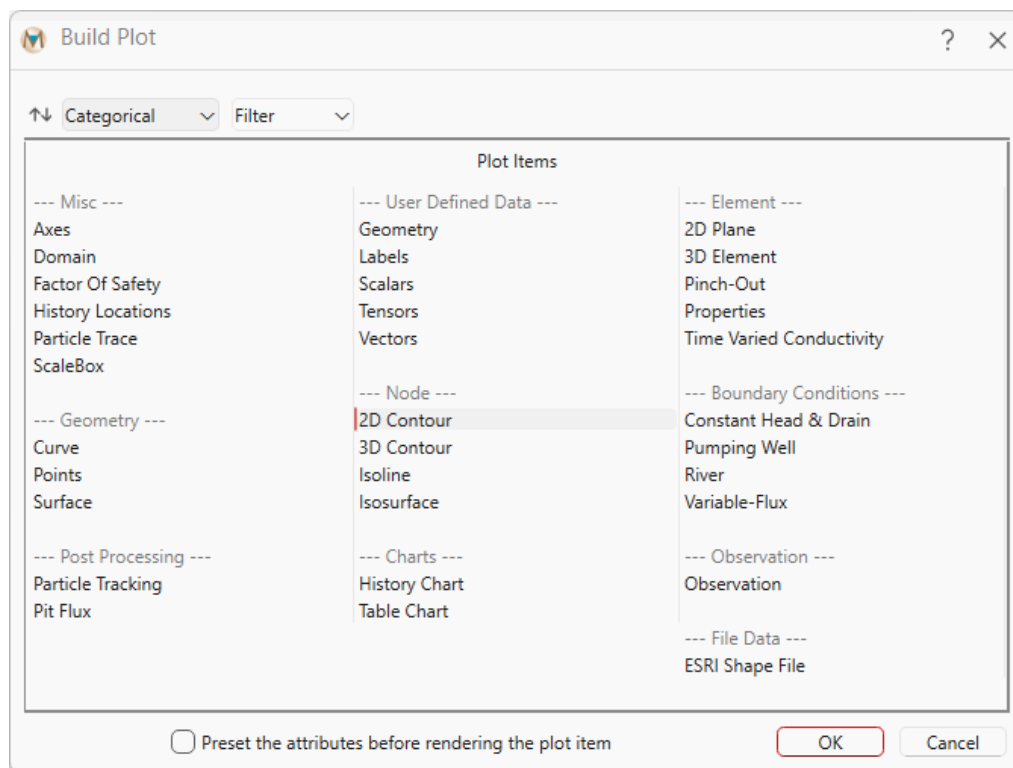


Figure 81. The “*2D Contour*” plot item.

On the “*Attributes*” tab, from the “*Color By*” drop-down menu, select “*Water Table*”. The water-table contour appears in the View Pane. To view the desired time step, enter it in the variable field or move the time step slider bar found on the Main Menu banner. The slider is shown in Figure 82. For the steady-state simulation, view the last time step (time step 100 in this tutorial).

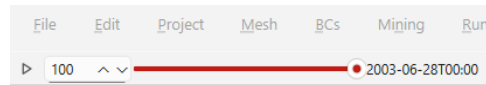


Figure 82. Time step slider.

To export the water-table contour as a bitmap, right click on the plot, go to "Export" and choose "Bitmap". The exported bitmap is provided in Figure 83.

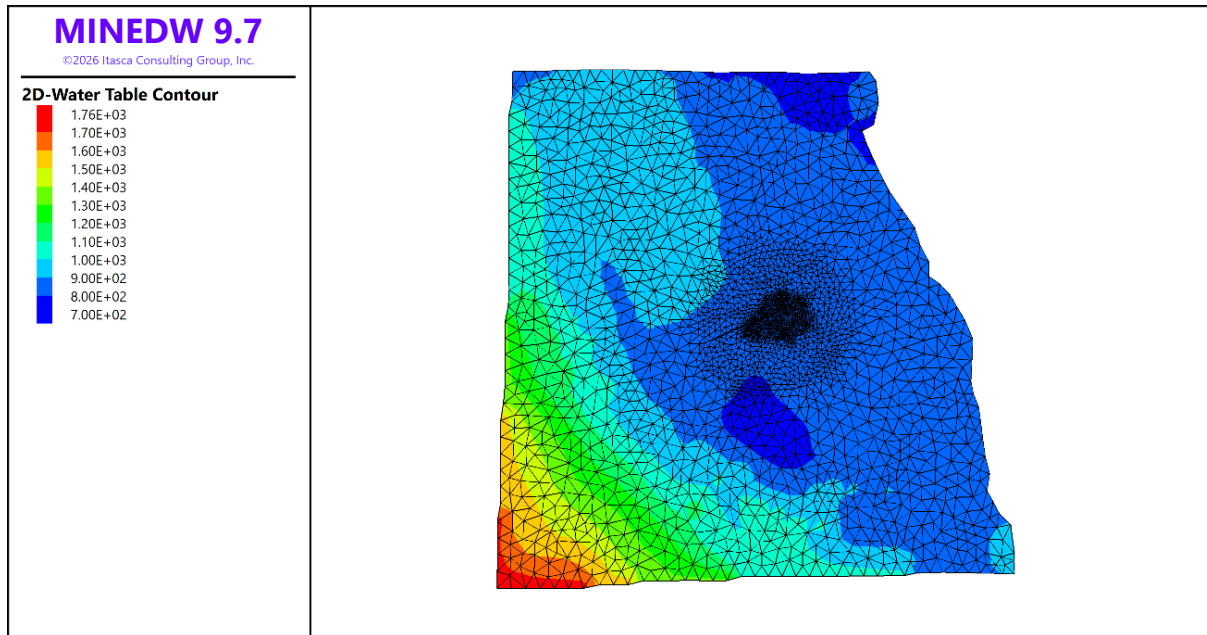


Figure 83. Water-table contours from steady-state simulations.

2.10 2-D or 3-D Head Contour

To create a 2-D or 3-D contour plot of the head distribution, "2D Contour" or "3D Contour" in the "Build Plot" window. A 2-D contour is ideal for creating a 2-D color flood image of head distribution within any of the model layers, while a 3-D contour item is best for creating 3-D images of head distributions or cross sections. Figure 84 is a cross section through the model displaying head. To recreate this image, choose "3D Contour" and then choose "Head" in the "Color By" drop-down menu on the "Attributes" tab. Next, activate the "Cutting Tool" located under "3D Head Countour". Enter the following values for the "Plane" on the "Attribute" tab: Origin = (461965, 9.07041e+06, 250), Normal = (-0.665092, 0.746761, 0) Dip/DD = (90, -41.6894). Then click the "Snap View" button. Note: Cross sections can be created using the "Create Plane with two selected points on the plot" tool, but the location of the cross section is not as precise. Select the "3D Contour" in the "Plot Items" pane and then expand the "Cutting Tool" item on the "Attributes" tab. The cross section can easily be modified to display pore "Pressures" or "Head Difference" by selecting the appropriate item from the drop-down menu next to "Color By" on the "Attributes" tab for the "3D Contour" plot item.

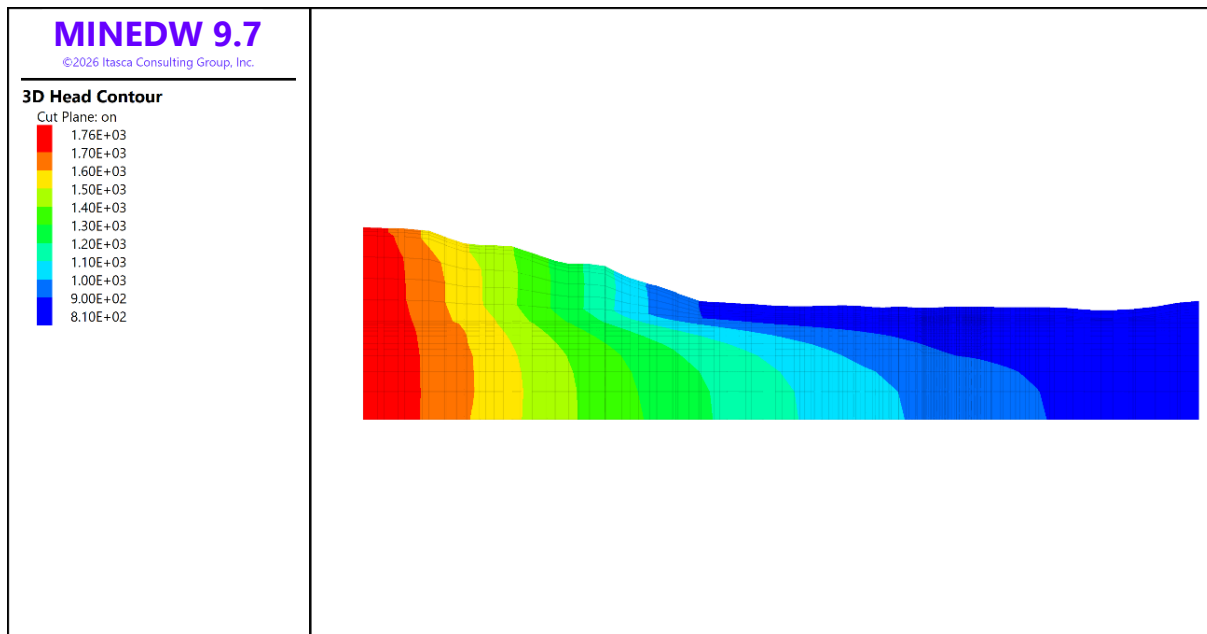


Figure 84. A 3-D cross section of head exported from MINEDW.

2.11 Geologic Cross Section and Water Table

MINEDW can create cross sections showing the geology represented in the model overlaid by the groundwater table. To construct a figure similar to the one shown in Figure 85, select "3D Element" (found under "Element") from the "Build Plot" window. Next, add a "Isosurface" (under "Node") and select "Pressure" from the "Color By" drop-down menu on the "Attributes" tab. Colors for both the "3D Element" and "Isosurface" plot items can be modified by expanding the "Contour" attribute and choosing a different color ramp.

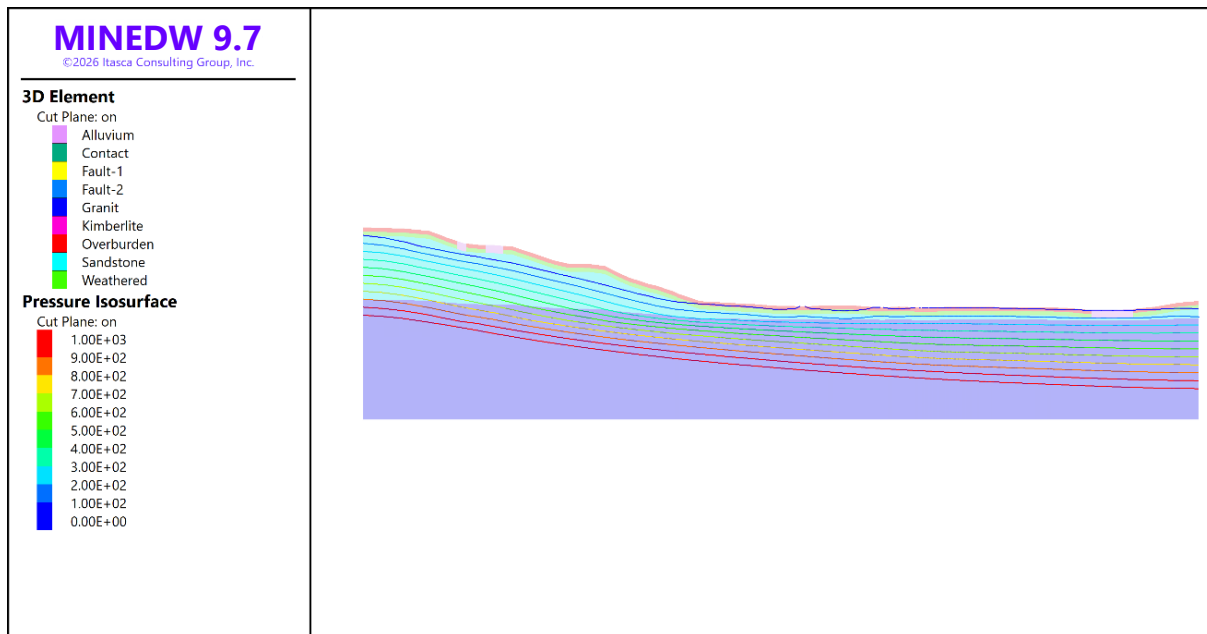


Figure 85. A cross section of the geology and pore pressure isosurface.

3.0 TUTORIAL 2. OPEN PIT, ZONE OF RELAXATION, PUMPING WELLS, AND PIT LAKE

3.1 Problem Description

Tutorial 2 demonstrates the setup of a transient-state simulation including an open pit, a ZOR, pumping wells, and a pit lake.

Tutorial 2 uses the same model as Tutorial 1. The simulation results from the steady-state simulation (Tutorial 1) are used as the initial conditions for Tutorial 2. In this part of the document, the procedure for creating a mining plan, ZOR, pit lake, and variable-flux boundaries will be discussed step by step.

3.1.1 Step 1. Opening the Project File

The model created in Tutorial 1 is used for this problem. To open the project file created in the first tutorial, click "File" on the Main Menu banner, then select "Open/Import". The "Open" dialog box appears, as shown in Figure 86.

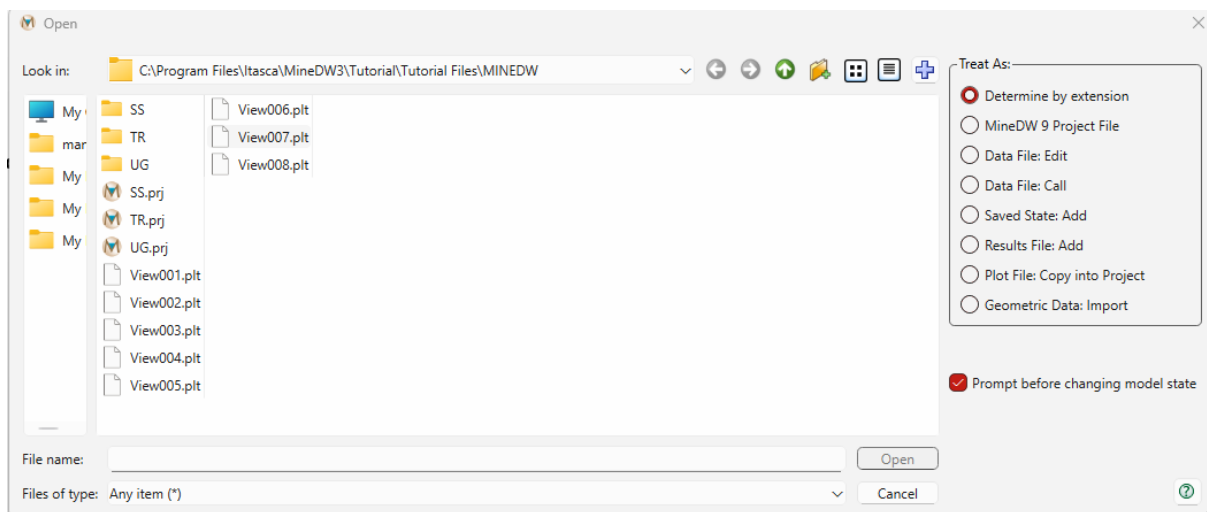
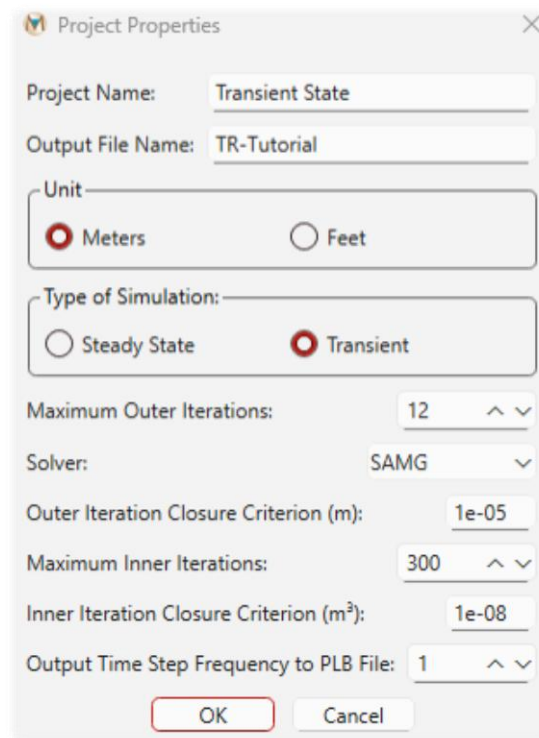


Figure 86. The "Load a MINEDW Project File" dialog box.

Select the project (.PRJ) file and click "Open". Click "File" on the Main Menu banner, then select "Save Project As" to save the project as a different .PRJ file for the transient simulation.

3.1.2 Step 2. Defining Project Properties

To define a project property, click "Project" and then select "Project Property". The "Project Properties" dialog box shown in Figure 87 appears. Make sure that all the fields shown in Figure 87 contain the values shown in Figure 87. For transient simulations, select the "Transient" option and then click "OK".



Project Properties

Project Name:

Output File Name:

Unit

☒ Meters ☐ Feet

Type of Simulation:

☐ Steady State ☒ Transient

Maximum Outer Iterations:

Solver:

Outer Iteration Closure Criterion (m):

Maximum Inner Iterations:

Inner Iteration Closure Criterion (m³):

Output Time Step Frequency to PLB File:

Figure 87. The “Project Properties” dialog box.

3.1.3 Step 3. Defining Model Time Steps

To define the time steps for the project, click “Project” on the Main Menu banner, then select “Time Steps”. The “Set Up Time Step” dialog box (Figure 88) appears. Set the “Maximum Time Steps” to 300. Set the first time step to 1 January 1995. The time-step length will be one month. For transient simulations, the multiplication factor (as discussed in Tutorial 1) is 1 and the time-step length is defined in the “Set Up Time Step” dialog box. Ensure that the “# of time steps for this simulation” slider is set to 300 and that the “Start at Time Step” combo box is set to 1 and click “OK”.

