MINEDW TUTORIAL

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.

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.

Tutorial 1. Steady-State Groundwater Flow Model

1.1. 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 1.1.

Table 1.1. 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

1.2. *MINEDW* Model Construction

Step 1. Generating a Mesh

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 1.1. 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.

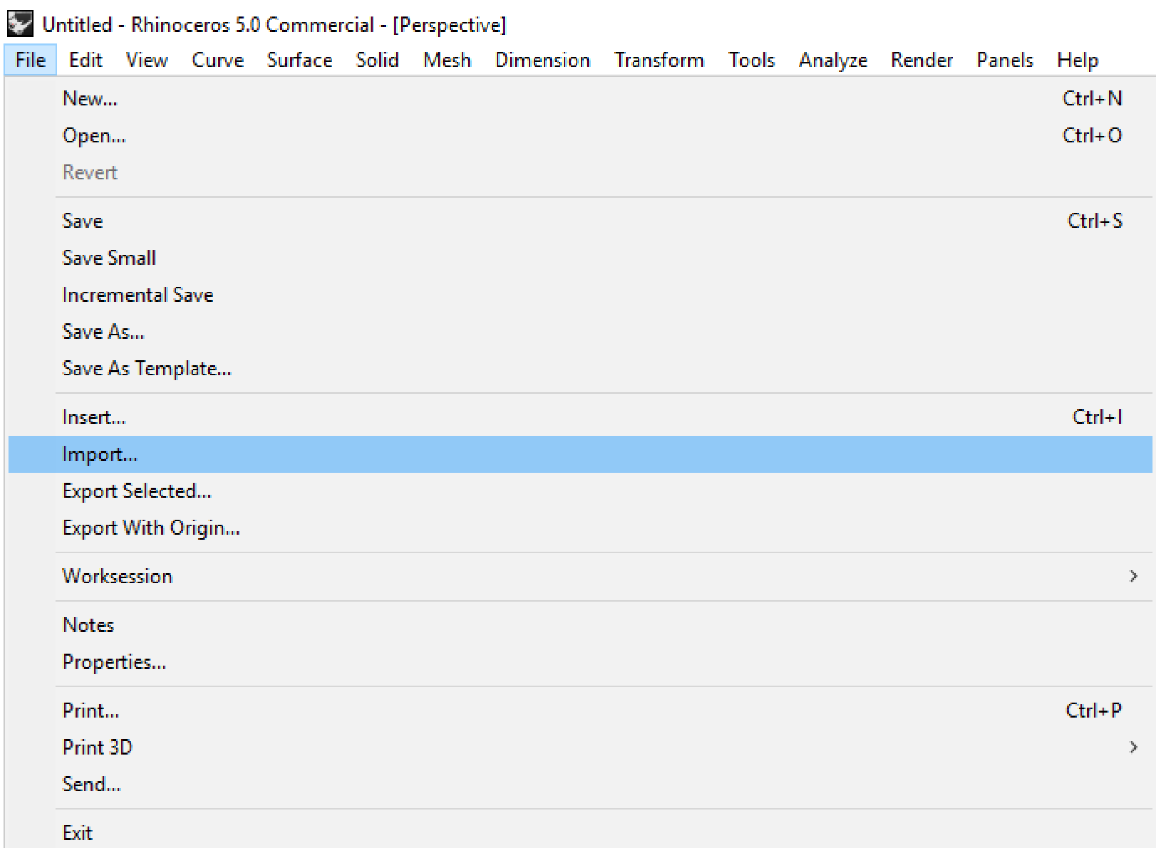


Figure 1.1. The "*File*" menu

Rhino has four default layout view ports to view the imported data: "*Perspective,"* *"Top," "Front,"* and *"Right"* (Figure 1.2). 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.

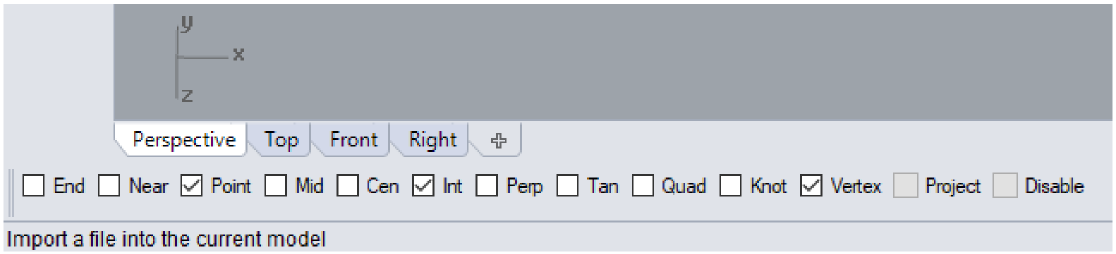


Figure 1.2. The default view ports in *Rhino*

After importing the required files, the result should appear similar to Figure 1.3. The *"MODEL BOUNDARY"* must be a completely closed region in order to generate a mesh. The *"OPEN PIT"* and "*TRANSITION ZONE"* (Figures 1.3 and 1.4) 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.