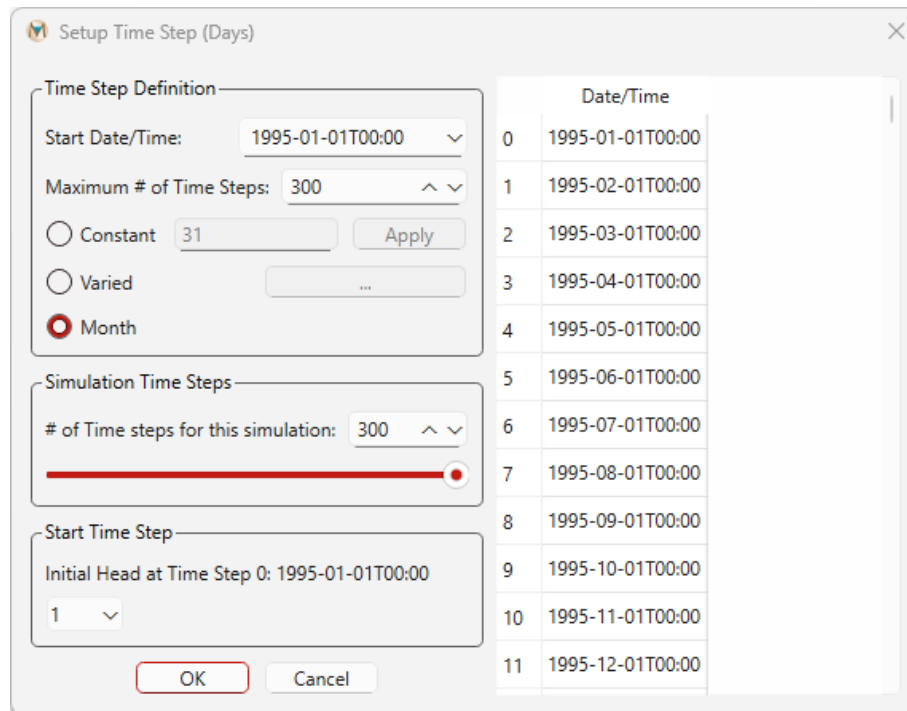


Figure 88. The “Set Up Time Step” dialog box.

3.1.4 Step 4. Defining Initial Heads

To define initial heads, click “Project” on the Main Menu banner, then select “Initial Heads”. The “Set Up Initial Head” dialog box appears. Choose the “Varied” option and select the water levels file (“*.MDL”) created during Tutorial 1 by clicking on the “...” button as shown in Figure 89, then click “OK”.

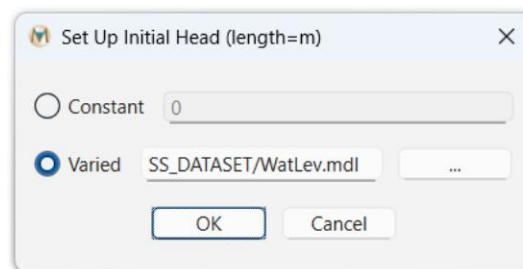


Figure 89. The “Set Up Initial Head” dialog box.

3.1.5 Step 5. Creating a Mining Plan for an Open Pit

The easiest method to create a mine plan in **MINEDW** is to obtain .DXF files of the pit topography at various points in time (e.g., at the end of every year of mining). Each .DXF file will need to be converted to a data file containing X, Y, Z coordinates of the pit surface for interpolation of the mine plan to a mining input file for **MINEDW**. In this example, there are 11 years of mining. .DXF

files of the planned pit shape at the end of each year of mining are found in the "Tutorial Files/OPENPIT" file directory.

Open the "Build Plot" dialog box by right clicking the plot pane area and selecting "Add Plot Item". Under the "Geometry" category, double-click "Surface". Then, to select the .DXF file for Year 1 go to the "Attributes" tab, click the "+" next to "Imported Sets", shown in the red box in Figure 90, browse to the file, select it, and click "Open".

Next, click on the triangle next to "Imported Sets" to expand it, right click on the "Year1" set and then click on "Export Points to File", shown in a green box in Figure 90. Be sure to use the same file name as the .DXF file to name the "*.DAT" file; otherwise, the next step will not work. Repeat these steps for the next 10 years of mining.

For any mining plan that is created in **MINEDW**, a pit footprint is required (e.g., the ultimate pit boundary) to define the nodes that will be used for interpolation. This boundary needs to be defined in a .BLN file format (Appendix B). This boundary is always the first record in any mining schedule when creating an open-pit mining plan. This file defines the perimeter of the open pit and does not contain ground-surface elevation information because all ground-surface elevations are assumed to be the top of the mesh.

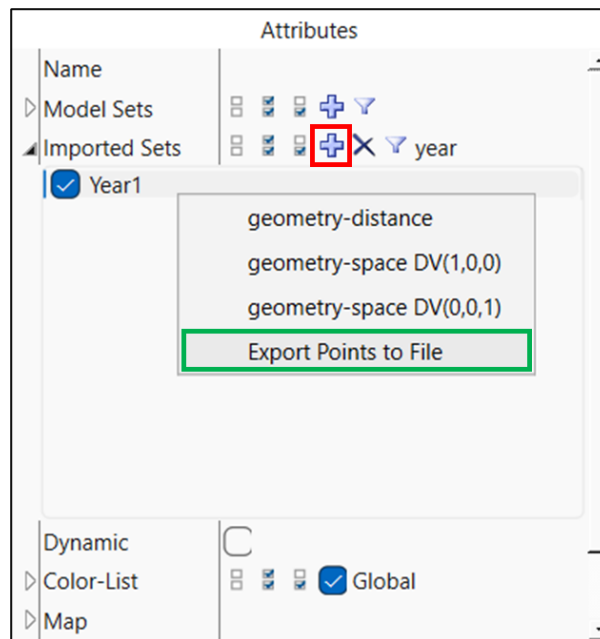


Figure 90. **Reading a .DXF file and saving as a data file.**

To create the mining plan, on the Main Menu banner, click "Mining", then select "Create Mining Plan". The "Create Mining Plan" dialog box (Figure 91) appears. The date, file name, and interpolation method can be entered manually by typing in the information under "Date", "File/Elevation", and "Interpolated With", respectively. Another option is to select an input file known as a Mining Schedule file (Appendix B) containing this information. Itasca has created a Mining Schedule file to illustrate this process. To use it, select the "Open" button and select the file "INPUT/OPENPIT/MINE_SCHEDULE.DAT". This file contains the dates and file names, as shown in Figure 91. After importing the mine plan file information, enter a file name for the new mining plan file in the "Created Mining Plan File Name:" box (for example, "MINE_FILE.DAT") at the bottom of the "Create Mining Plan" dialog box (Figure 91). In order to create a mining file, all input mining files must be located in the same directory (in this example, the files "MINE_SCHEDULE.DAT", "ULTIMATE.BLN", "YEAR1.DAT", "YEAR2.DAT", etc. were moved from "INPUT/OPENPIT" to the main simulation directory). Next, click the "Options..." button to open the "Mining Plan Interpolation" dialog box. Uncheck the box next to "Not interpolated if no data found in Range". This will exclude locations that do not have associated elevation data. Click "OK" on the "Mining Plan Interpolation" dialog box to accept the changes and then click "OK" on the "Create Mining Plan" dialog box to create the mining plan file for the **MINEDW** model.

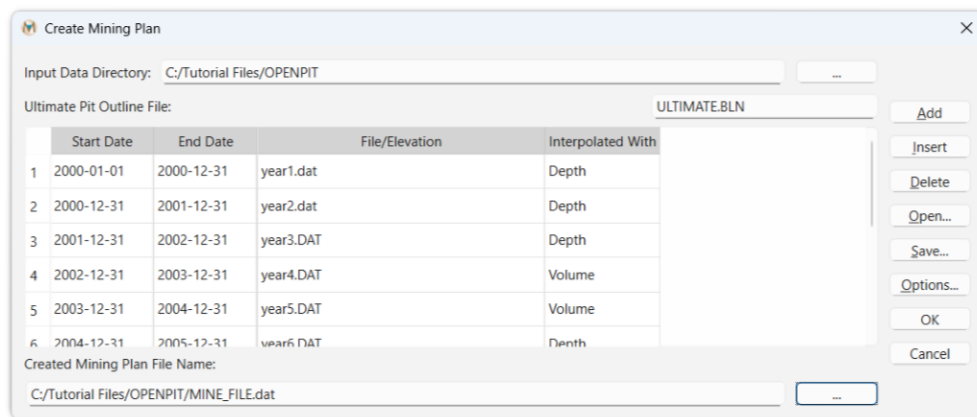


Figure 91. The "Create Mining Plan" dialog box.

To assign a mining file to the model, click "Mining" on the Main Menu banner and select "Open Pit". The "Open Pit" dialog box appears (Figure 92).

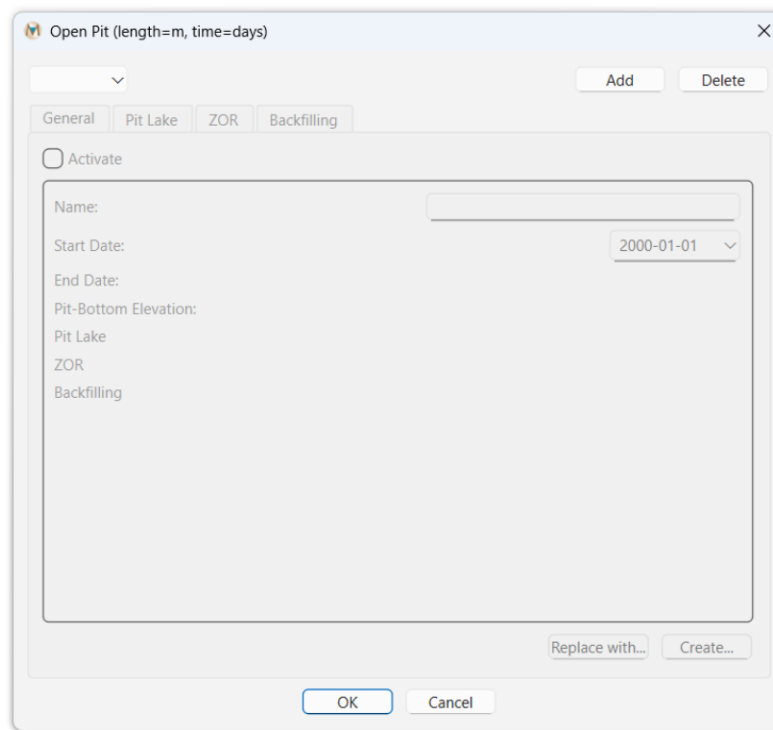


Figure 92. The “Open Pit” dialog box.

Click “Add” and browse to the newly created mining file, select it, and click “Open”. After importing the mining file, the “General” tab in the “Open Pit” dialog box will show the “Start Date” and “End Date” for mining as well as the “Pit-Bottom Elevation”(Figure 93). This tab will also indicate whether a “Pit Lake”, “ZOR”, or “Backfilling” will be simulated. Immediately after importing the mining file, these options should indicate that none will be simulated. Steps 5 through 7 describe how to add these features to a model simulation after creating and importing a mine plan. No additional open-pit mines will be simulated in this tutorial, but if required, multiple open-pit mines can be added to the model for simulation by repeating the steps described above for creating and importing mining files.

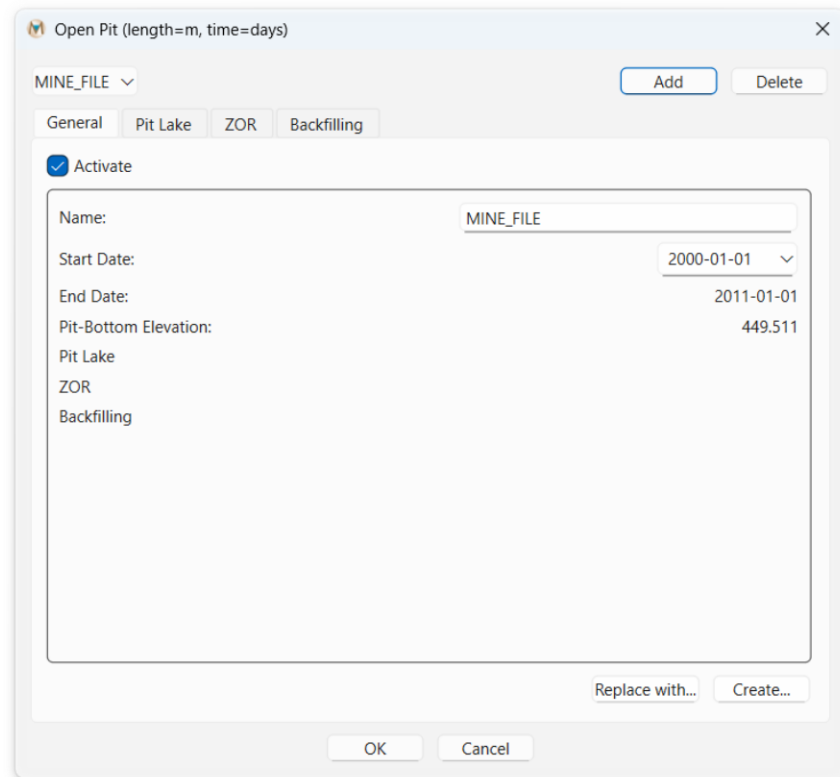
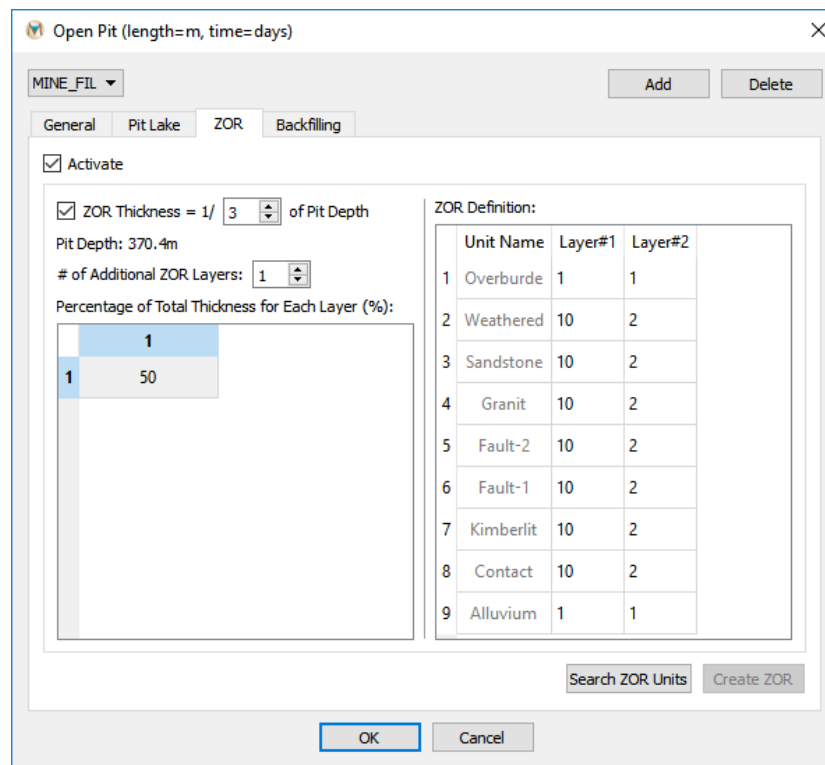


Figure 93. The “Open Pit” dialog box showing the “General” tab.

3.1.6 Step 6. Creating a Zone of Relaxation for an Open Pit

In **MINEDW**, time-variant hydraulic conductivity (K) can be simulated for situations such as a ZOR around open-pit excavations, backfilling operations, longwall and room-and-pillar coal mining, and freeze-thaw conditions.

To create a ZOR for this open-pit simulation, select “Open Pit...” from the “Mining” drop-down menu on the Main Menu banner. The “Open Pit” dialog box (Figure 93) appears. Select the “ZOR” tab to create the ZOR. **MINEDW** offers two options for defining the thickness of the ZOR: (1) As a ratio of the pit depth or (2) by a user-defined thickness (both options are discussed in detail in Section 7.6.4 of the **MINEDW** manual). In this tutorial, the ZOR will be defined as a ratio of the pit depth and divided into two layers. To create the ZOR, check the box next to “ZOR thickness = 1/” and enter “3” as the denominator in the box. In the box next to “# of Additional ZOR Layers”, ensure that the value is “1”. **MINEDW** will add a default layer to the one layer defined by the user for a total of two layers within the ZOR. Only the thickness of the first ZOR layer needs to be defined because the remainder of the ZOR will automatically be assigned to the default ZOR layer created by **MINEDW**. The thickness of each layer is defined as a percentage of the total ZOR thickness. In this tutorial, both ZOR layers will be equal in thickness (i.e., each layer will be 50% of the ZOR) (see Figure 94).



Open Pit (length=m, time=days)

MINE_FIL Add Delete

General Pit Lake ZOR Backfilling

☒ Activate

☒ ZOR Thickness = 1/ 3 of Pit Depth

Pit Depth: 370.4m

of Additional ZOR Layers: 1

Percentage of Total Thickness for Each Layer (%):

	1
1	50

ZOR Definition:

Unit Name	Layer#1	Layer#2
1 Overburde	1	1
2 Weathered	10	2
3 Sandstone	10	2
4 Granit	10	2
5 Fault-2	10	2
6 Fault-1	10	2
7 Kimberlit	10	2
8 Contact	10	2
9 Alluvium	1	1

Search ZOR Units Create ZOR

OK Cancel

Figure 94. The "Open Pit" dialog box showing the "ZOR" tab.

Next, click "Search ZOR Unit", located at the bottom of the dialog box. The geological zones that lie within the defined ZOR will appear (Figure 94). The numbers next to each geological unit represent the factor by which the K in each ZOR layer will change. The default is 1, or no change in K values. Numbers greater than 1 represent an increase in the K values (i.e., to double the K values, enter "2"), while numbers less than 1 represent a decrease in the K values.

In this tutorial, the K values are 10 and 2 times greater than the original K values. In the "ZOR Definition" dialog box, enter "10" next to the units in "Layer#1" except for the "Overburden" and "Alluvium" zones, as they will be removed prior to mining (leave as 1). See Figure 94 for more details about assigning values to the second layer of the ZOR. After entering the appropriate data in the "ZOR" tab of the "Open Pit" dialog box, click "Create" and then check the box next to "Activate" (this option only becomes active after the ZOR has been correctly defined). Finally, click "OK" at the bottom of the dialog box to apply the changes to the model and exit the "Open Pit" dialog box.

Once a ZOR has been created, it can be visualized by selecting in the "Build Plot" window the "Time Varied Conductivity" plot item under the "Element" group (Figure 95). The ZOR that was created is shown in Figure 96.

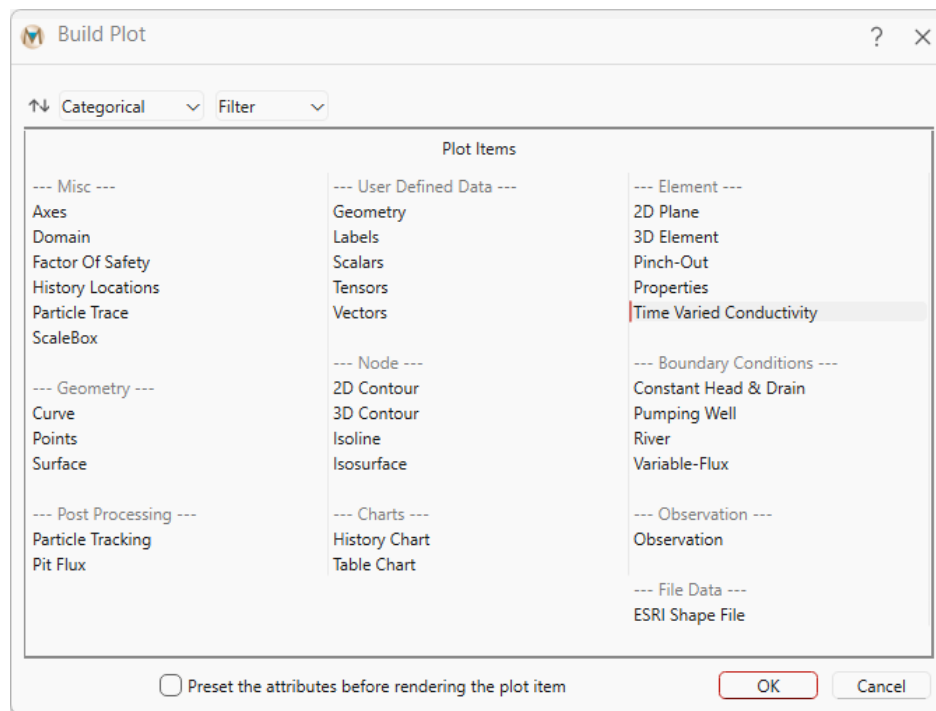


Figure 95. The “Time Varied Conductivity” plot item.

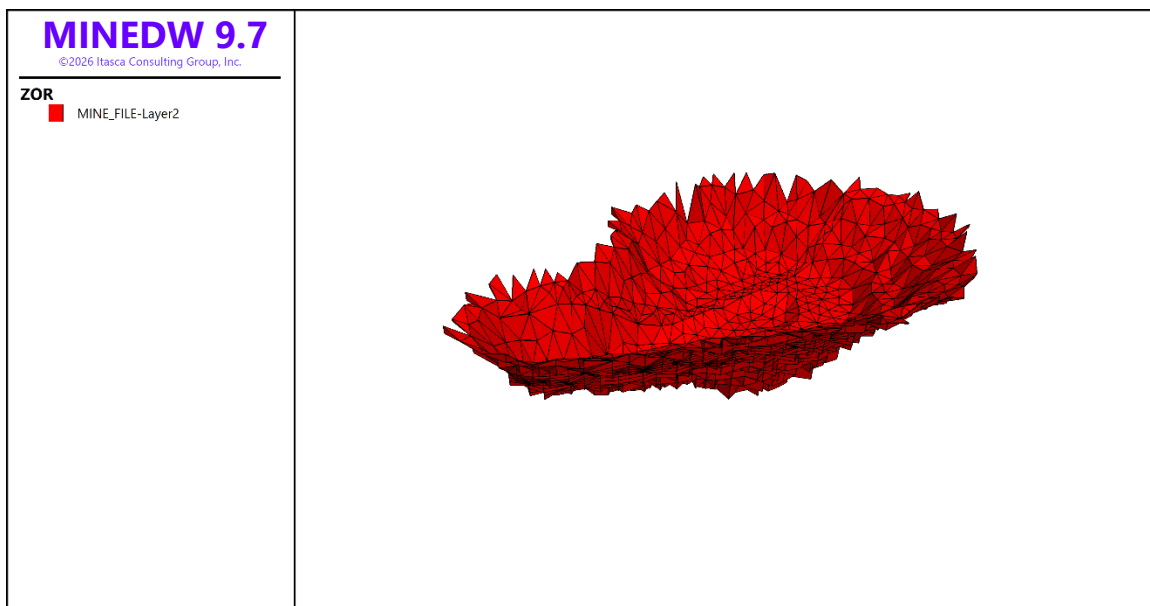


Figure 96. ZOR defined in the model.

3.1.7 Step 7. Open-Pit Backfilling

When mine operations and dewatering activities cease, a pit lake may form if the pit-bottom elevation is below the recovering water-table elevation. To simulate the pit-lake formation in **MINEDW**, select "Open Pit" from the "Mining" drop-down menu. In the "Open Pit" dialog box, ensure the "MINE_FILE" plan is showing in the drop-down box in the upper left-hand corner and then select the "Pit Lake" tab (Figure 93). Enter a date in the "Start Date:" box for the start of the pit-lake formation. Next, enter a value for "Initial Lake Elevation". This value must be equal to or greater than the ultimate pit-bottom elevation. In this tutorial, the open-pit "Initial Lake Elevation" should be the minimum pit-bottom elevation of 450 m (the bottom of the pit at the end of mining, (Figure 93). If evaporation from the pit-lake surface is to be considered, enter the evaporation rate value in the "Evaporation (mm/yr)" box. For this model simulation, evaporation is assigned a rate of 1,200 millimeters per year (mm/yr). Enter the number of stages to use to calculate the pit-lake volume in the "# of Stages" box. For this example, 15 stages (Figure 97) are used. Increasing the number of stages will give a more accurate estimation of inflow to the open pit but may cause convergence problems in the model simulation. As a rule of thumb, start with stages that are approximately 50 m in depth; if greater accuracy is required, increase the number of stages. Next, check the box next to "Activate" and click "OK" at the bottom of the dialog box to apply the changes to the model and exit the "Open Pit" dialog box.

Open Pit (length=m, time=days)

MINE_FILE [dropdown] [Add] [Delete]

General Pit Lake ZOR Backfilling

☒ Activate

Start Date: 2011-02-01

Initial Lake Elevation: 450

Evaporation (mm/year): 218

of Stages: 15 [Create] [Import...]

	Elevation	Area	Volume
1	820.232	0	0
2	820.232	536...	17.9524
3	820.232	536...	35.9049
4	820.232	536...	53.8575
5	820.232	536...	71.8102
6	820.232	536...	89.763

☐ Additional pumping/recharge, if elevation is lower than 820.232 ☐ Define Rate

[OK] [Cancel]

Figure 97. The "Pit Lake" tab.

3.1.8 Step 8. Adding Pumping Wells

Pumping wells are extensively used in open-pit dewatering systems. In **MINEDW**, there are three options available for pumping wells, all of which are explained in detail in the **MINEDW** Manual section 7.4.4. Pumping wells can be defined in the model by defining pumping rates, specified heads, or pumping rates with a lowest pumping elevation (LPE).

"Pumping Rate", "Pumping with LPE", and "Specified Head" wells will be added in one step. During the calibration and predictive simulations, it is important to use the correct type of pumping well. "Pumping Rate" and "Pumping with LPE" are two good choices for wells to use during calibration because the user can assign actual pumping rates from the field to these wells in the model. A "Specified Head" well is useful for predictive simulations, as it allows the user to specify the level to which they wish to lower the phreatic surface without needing to manually adjust the pumping rate.

To add pumping wells to the model, from the "Build Plot" window click "3D Contour". Next, toggle to layer 27 using the toggle buttons next to the "Layer" attribute on the "Attributes" tab (Figure 98). The layer where the pump is to be simulated must be selected in order for the user to correctly construct the pumping wells in the model. Nodes above the pumping wells are then used to simulate the well screen.

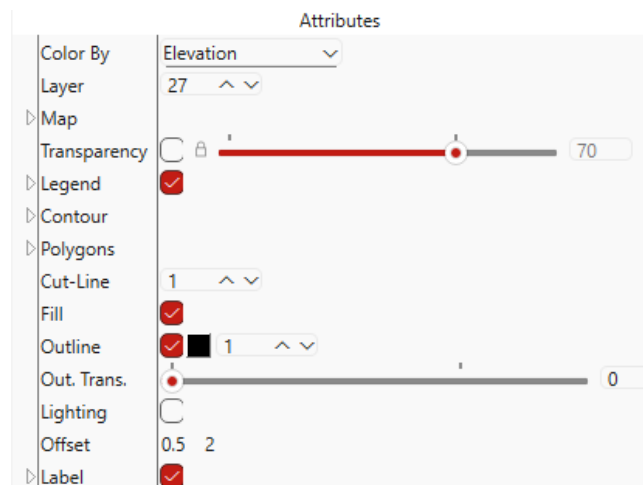


Figure 98. The "Control Panel" pane with the "Attributes" tab selected to show the correct layer for pumping wells.

In the **MINEDW** main window, click the "Select" button on the Main Menu banner. To locate the pumps in the model domain, use the pump locations that were used to create the mesh. Add a "Point Data File" plot item, which is found under the "Geometry" item in the "Buil Plot" window (Figure 99). Use the "File" attribute on the "Attributes" tab to open the "Select Data File" dialog box. Use the "Select Data File" dialog box to navigate to the location where the input data for these tutorials are located and select the "Pumps.txt" file in the "INPUT/MESH_INPUT/DAT" directory.

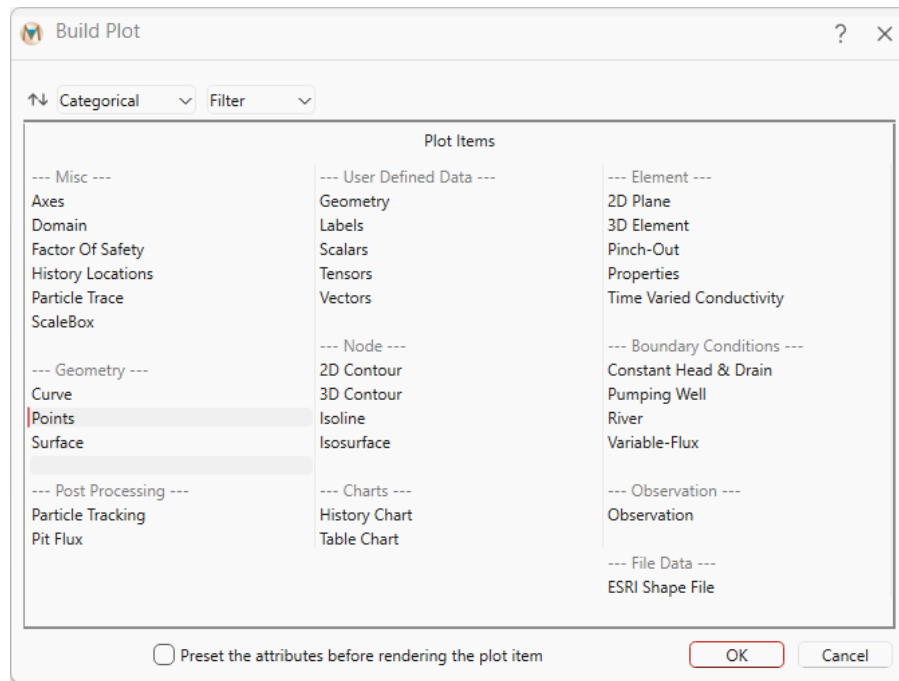


Figure 99. The "Point Data" plot item.

Select the nodes below the points now displayed in the View Pane and then press [Enter]. In the "Assign Properties for Nodes" dialog box, select the "Add to Pumping Wells" option (Figure 100), then click "OK". For this tutorial, the goal will be to dewater the open pit during active mining using any combination of the three pumping well types.

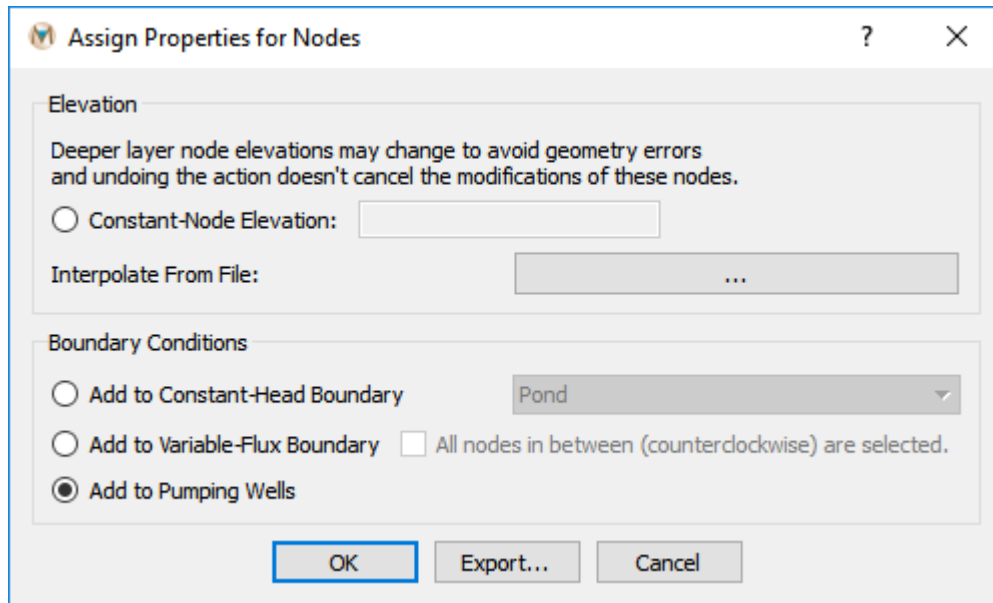


Figure 100. The "Assign Properties for Nodes" dialog box showing the "Add to Pumping Wells" option selected.

Next, open the "Pumping Well" dialog box by clicking on "BCs" on the Main Menu banner, then select "Pumping Well". The newly added pumping well nodes appear in the "Pumping Well" dialog box (Figure 101).

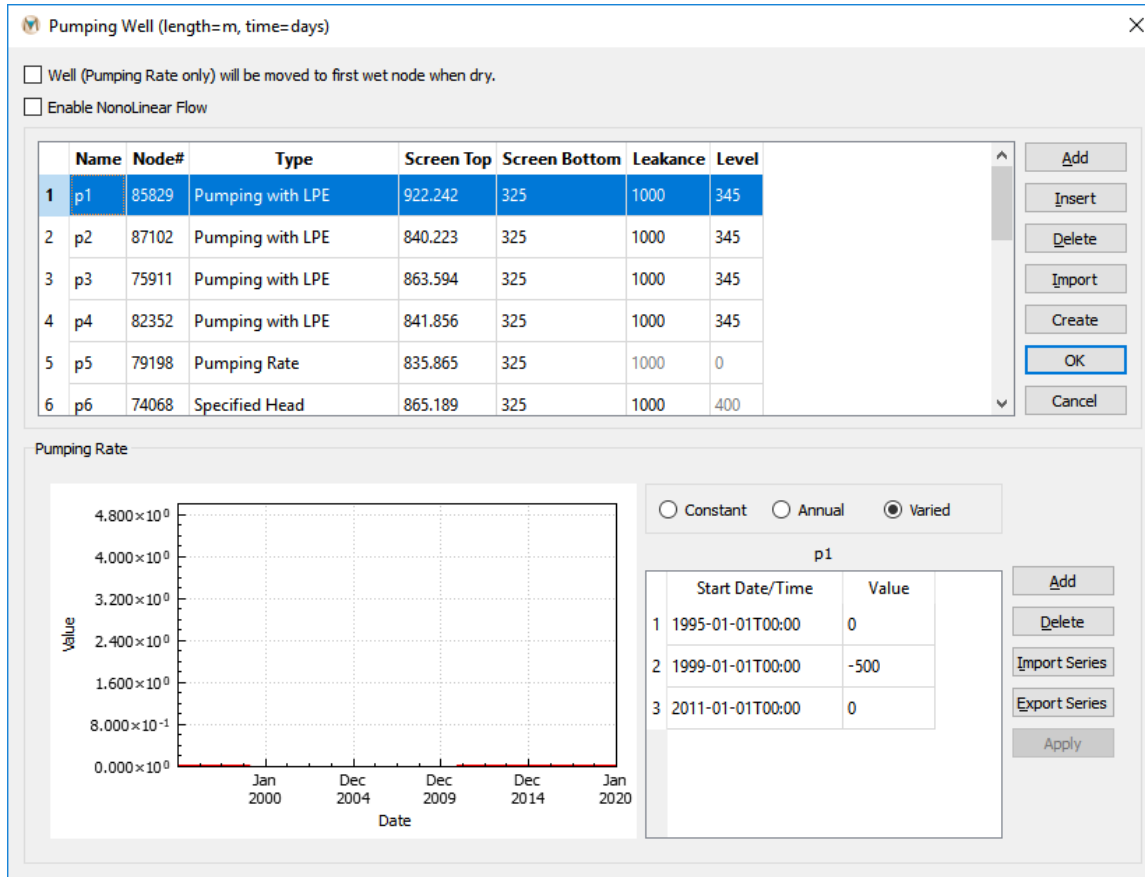


Figure 101. The "Edit Pumping Wells" dialog box.

Set the "Type" as "Pumping with LPE" for the first several pumping wells (Figure 100 and Figure 101). Define the "Screen Top" (initial screen top will be the ground-surface elevation), "Screen Bottom" (initial screen bottom will be the selected node elevation), LPE ("Level"), and pumping rate for all LPE wells. Choose a starting time for pumping to begin; this may be one of the parameters used to achieve dewatered conditions through the life of the mine. To turn pumping off, set the pumping rate to -9,999,999 in the last pumping-rate record of the dialog box. It is important that the last pumping-rate record be a large negative number if pumping is to be turned off. If it is not the last record or it is not a large negative number, **MINEDW** will not recognize that the well is to be turned off and will continue to extract water at that well. Pumping in this tutorial should end on or before the end of mining (Figure 101). If the pumping rate changes with time, go to the lower right window shown in Figure 101 to include pumping rates for different times. Pumping-rate data can be imported directly using the "Import" function at the lower right of the dialog box. The structure of the pumping rate file is described in Appendix B.