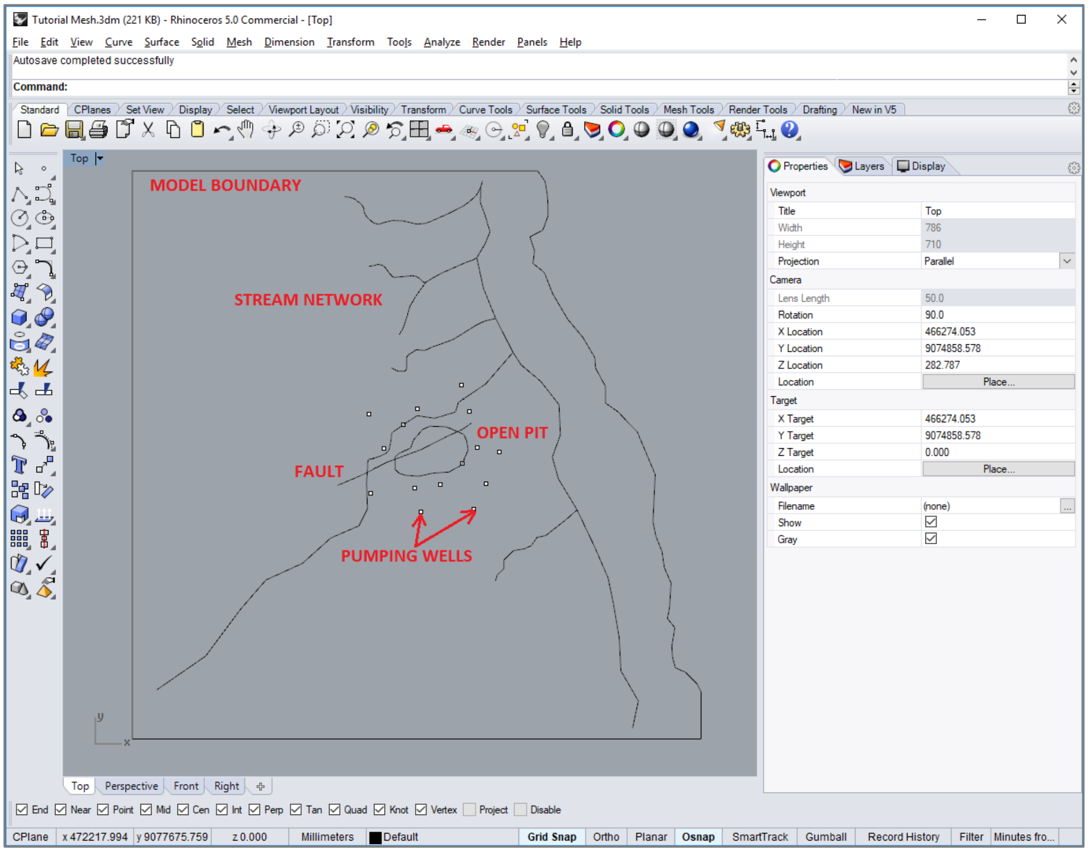


Figure 1.3. The overlay data used to create a mesh

To create the transition zone, click on the "*Polyline*" tool on the "*Standard*" tab (Figure 1.4). 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 1.4.