For “Pumping Rate” wells, select “Pumping Rate” under “Type” (Figure 102) and then enter the “Screen Top” and “Screen Bottom” information. In the time-series window, choose whether the pumping rate is “Varied”, “Annual”, or “Constant” and then enter the pumping-rate data. Pumping for the tutorial model should end at the end of mining or before, so ensure that the pumping rate is 0 on or before 1 January 2011.

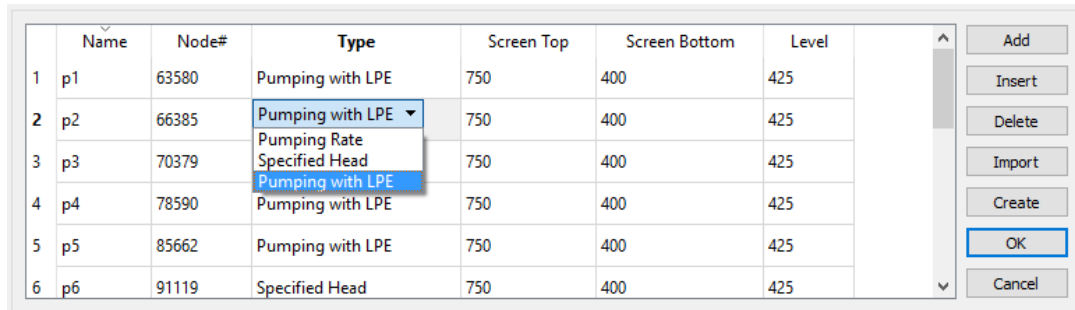


Figure 102. The “Edit Pumping Wells” dialog box showing the pumping well types available in MINEDW.

“Specified Head” wells can be used to lower the head in the groundwater system surrounding the well to a specified head level. This feature is useful for predictive simulations in which the pump and well must be designed to meet the dewatering objectives, but the pumping rate required to achieve the objective is unknown. To add “Specified Head” wells to the model, select “Specified Head” under “Type” and then enter the information for “Screen Top” and “Screen Bottom” and enter the specified head values in the rate time-series window below. For this tutorial, the “Specified Head” wells will be deactivated on 1 January 2011 by entering “-9,999,999” in the pumping-rate window located in the lower right of the “Pumping Well” dialog box. Any pumping-rate information entered in the pumping-rate window is ignored for “Specified Head” wells because abstraction rates are estimated by the hydraulic properties of the aquifer and hydraulic head at that point. The only value entered in the pumping-rate box that is recognized by **MINEDW** for “Specified Head” wells is a large negative number, such as “-9,999,999”, which will deactivate the “Specified Head” well.

3.1.9 Step 9. Defining the Boundary Conditions

3.1.9.1 Variable-Flux Boundary

A “Variable-Flux” boundary condition is assigned from the top of the model to the bottom along portions of the model domain. Ensure that Layer 5 is selected by clicking on the “Attributes” tab in the “Control Panel” pane, checking that “Layer” is set to 5, as shown in Figure 103.

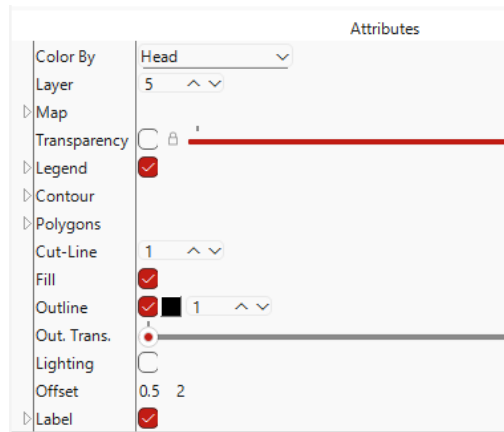


Figure 103. The “Attributes” tab.

From the **MINEDW** main menu, click the “Select with Rectangle” button on the Main Menu banner. Select the beginning node (node at top left corner) and ending node (the next node to the right) (Figure 104). The order of selection of the nodes is important because **MINEDW** will assign the nodes between the two selected nodes to the variable-flux boundary condition in a counterclockwise direction. After selecting the two nodes, press [Enter]. The “Assign Properties for Nodes” dialog box appears (Figure 105). Select “Add to Variable-Flux Boundary” and check the “All nodes in between (counterclockwise)” box, as shown in Figure 105 below. Enter “32” or whatever layer is the bottom layer in the “From Current Layer #1 To” box on the right.

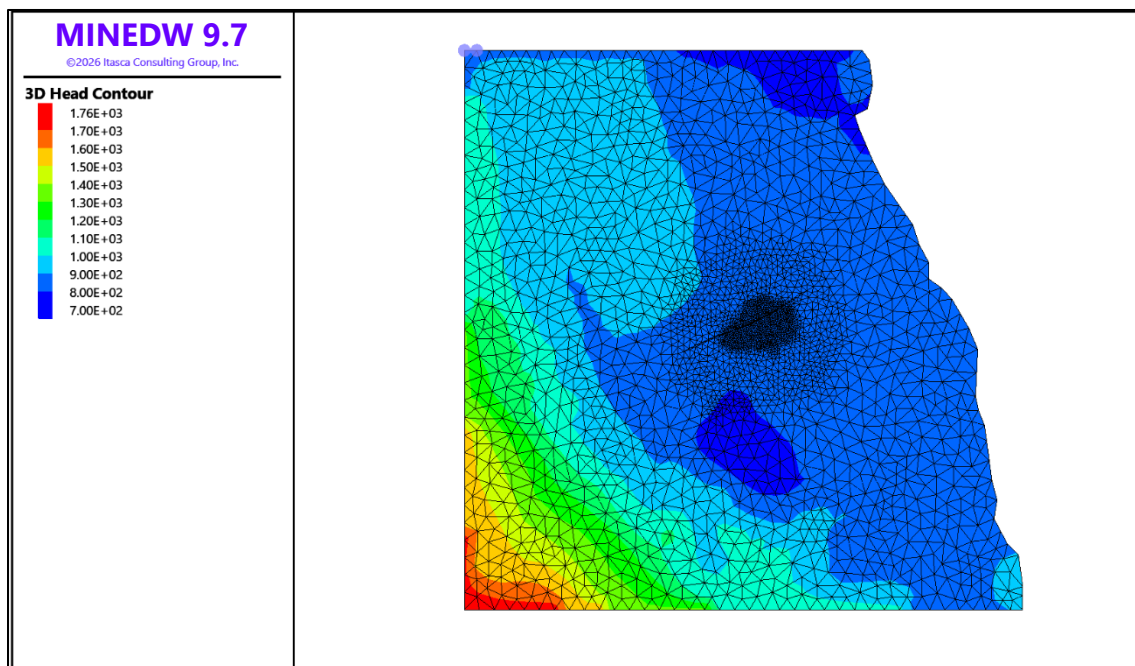


Figure 104. The View Pane showing the first and last node of the variable-flux boundary.

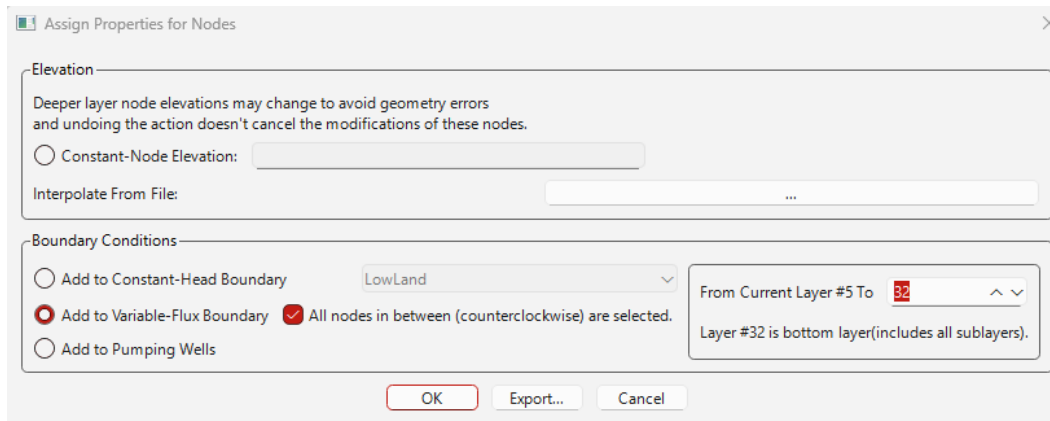


Figure 105. The “Assign Properties for Nodes” dialog box showing the “Add to Variable-Flux Boundary” option selected.

All nodes between the selected beginning node and ending node will be selected and assigned as a variable-flux boundary condition. The assignment will be for multiple layers since “32” was entered as the bottom layer, as shown in the dialog box above. Click “OK”.

The flow rates from the constant-head boundary condition around the perimeter of the model should be imported as flow rates for the variable-flux boundary condition that was just added to the model. To do this, click on “BCs”, then select “Variable Flux” to open the “Variable-Flux Boundary” dialog box. At the bottom of the “Variable-Flux Boundary” dialog box is the “Import From DRN File” button. Click on this button to open the “Open Variable Flux File” window. Use this window to browse to the location where the steady-state simulation was conducted and select the file with the .DRN extension and click “Open”. Flow rates for the variable-flux boundary condition will be imported. Check the box next to “Apply Constant Flux From Steady State Simulation” to apply the flow to the model simulation. Figure 106 shows the “Variable-Flux Boundary” dialog box with the imported flow rates at the bottom of the dialog box.

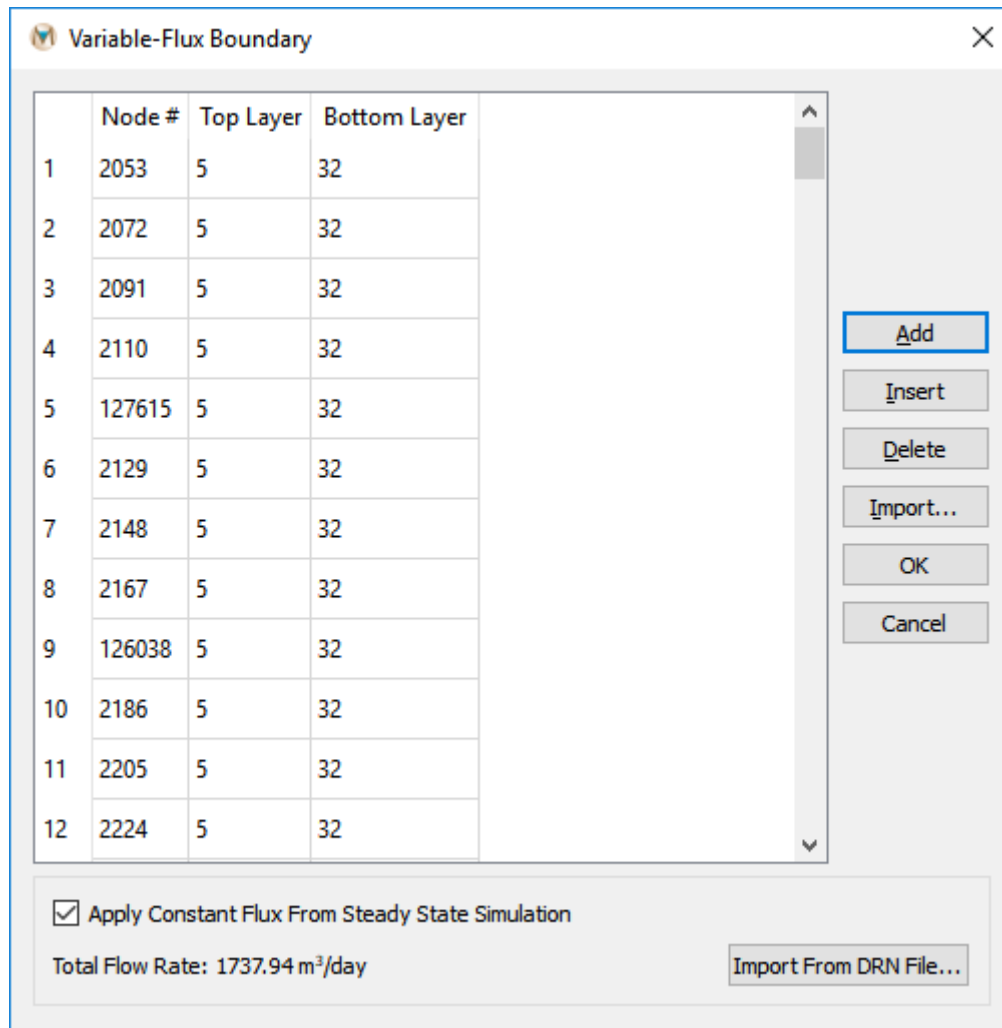


Figure 106. Imported flow rates.

The next step will be to remove the constant-head boundary conditions around the perimeter of the model domain that were used in the steady-state simulation. The model should not have two types of boundary conditions assigned to one node. To remove the perimeter constant-head boundary condition, select the "Constant Head" item from the "BCs" menu on the Main Menu banner. Using the "Constant-Head Boundary" dialog box, select the "MtBlock" group from the "Group Definition" drop-down box. Next, click the "Delete" button located at the top center of the "Constant-Head Boundary" dialog box. A warning will appear indicating that the group is not empty and will ask if the user wishes to proceed. Click "Yes", and this will delete this constant-head group. Repeat these steps for the "LowLand" group, but do not delete the "Pond" group.

3.2 Running the Model

To run the model, click "Run" on the Main Menu banner, then select "Validate". If the model passes the validation process, click "Run" on the Main Menu banner and select "Create Data Set". Select the folder where you want to create the data set. A data set must be created in order to run the model in MINEDW. Next, click "Run" on the Main Menu banner and select "Execute" to run the model. The following dialog box appears (Figure 107).

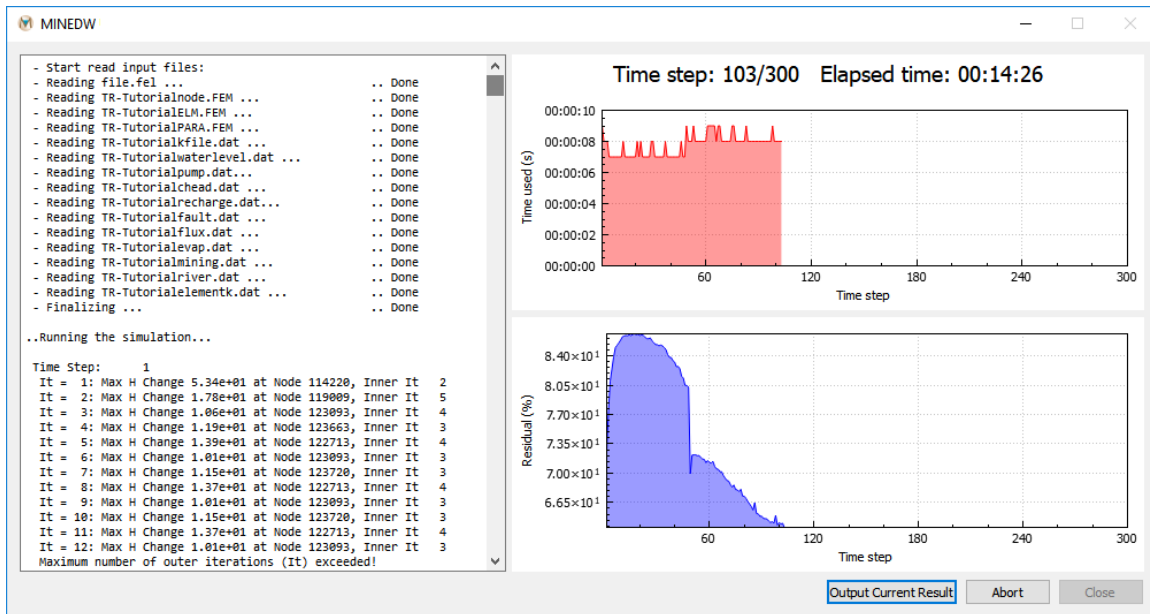


Figure 107. Screen display of output from the model run.

3.3 Results

To visualize and export the simulation results once the model run has finished, click "Results" on the Main Menu banner, and then choose "Read Results". Select the "*.PLB" file from the folder where **MINEDW** was executed. To view the desired time step, enter it in the variable field or move the time-step slider (found on the right side of the Main Menu banner, as shown in Figure 108) to the desired time step.

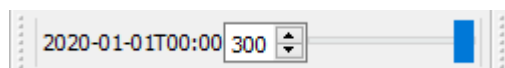
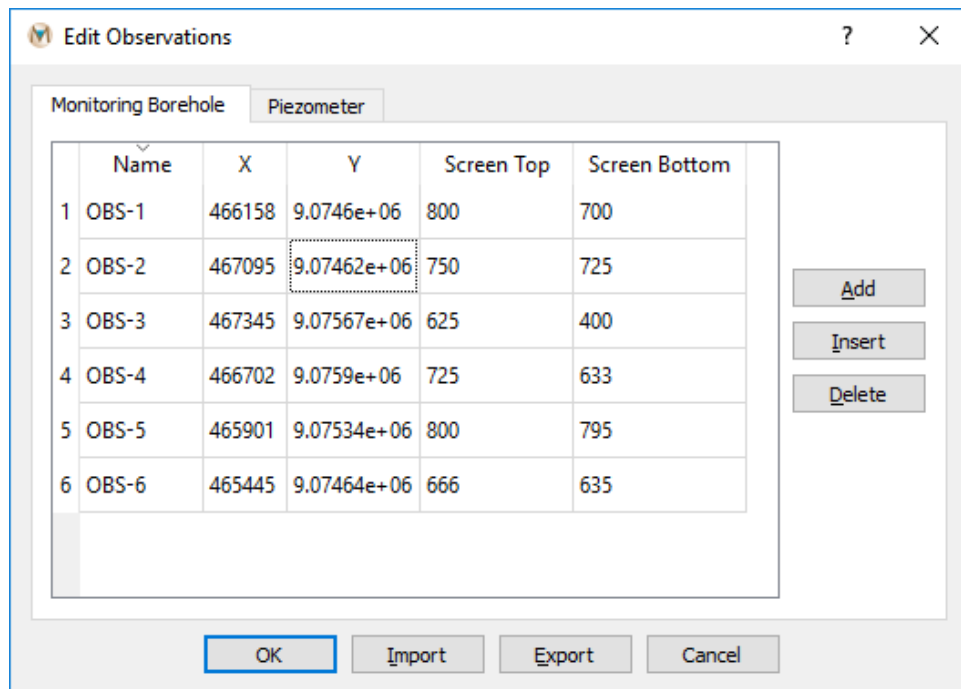


Figure 108. Time-step slider.

3.4 Creating Monitoring Points

Model-calculated head values can be extracted using observations from either monitoring boreholes or piezometers. Monitoring boreholes will provide the head along a screened interval, while piezometers provide the head at a single vertical point. Observation locations can be defined by using the "Observations" function under "Results". When "Observations" is selected from the "Results" drop-down menu, the "Edit Observations" dialog box appears (Figure 109). Select the

"Monitoring Borehole" tab, click "Add" and enter the "Name", "X" coordinate, "Y" coordinate, "Screen Top", and "Screen Bottom" information for each observation point as shown in the figure below, and then click "OK". To add "Piezometer" observation points, click on the "Piezometer" tab, then click "Add" at the top of the dialog box (Figure 110). This will add a piezometer location, but observation points, or "Transducers" as they are labeled in the legend, must be specified too. Each piezometer may have multiple transducers, each with its own X, Y, and Z coordinates. After adding a piezometer, click "Add" in the middle of the dialog box (Figure 110) to add a transducer. Define the X, Y, and Z coordinates of the transducer, then click "OK". It is also possible to import the observation points from a text file that is space delimited. The observation file format is provided in Appendix B of the **MINEDW** manual. The easiest way to learn the file format that **MINEDW** requires is to first create one by defining a "Monitoring Borehole" and "Piezometer" and then exporting to a "*.DAT" file. This can be done by clicking the "Export" button at the bottom of the "Edit Observations" dialog box.



	Name	X	Y	Screen Top	Screen Bottom
1	OBS-1	466158	9.0746e+06	800	700
2	OBS-2	467095	9.07462e+06	750	725
3	OBS-3	467345	9.07567e+06	625	400
4	OBS-4	466702	9.0759e+06	725	633
5	OBS-5	465901	9.07534e+06	800	795
6	OBS-6	465445	9.07464e+06	666	635

Figure 109. The "Edit Observations" dialog box.

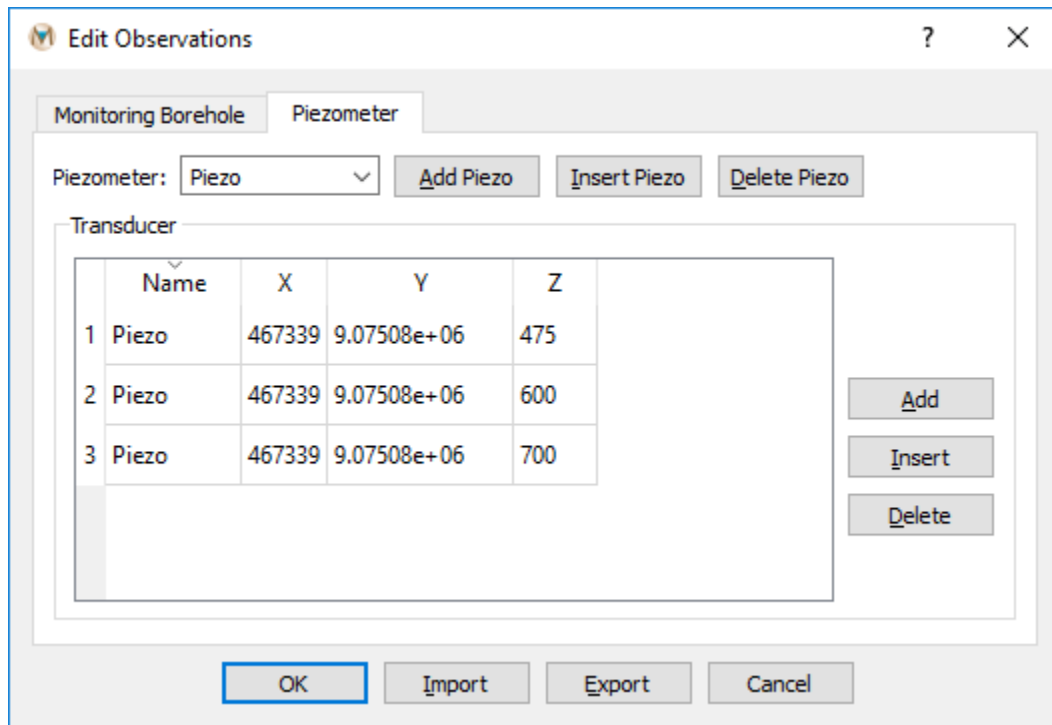


Figure 110. The “Edit Observations” dialog box with the “Piezometer” tab selected.

3.5 Creating Hydrographs

To create hydrographs from simulation data, select the “Hydrograph” option from the “Results” drop-down menu on the Main Menu banner. The water-level data for the observation will be exported to a “*.DAT” file. The exported file includes time steps, the corresponding date, well identification (ID), and corresponding water-level data for the entire model run. The .DAT file can be imported into Microsoft® Excel or other software to create tables and graphs.

3.6 Pit-Node Elevation and Water Table

The pit-node elevation and the water-table elevation can be plotted against time. To create this plot, click “Results” on the Main Menu banner, and then choose “Pit Node Elevation and Water Table”. The “Elevation of Pit Node” dialog box appears (Figure 111). Enter the “Node Number” on the left side of the window and then click “View”. For the specified node, the change in the water table beneath the selected pit node and the pit elevation will be plotted as shown in Figure 111.

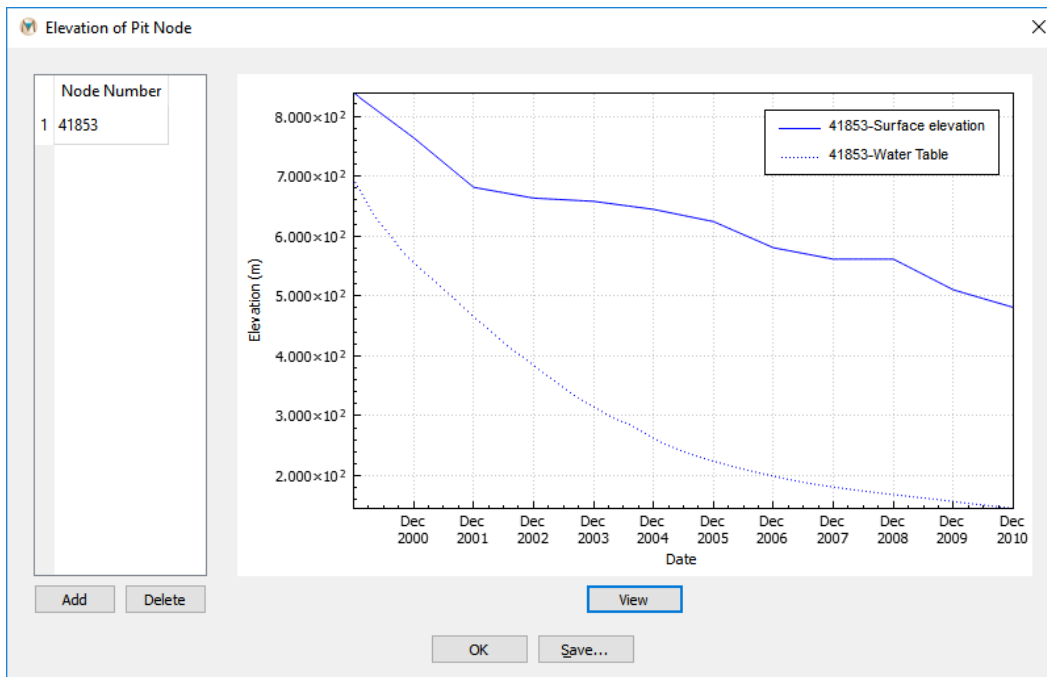


Figure 111. "Elevation of Pit Node" dialog box.

If multiple nodes need to be plotted, click "Add" and insert the additional "Node Numbers". Then click "View" again (Figure 112). The data used to create the pit-node and water-table plots can also be exported to a "*.DAT" file by clicking "Save". In the "Save Pit Node Elevation File" dialog box, navigate to the location where the file is to be saved and enter a "File name", then click "Save".

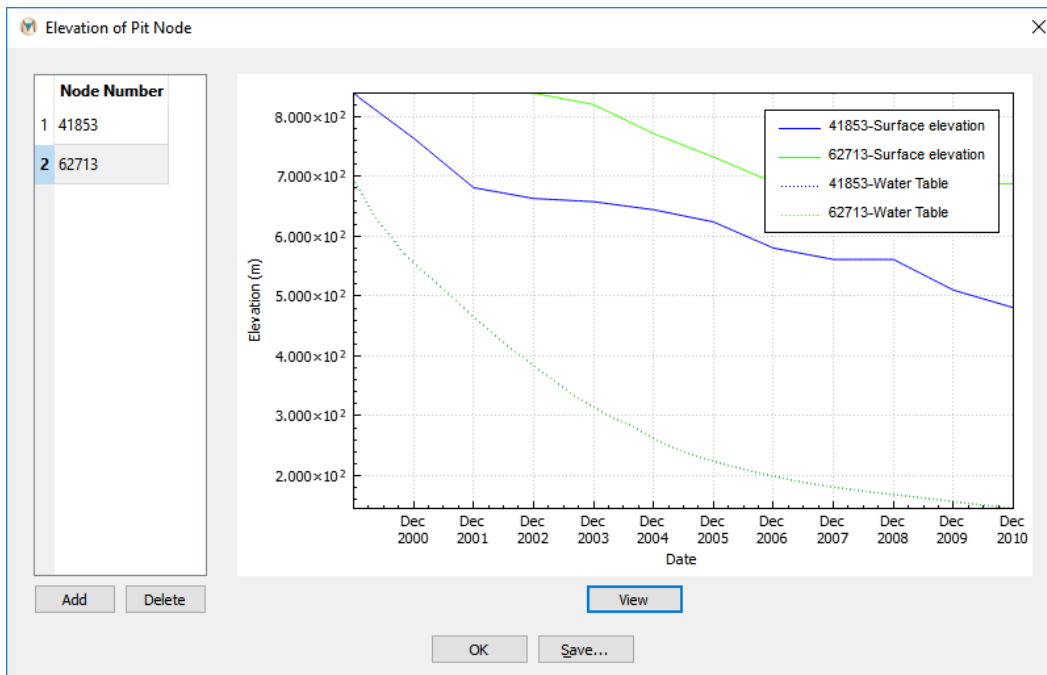


Figure 112. "Elevation of Pit Node" dialog box.

3.7 Head Distribution in 3-D

To plot the head distribution, select the "List" tab from the "Control Panel" pane, expand the "Node" item, and then double-click "3D Contour" (Figure 113). Next, select the "Attributes" tab from the "Control Panel" pane and select "Head" from the "Color By" drop-down menu. The head distribution appears in the View Pane. By changing the time step with the time-step slider, the desired time step can be viewed.

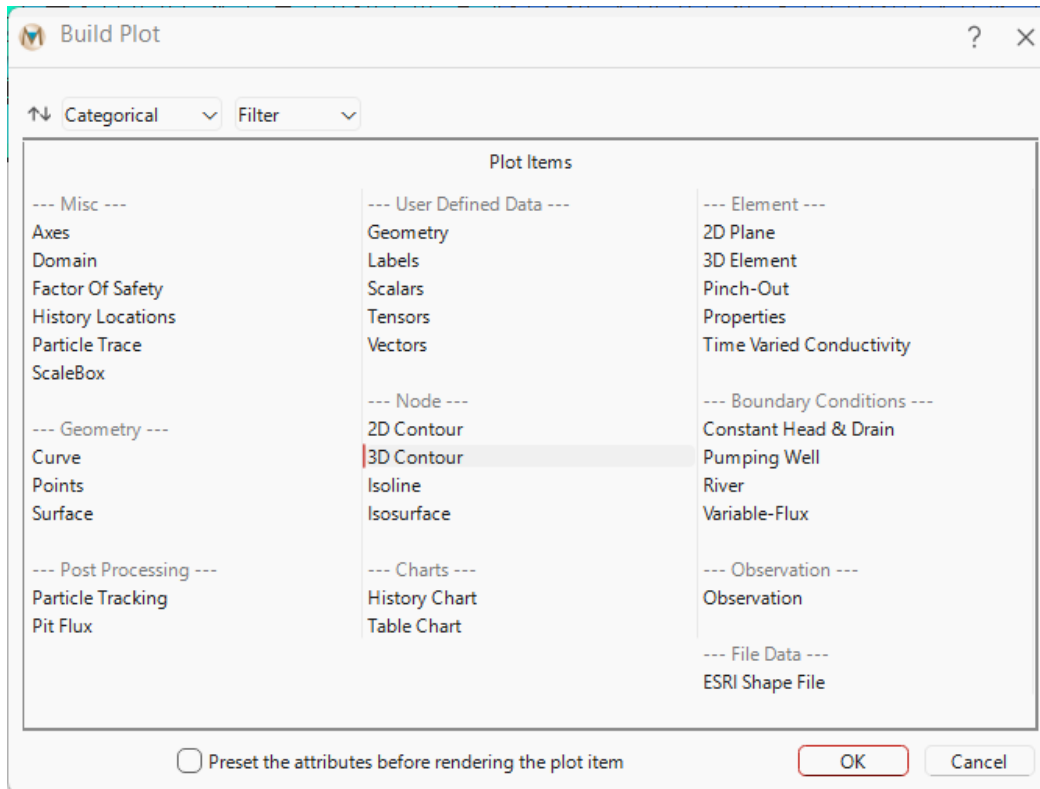


Figure 113. The "Control Panel" pane with the "Node" item expanded to show the "3D Contour" option.

To export the head distribution in 3-D for the desired time step as a bitmap, select "Export Base" from the "File" menu on the Main Menu banner and choose the "Bitmap" option. The exported bitmap is provided in Figure 114.

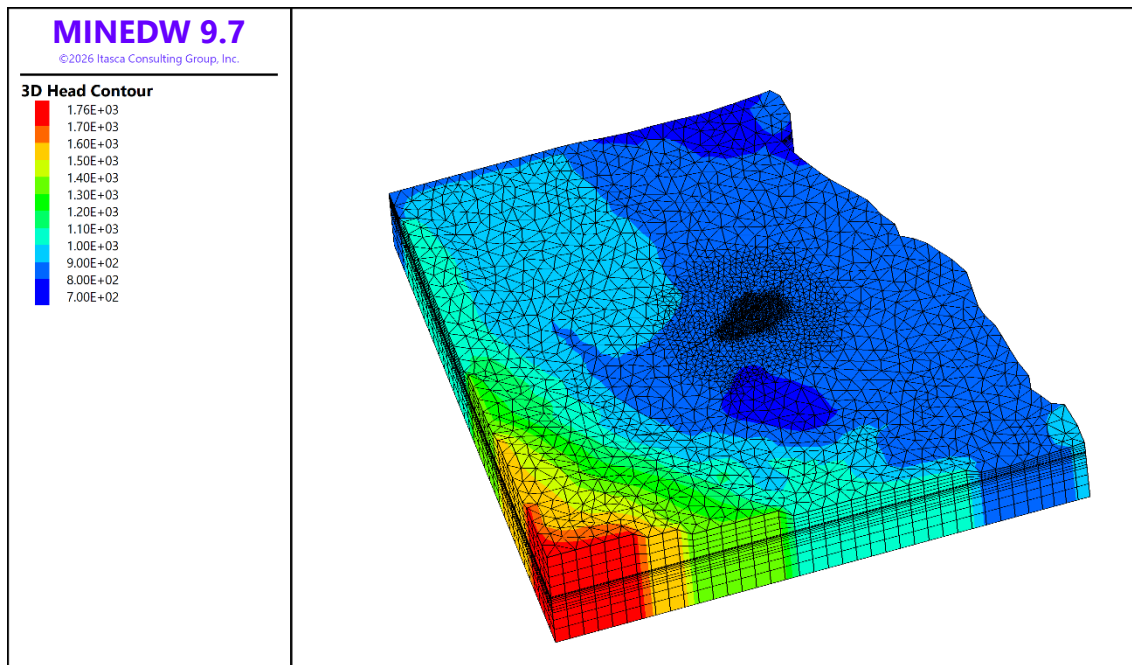


Figure 114. Screen display of head distribution in 3-D.

3.8 Head Distribution Cross Section

With the "3D Contour" plot item still displaying in the View Pane, click the "Create Plane" tool on the Main Menu banner. Choose two points on the "3D Contour" plot item, and a cross section through the model domain will be created. Cycle through the time steps representing the model simulation by using the time-step slider or entering the desired time step in the time-step slider box. To visualize the cross section in 3-D, select "3D Contour" under the "Control Panel", then expand "Cutplane" on the "Attributes" tab, and select the "Front" box (Figure 115).

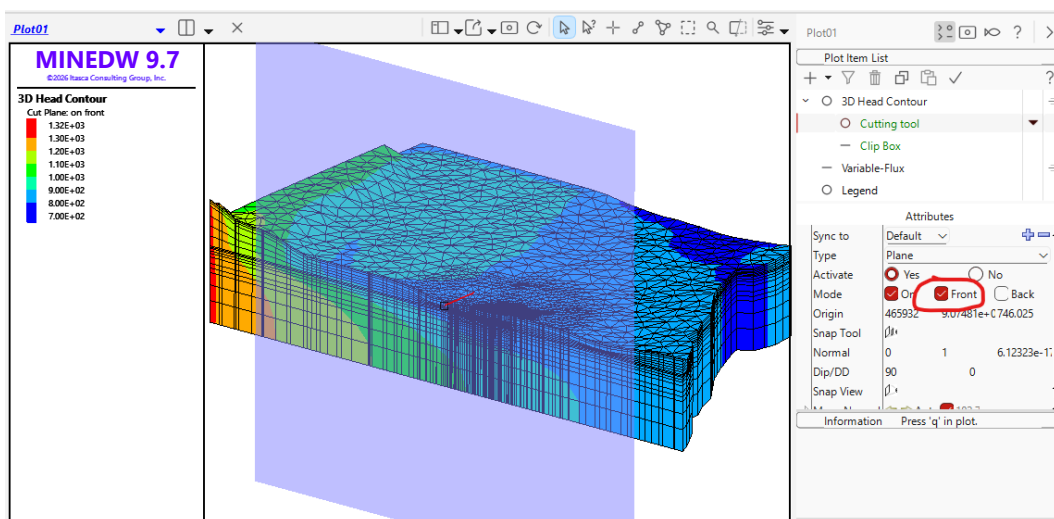


Figure 115. The "Control Panel/Plot Items" pane with the "3D Contour" item showing the "Attributes" tab.

To export the head distribution in 2-D for the desired time step as a bitmap, select "Export Base" from the "File" item on the Main Menu banner and choose the "Bitmap" option. The exported bitmap is provided in Figure 116.

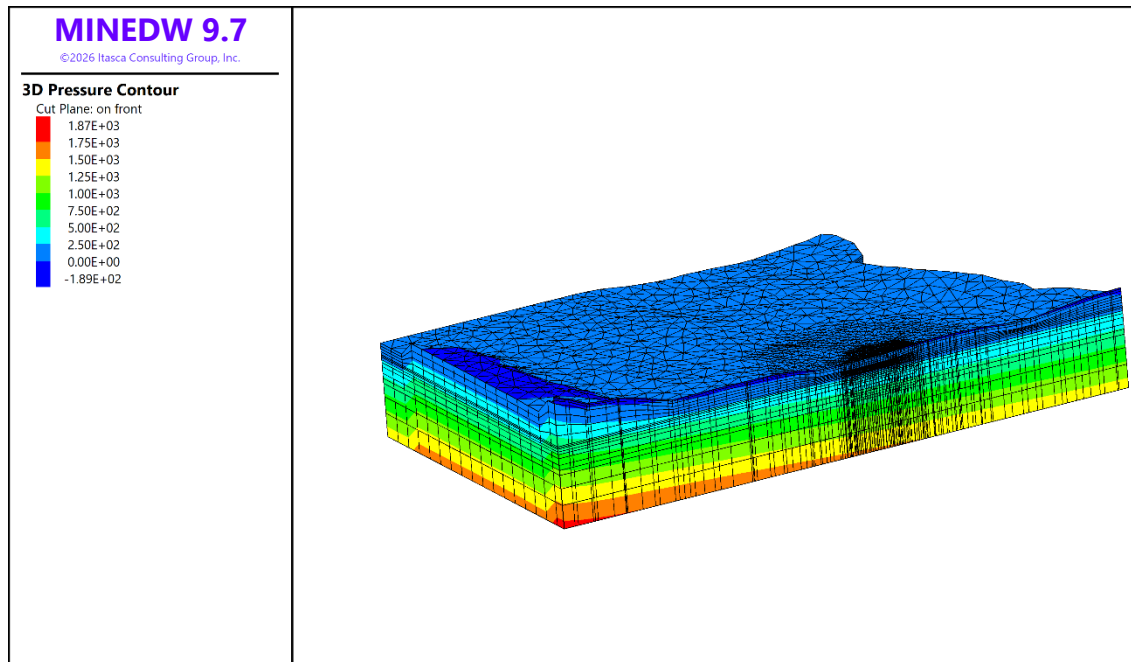


Figure 116. Screen display of output from the model run.

3.9 Exporting Pore Pressures to a Different Grid in 2-D

MINEDW can export pore pressures in both 2-D and 3-D to a user-specified grid for use in other models, such as geomechanical models. The data are exported as a "*.DAT" file for use in geomechanical models. To export pore pressures in cross section (2-D), select the desired time step, then click "Results" on the Main Menu banner, and then choose "Export Cross-Section Pore Pressure". The "Output Cross-Section Pore Pressure" dialog box appears (Figure 117).

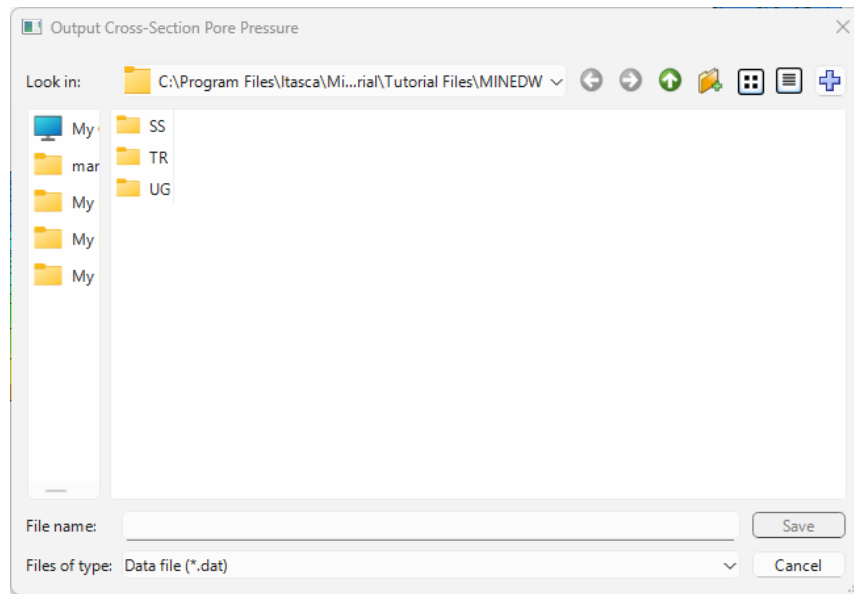


Figure 117. The “Output Cross-Section Pore Pressure” dialog box.

Enter a file name and click “Save”. The “Grid – Export Cross Section Pore Pressure” dialog box appears. Choose the interpolation method (“Inverse Distance” or “Kriging”) and define the related parameters (Figure 118), then click “OK”. The pore-pressure data are saved to the specified file and can be imported into any contour-mapping software, such as Golden Software’s *Surfer* or *ArcGIS ArcMap*, to plot the contours.

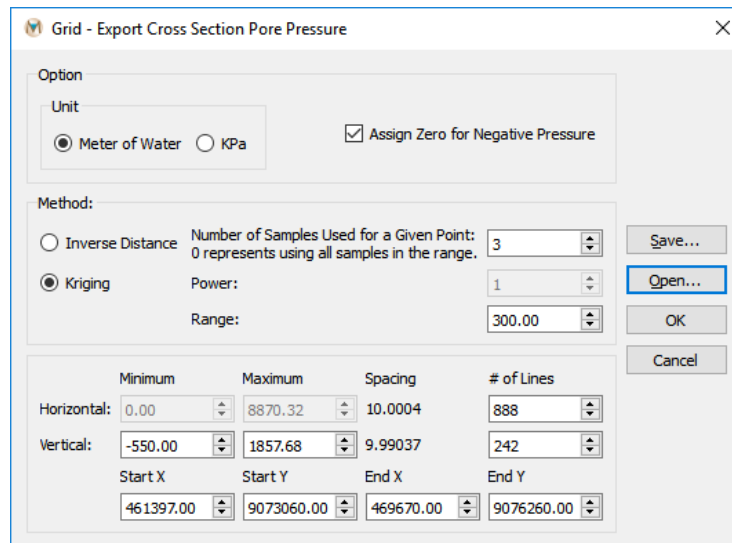


Figure 118. The “Grid – Export Cross Section Pore Pressure” dialog box.

3.10 Exporting Pore Pressures to a Different Grid in 3-D

To export the pore pressures in 3-D, select the desired time step, then click "Results" on the Main Menu banner, and then select "Export Pore Pressure". The "Output Pore Pressure" dialog box appears (Figure 119).

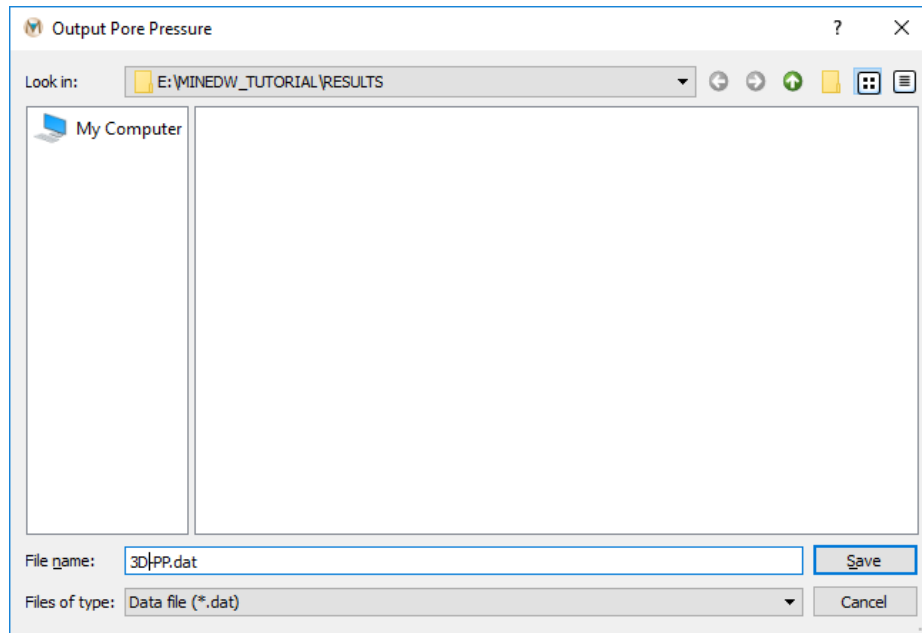


Figure 119. The "Output Pore Pressure" dialog box.

Enter a file in the file name box at the bottom of the "Output Pore Pressure" dialog box and click "Save". The "Grid – Export Pore Pressure" dialog box appears. Choose the interpolation method ("Inverse Distance" or "Kriging") and define the related parameters (Figure 120), then click "OK".

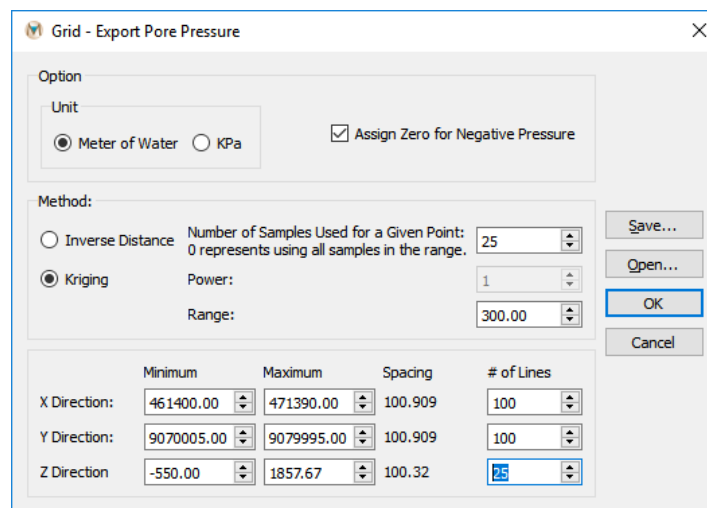


Figure 120. The "Grid – Export Pore Pressure" dialog box.

3.11 Seepage Component to Pit

MINEDW also computes the total seepage to a pit and contributions of each geological unit to seepage through time. To compute seepage to a pit, click "Results" on the Main Menu banner, and then choose "Seepage Component to Pit". The "Flow of Pits" dialog box appears (Figure 121). Select the "*.SEP" file from the folder where **MINEDW** was executed. Select "Geological Units" to determine the contributions of each geological unit to seepage.

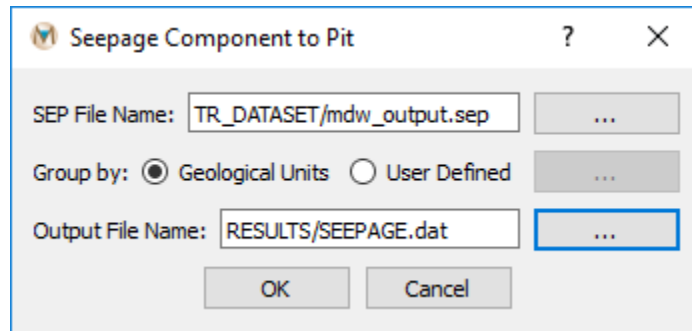


Figure 121. The "Seepage Component to Pit" dialog box.

To define an output name and location, click on the "..." button to the right of the "Output File Name" box. The "Save Pit Flow File" dialog box appears (Figure 122). Type the name of the file to save at the bottom (e.g., "SEEPAGE") and click "Save" and then click "OK". The seepage file is created.

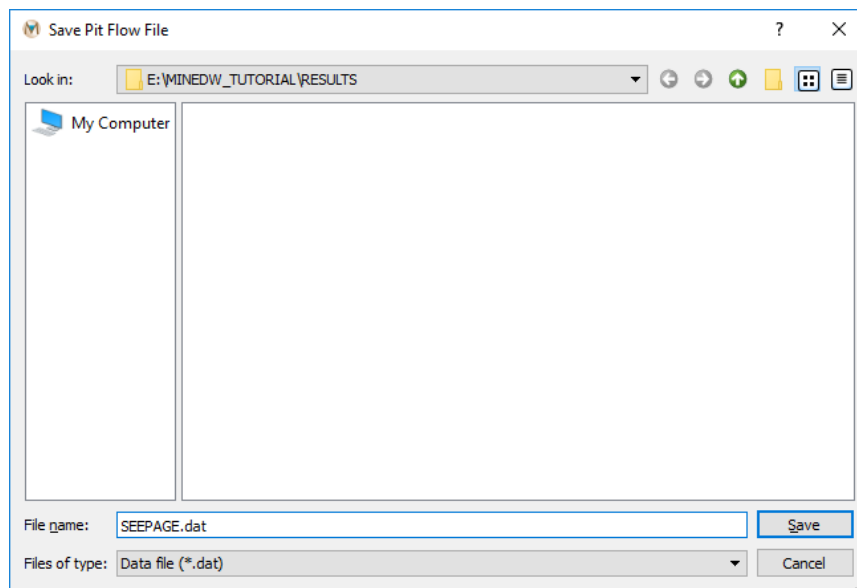


Figure 122. The "Save Pit Flow File" dialog box.

The "SEEPAGE.DAT" file can be imported to Microsoft® Excel or other plotting software to create time-series plots. Total seepage vs. time (Figure 123) and contributions from each geological unit to the seepage through time (Figure 124) can be plotted.

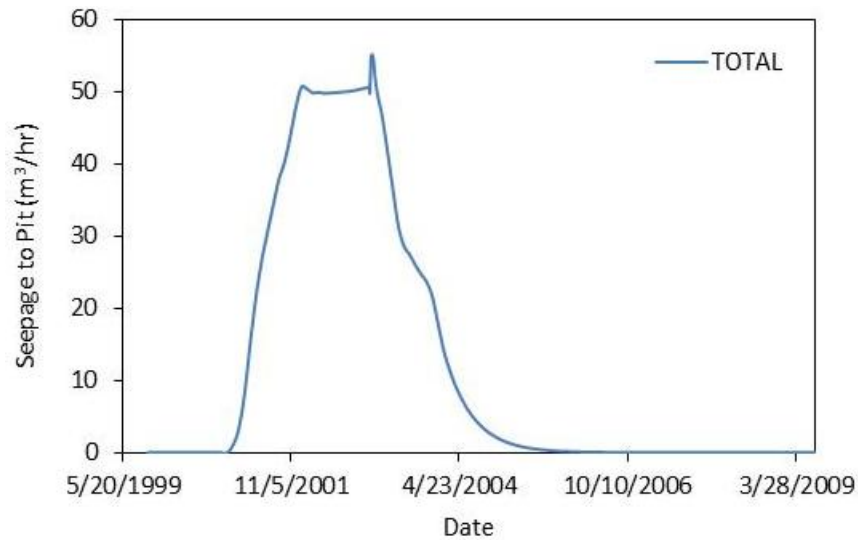


Figure 123. Total seepage to the pit through time.

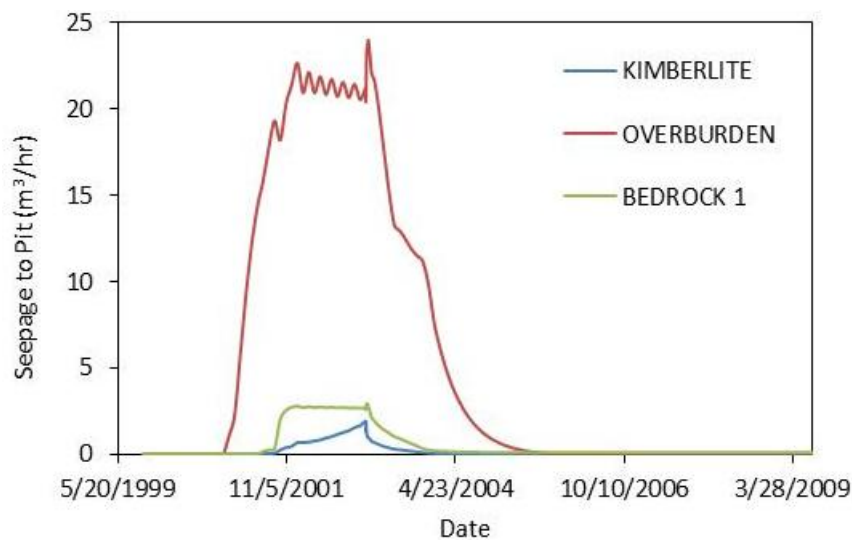


Figure 124. Contributions from each geological unit of seepage through time.

3.12 Pit Lake – Water Level vs. Time

The water-level change in a pit lake can also be computed in **MINEDW**. To plot pit-lake level changes through time, import the "PITLEV.LAK" file from the folder where **MINEDW** was executed into Microsoft® Excel. Figure 125 shows the pit-lake level changes through time. When analyzing

the water levels in the pit lake, use only the water level *after* the pit lake begins to form and the water level begins to rise.

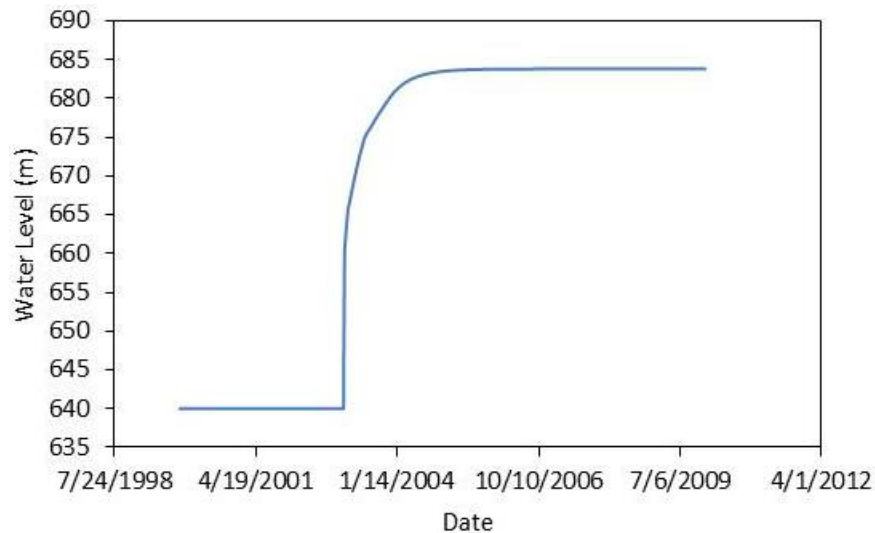


Figure 125. Pit-lake level changes through time.

3.13 Pumping Wells – Discharge vs. Time

Discharge through time from pumping wells is recorded in the "*.FLW" file by **MINEDW**. The discharge rate for "Pumping with LPE" or "Specified Head" wells, unlike "Pumping Rate" wells, is calculated during the model run. To plot pumping-rate data through time, import the "*.FLW" file into Microsoft® Excel or other plotting software and select the data for the desired pumping wells.

3.14 Particle Tracking

Particle tracking can be used to determine the capture zone of a well or the fate of groundwater within a zone of interest. To evaluate the capture zone of a well, backward particle tracking should be used, while forward particle tracking can be used to predict the fate of groundwater within a zone of interest. The examples provided below and in the "INPUT/PARTICLE TRACKING" directory employ both forward and reverse particle tracking.

To set up reverse particle tracking, choose points along the well screen of any one of the simulated pumping wells. The "INPUT/PARTICLE TRACKING" directory contains two files, "P9_points.DAT" and "P15_points.DAT", that contain points centered along the simulated screens for two pumps in this tutorial that can be used as a starting point to construct reverse particle tracks. Open the "Particle Tracking" dialog box by clicking on "Results" on the Main Menu banner and selecting "Particle Tracking". For each of the fields in the "Particle Tracking" dialog box, enter the values shown in Figure 126. Alternatively, an input file containing all of the necessary parameters for particle tracking is included in the "INPUT/PARTICLE TRACKING" directory. To use this file to populate the required fields in the "Particle Tracking" dialog box, click on "Open" at the bottom

center of the dialog box. Navigate to the location of the "INPUT/PARTICLE TRACKING" directory using the "Open Particle Tracking File" dialog box, select the file "P9 & P15_INPUT (reverse).dat", and click "Open". The "Particle Tracking" dialog box should now appear with all the fields filled in except "Result Output", as shown in Figure 126. Enter a file name for the output or, to select a different directory from "INPUT/PARTICLE TRACKING" in which to create the output file, click on the button above and browse to the desired location. Once the output file name and/or directory have been specified, click "OK" at the bottom of the "Particle Tracking" dialog box to begin the particle tracking calculations.

Particle Tracking

Start Time: 2010-12-31T00:00

Tracking Time (d): 89100

Step Size (m): 20

Tracking Direction: ☐ Forward ☒ Backward

☒ Capture

Result Output: ...

P9 & P15_OUTPUT (reverse).dat

	X	Y	Z
1	465815	9.07515e+06	835
2	465815	9.07515e+06	830
3	465815	9.07515e+06	825
4	465815	9.07515e+06	820
5	465815	9.07515e+06	815
6	465815	9.07515e+06	810
7	465815	9.07515e+06	805
8	465815	9.07515e+06	800
9	465815	9.07515e+06	795
10	465815	9.07515e+06	790
11	465815	9.07515e+06	785
12	465815	9.07515e+06	780

Add Delete

OK Advanced Open Save Cancel

Figure 126. Parameters for reverse particle tracking from a pumping well.

Once **MINEDW** has finished the particle tracking calculations, the particle tracks can be visualized in the View Pane by adding a "Particle Tracking" plot item. From the "List" tab in the "Control Panel" pane, double-click on "Post Processing" and then double-click on "Particle Tracking". Click on the empty field next to the "File" attribute on the "Attributes" tab for the "Particle Tracking" plot item. Using the "Select particle tracking file" dialog box that opens, navigate to the location where the particle tracking file was created by **MINEDW**, select the file, and click "Open".

The particle tracks may not immediately display depending on the currently selected time step. Using the time-step slider at the top of **MINEDW**, change the time step to the date selected as the start date for the reverse particle tracks. Results similar to those shown in Figure 127 below should display. This same process can be followed to evaluate the capture zone of any one of the wells used in a simulation.

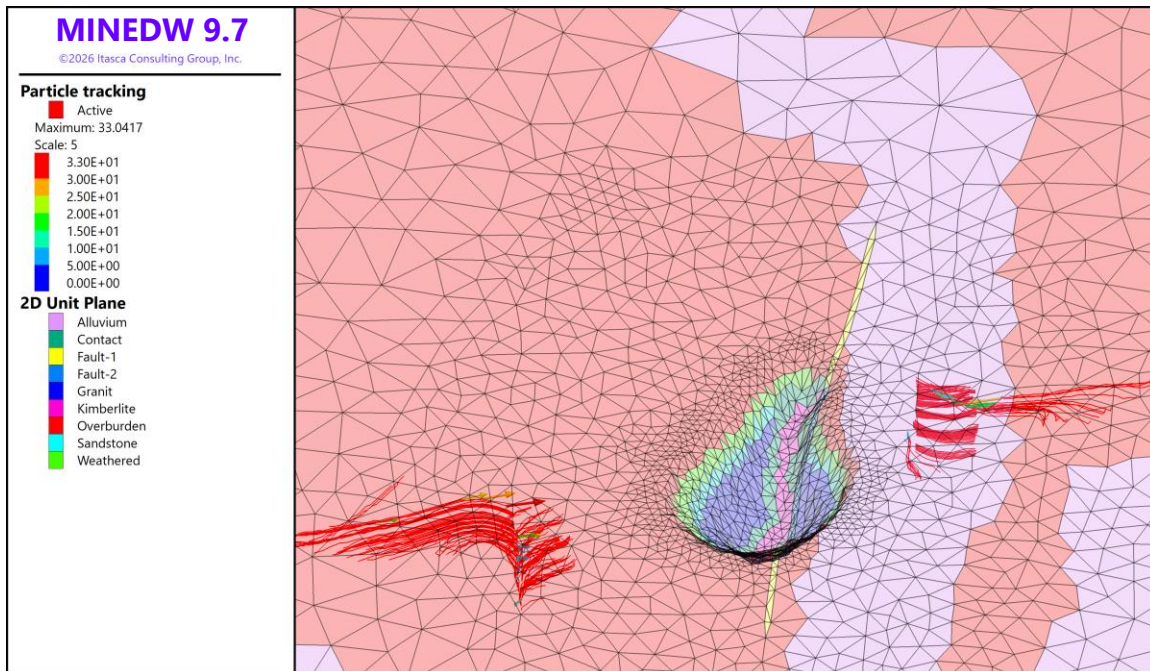


Figure 127. Reverse particle tracking results for two pumping wells.

Forward particle tracking can be used to evaluate if groundwater from a particular region is captured by pumping wells in the model. To perform forward particle tracking, locate the file named "Region_points.dat" in the directory INPUT/PARTICLE TRACKING. This file contains a group of points located slightly upgradient of the pumping well field in this tutorial. For simplicity, use these points and the particle tracking input file named "Region_INPUT (forward).dat" to populate the "Particle Tracking" dialog box as described in the previous example. Once all of the fields of the "Particle Tracking" dialog box have been filled in, run the particle tracking calculation by clicking "OK". Next, add a "Particle Tracking" plot item to the View Pane and select the recently created particle track file. Figure 128 shows the results of particle tracking for the points in the "Region_INPUT (forward).dat" file.

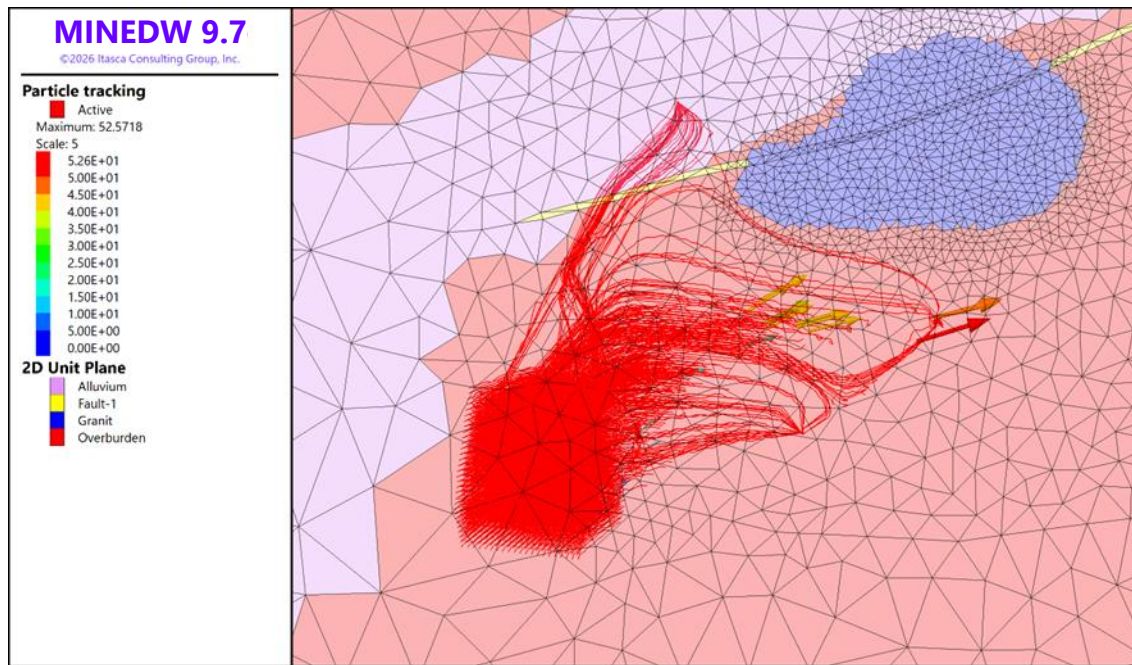


Figure 128. *Forward particle tracks for an area upgradient of the simulated well field.*

Only the particles placed within the zones of higher hydraulic conductivity are captured by the pumping wells. The lower geologic units have hydraulic conductivities several orders of magnitude lower than the upper units. Repeat the particle tracking exercise for any desired location by repeating the steps described above.

4.0 TUTORIAL 3. UNDERGROUND MINING

4.1 Problem Description

Tutorial 3 uses the steady-state (pre-mining) model generated in Tutorial 1 and describes how to construct a transient model for an underground mine that uses the sublevel caving method. This tutorial describes the procedure for creating an enhanced permeability zone for displaced rocks caused by sublevel caving. This enhanced permeability zone is referred to as the “zone of relaxation for caving” in this tutorial. Incorporating this permeable zone into the model will be discussed below.

4.2 Transient Simulation

The results from the steady-state simulation in Tutorial 1 are used as the initial conditions for the transient simulation.

4.2.1 Step 1. Opening the Project File

The model created in Tutorial 1 is used for this problem. To open the project file created in the first tutorial, click “File” on the Main Menu banner, then select “Open Project”. The “Load a **MINEDW** Project File” dialog box appears, as shown in Figure 129.

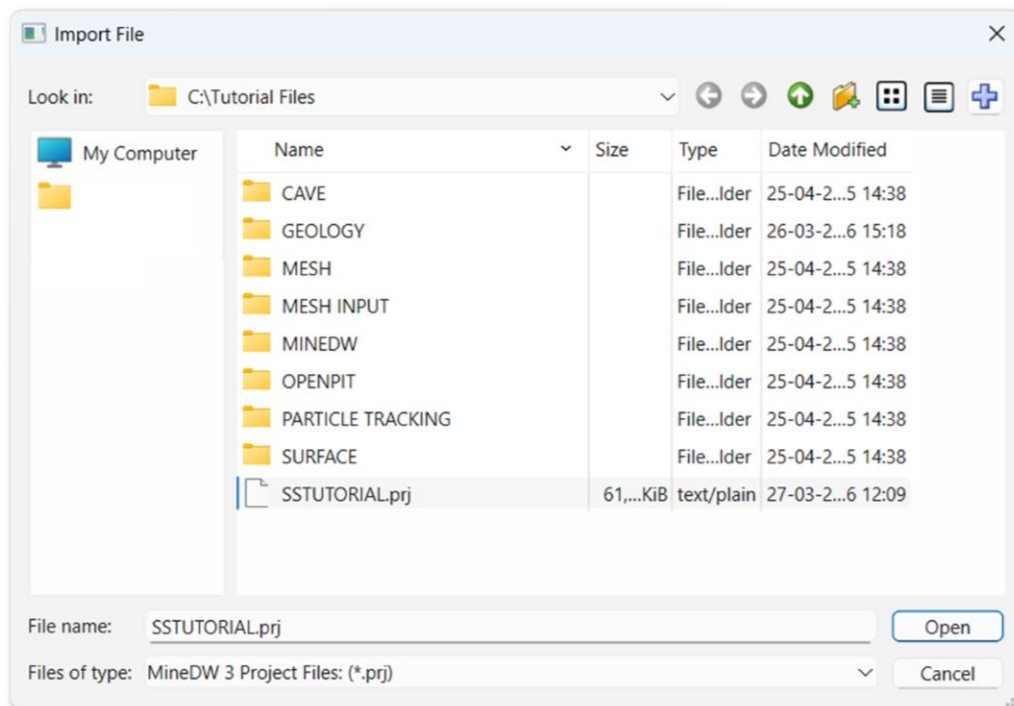


Figure 129. The “Load a MINEDW Project File” dialog box.

Select the project (.PRJ) file and click "Open". Next, select "File" from the Main Menu banner and then select "Save Project As" to save the project as a different .PRJ file for the transient underground simulation (e.g., UG.PRJ).

4.2.2 Step 2. Defining Project Properties

To change the project property, click "Project" and then select "Project Property". The "Project Properties" dialog box shown in Figure 130 appears. Make sure all the fields contain the same values as shown in Figure 130. Select the "Transient" option for this simulation.

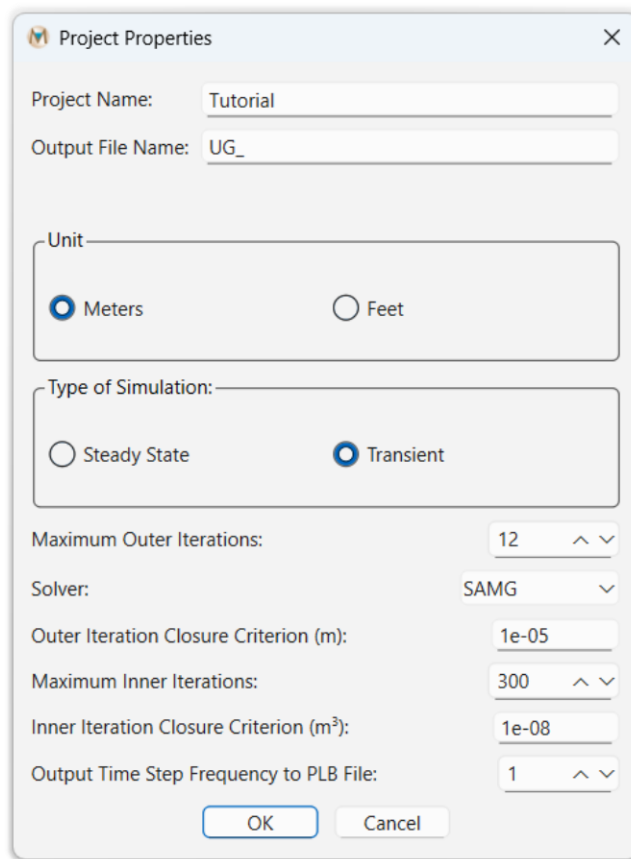


Figure 130. The "Project Properties" dialog box.

4.2.3 Step 3. Defining Initial Heads

To define initial heads, click "Project" on the Main Menu banner, then select "Initial Heads". The "Set Up Initial Head" dialog box appears. Choose the "Varied" option and select the water levels file (*.MDL) created during Tutorial 1 by clicking on the "..." button as shown in Figure 131, then click "OK".

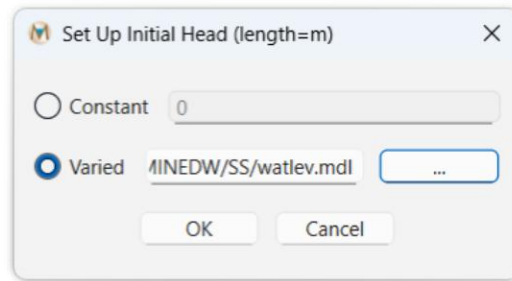


Figure 131. The “Set Up Initial Head” dialog box.

4.2.4 Step 4. Creating the ZOR file for Caving

In **MINEDW**, the enhanced permeability of displaced rock as a result of sublevel caving can be simulated. The two zones can be represented with different K values for the displaced rock.

To create the enhanced permeability zone of displaced rock, select “Zone of Relaxation for Caving” from the “Mining” drop-down menu on the Main Menu banner. The “Create Zone of Relaxation for Cave Zone” dialog box appears (Figure 132).

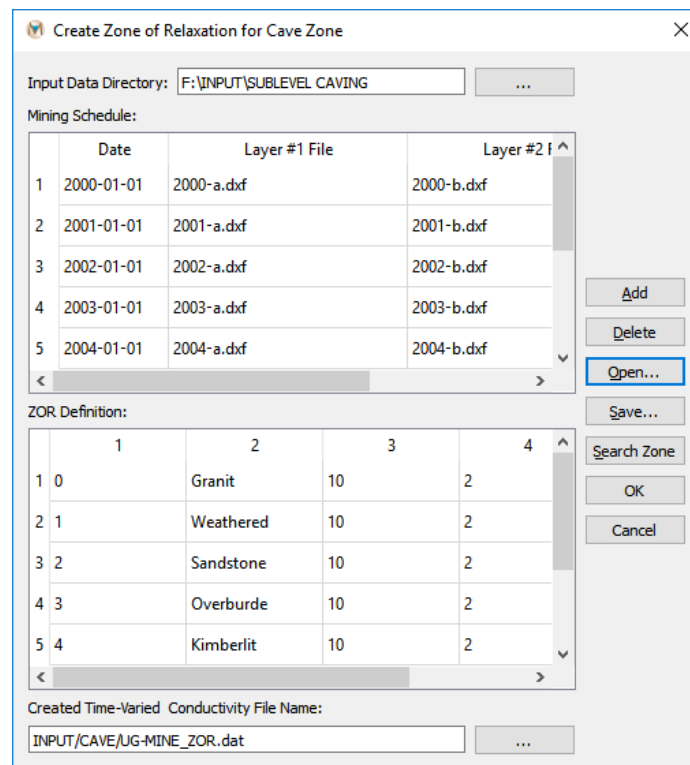


Figure 132. The “Create Zone of Relaxation for Cave Zone” dialog box.

First, enter or select the input data directory where the files you will be working with are located. This can be done by entering the path directly into the box next to *"Input Data Directory"* or by clicking the *"..."* button and navigating to the location of the files. Next, enter the dates of the mining schedule and the related .DXF file names in the *"Date"*, *"Layer #1 File"*, and *"Layer #2 File"* columns manually or use an underground-mining schedule file (Appendix B) to load data for Years 0 to 9. To use an underground-mining schedule file to load dates and file names, click on the *"Open"* button and select the *"UG.DAT"* file found in the *"INPUT"* folder. When the *"UG.DAT"* file is selected, the dates and file names will be listed as shown in Figure 3.3. In this example, *"2000_A.DXF"* represents a zone with the most displaced rock and *"2000_B.DXF"* represents a second zone with less displaced rock for the first year of mining. The other .DXF files are for subsequent years of mining.

Click *"Search Zone"*, and the geological units that are within the displaced volume will appear. Leaving the value under the field labeled *"Layer #..."* equal to 1 will exclude that geologic unit from the ZOR. For this simulation, use values of 25 and 10 for *"Layer #1"* and *"Layer #2"*, respectively. Select the folder where you wish to create the zone of relaxation for caving file by clicking the *"..."* button and navigating to the location. Next, enter the file name in the box at the bottom of the dialog box and then click *"OK"*.

4.2.5 Step 5. Adding the ZOR for Caving to the Model

To include the block caving ZOR in the model, select *"Underground Mining and Dewatering"* under the *"Mining"* drop-down menu on the Main Menu banner. The *"Dewatering and Underground Mining"* dialog box appears (Figure 133). Click *"Add"* and then select the *"ZOR"* tab. On the *"ZOR"* tab, click on *"Assign"* and, using the *"Open Time Varied Conductivity File"* dialog box, browse to the location where the ZOR file is located, select it, and click *"Open"*. After choosing the ZOR file, make sure that the *"Activate"* box on the *"ZOR"* tab is checked, then click *"OK"*. Multiple cave ZORs can be added to the model using this dialog box.

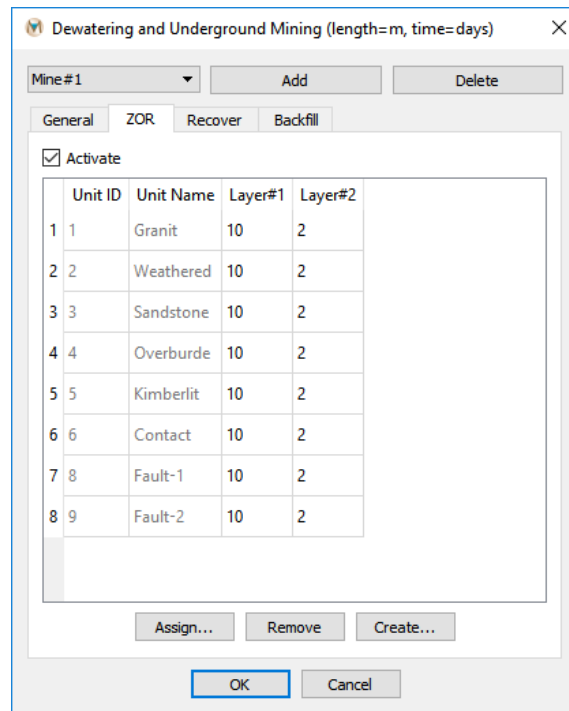


Figure 133. The “Time-Variied Conductivity” dialog box.

4.2.6 Step 6. Adding the Drain Nodes for the Cave Zone

Drain nodes are used to simulate groundwater extraction via the underground workings. To add the drain nodes to the model for simulation, begin by creating the necessary groups in the “Constant-Head Boundary” dialog box. Click on “BCs” on the Main Menu banner, then select “Constant Head” from the drop-down menu. Using the “Constant-Head Boundary” dialog box, click “Add” in the top center of the dialog box. Change the group name to “2000” for the first drain group. Select “Drain” from the drop-down box next to the group name. Repeat these steps to add drain groups “2001” through “2009”. Once the groups have been created in the “Constant-Head Boundary” dialog box, click “OK” on the left-hand side of the dialog box.

The drain groups have been created in the model, but currently no drains are assigned to the model domain. To correctly assign the drains to the model to simulate the dewatering of the underground workings, begin by adding the file named “Mine_Workings.DXF” in the “Input/Cave” directory to the View Pane. Using the “Layer” attribute, turn on only the “2000” layer. Next, add a “3D Contour” plot item to the View Pane. Toggle through the “Layers”, starting from layer 1, until the top of the “Mine_Workings.DXF” file becomes visible. Using the “Select” tool on the Main Menu banner, select the nodes around this portion of the “Mine_Workings.DXF” file. Once the nodes are selected, press [Enter] to open the “Assign Properties for Nodes” dialog box. Check the radio button next to “Add to Constant-Head Boundary” and then select the “2000” group from the constant-head group drop-down box, then click “OK”. Repeat these steps for all of the layers in the “Mine_Workings.DXF” file. The results should look similar to those displayed in Figure 134.

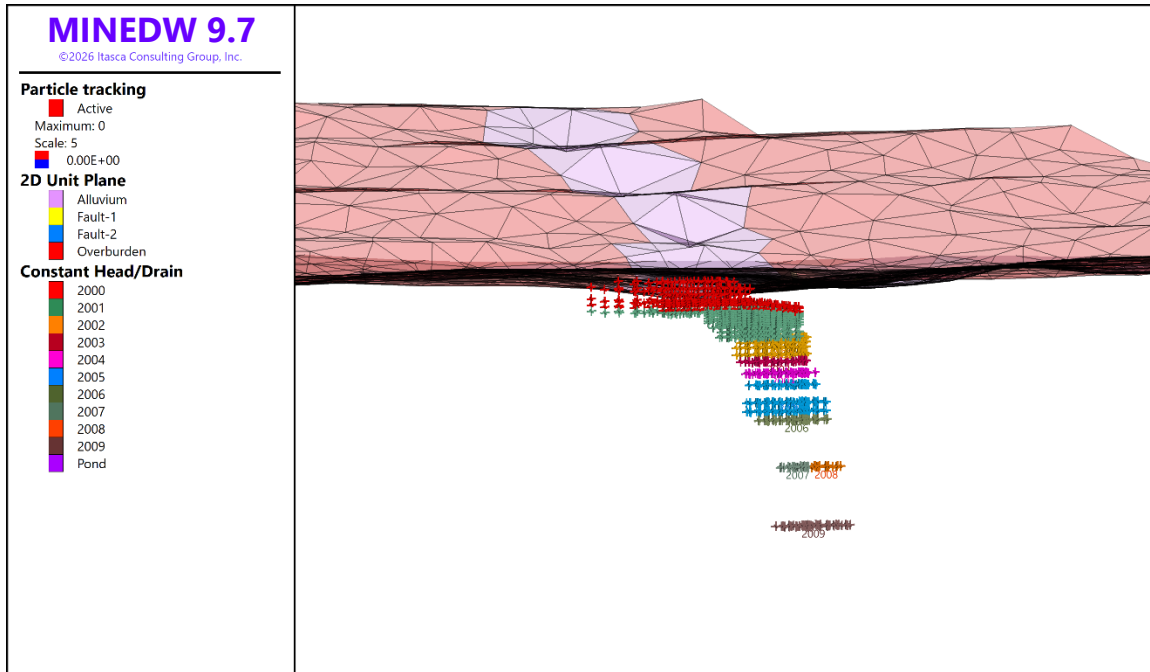


Figure 134. The drain nodes used to simulate the underground workings.

Once the drain nodes have been assigned to the model domain, the activation time will need to be modified for each one of the drain nodes. Each drain group should be activated on the date indicated by the group name to simulate the progressive excavation of the underground workings. For example, group "2000" should be activated on 1 January 2000 and group "2001" should be activated on 1 January 2001 and so on. All the drain groups should cease operation on 1 January 2010 to simulate the end of mining. To turn off the drain nodes, enter "-9,999,999" for the drain level in the time-series window. Now that the underground mine has been added to the model, they should be added as part of the mining simulation.

Adding the created drain nodes to the mining simulation will instruct **MINEDW** to summarize flow to these drain nodes in a special file ending in **"*.MNE"**. The summarized flow data in the **"*.MNE"** file could be calculated from the data reported in the **"*.FLW"** file, but adding the drain nodes to the mining simulation simplifies these steps. To add the drain nodes that are dewatering the underground workings to the mining simulation, open the **"Dewatering and Underground Mining"** dialog box by clicking **"Mining"** then selecting **"Underground Mining and Dewatering"** on the Main Menu banner. Under the **"General"** tab in the **"Dewatering and Underground Mining"** dialog box there is a drop-down box next to the window titled **"Drain Group Used"**. The drop-down box should contain the constant-head and drain groups that were previously added to the model using the **"Constant-Head Boundary"** dialog box. Select the **"2000"** group, then below the drop-down box, click **"Add"**. The drain group name should now be visible in the **"Drain Group"**

Used" window and should not appear in the drop-down box. Repeat these steps for drain groups "2001" through "2009". The completed "Dewatering and Underground Mining" dialog box should appear similar to what is shown in Figure 135. Now the model is ready to run. Refer to the following section for running the underground mining simulation.

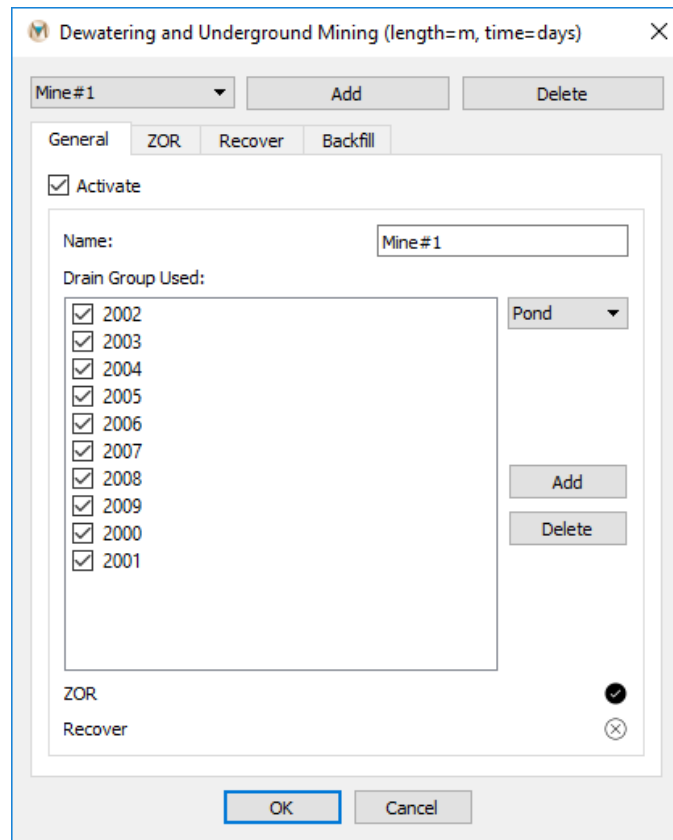


Figure 135. Drain groups used to simulate underground mining

4.3 Running the Model

To run the model, click "Run" on the Main Menu banner, then select "Validate". If the model passes the validation process, then click "Run" on the Main Menu banner and select "Create Data Set". Select the folder where you want to create the data set. Next, click "Run" on the Main Menu banner and select "Execute" to run the model.

4.4 Results

Once the simulation has ended, click "Results" on the Main Menu banner, and then choose "Read Results". Select the "*.PLB" file from the folder where **MINEDW** was executed to import the simulation results into **MINEDW** for visualization. Next, add a "3D Contour" plot item to the View Pane and use the time-step slider to view the desired time step. Figure 136 displays a cross section of head at the end of mining just before dewatering operations end.

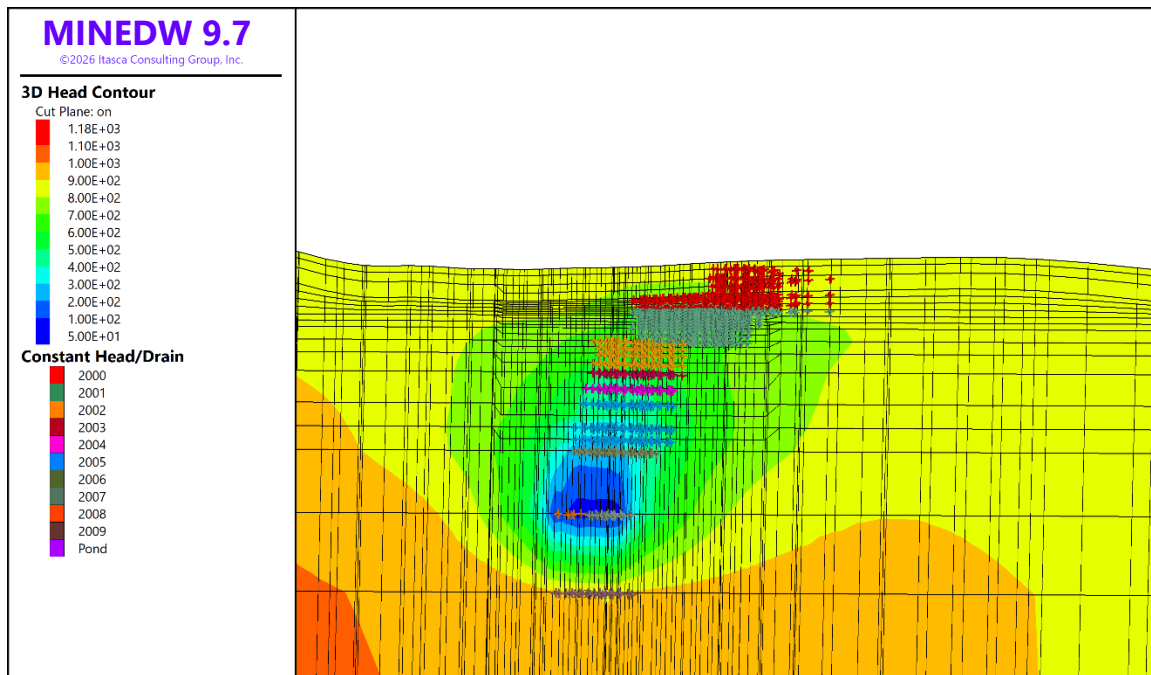


Figure 136. Cross section through the mining area displaying hydraulic head.

As previously mentioned, **MINEDW** computes the total inflow to the underground workings. This information is recorded in the file `"*.MNE"` extension in the folder where **MINEDW** was executed. To plot seepage to the underground workings below the cave zone, open the `"*.MNE"` file using Microsoft® Excel or other plotting software. Figure 137 shows total inflow to the underground workings.

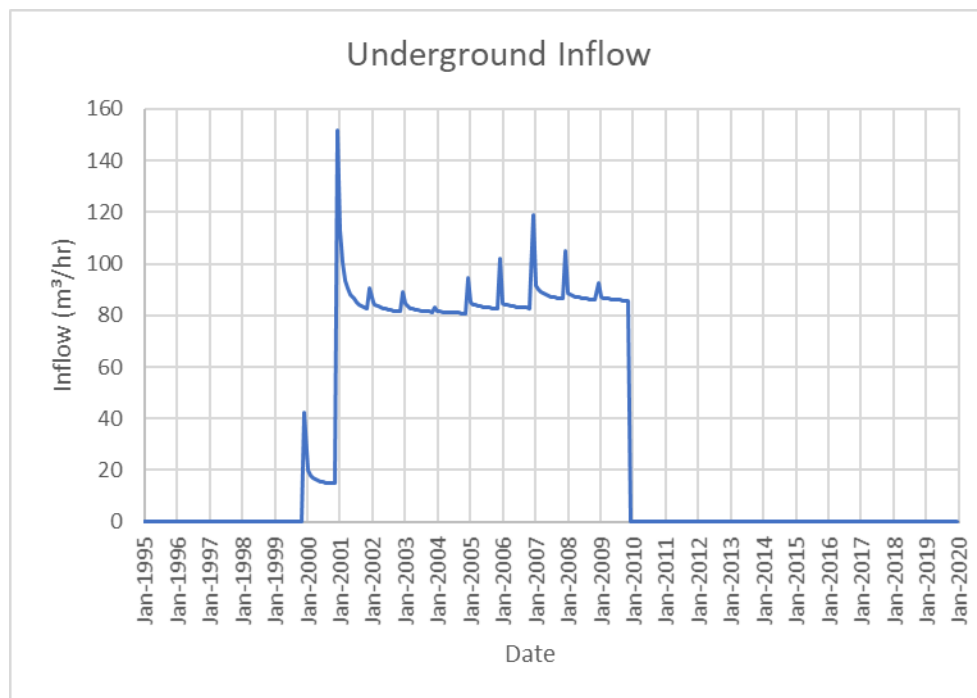


Figure 137. Total inflow to the underground workings through time.

For additional outputs that can be generated from a model simulation, please refer to section 2.3. All of the outputs, with the exception of the “*Pit Node Elevation and Water Table*”, “*Seepage Component to Pit*”, and “*Pit Lake Water Level vs. Time*”, can be generated for the underground mining simulation. This tutorial did not use pumping wells for dewatering the underground mine; however, if pumping wells had been used, the discharge over time could also be plotted. Discharge from pumping wells is recorded in the file with the extension “*.FLW”.