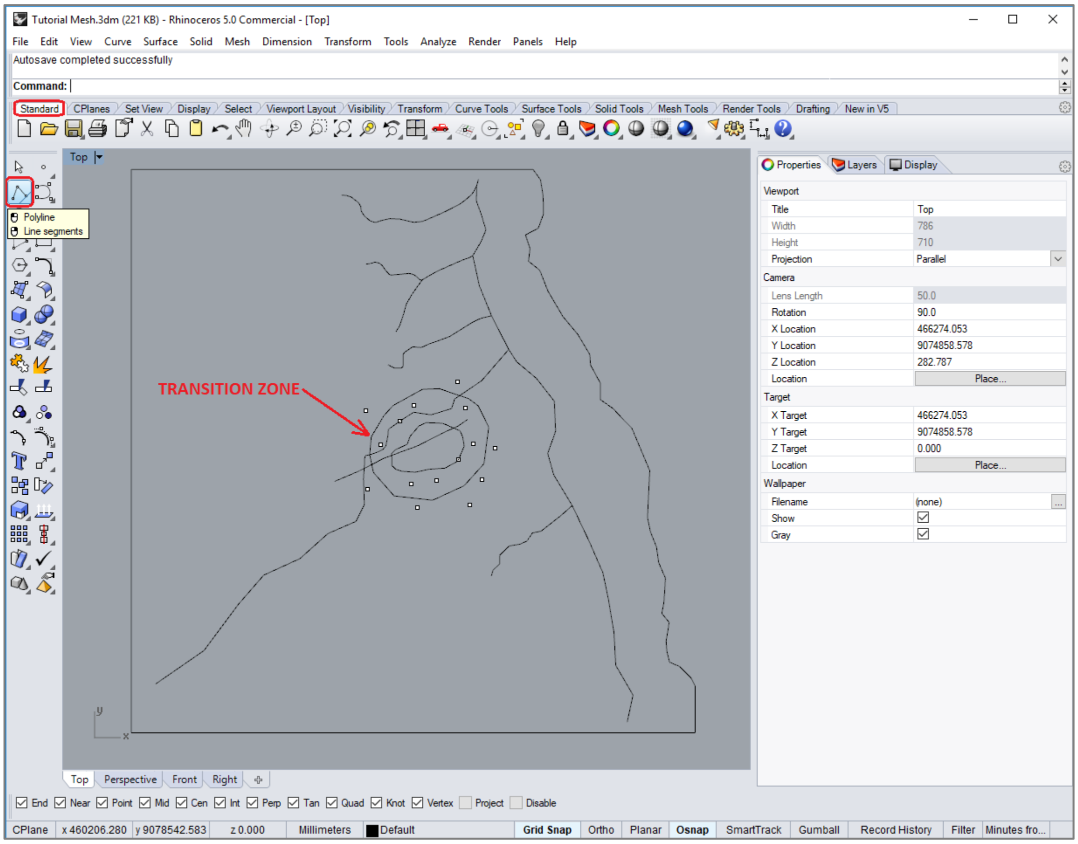


Figure 1.4. The completed transition zone

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 1.5).

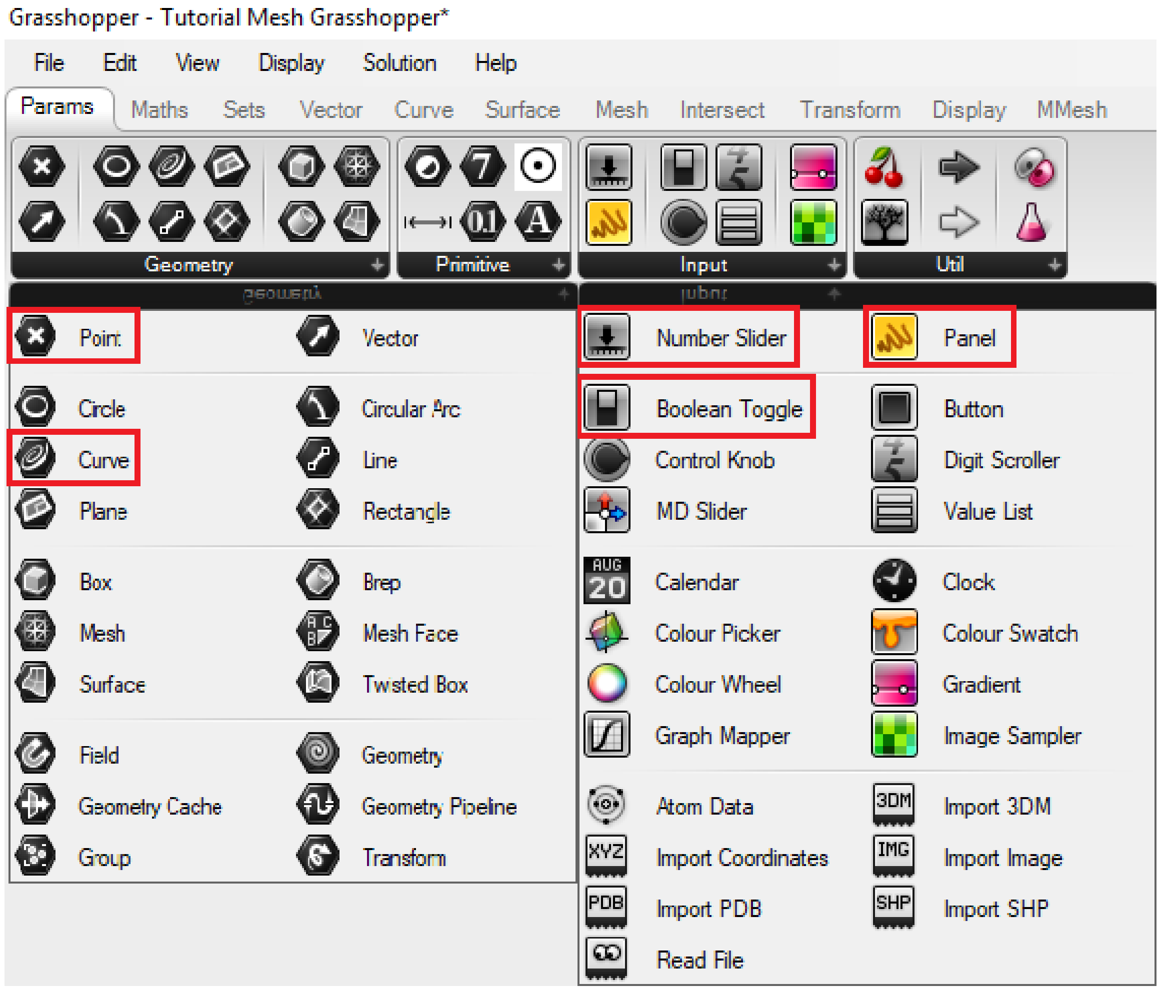


Figure 1.5. *Grasshopper* components used to create the finite-element mesh

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 1.6).

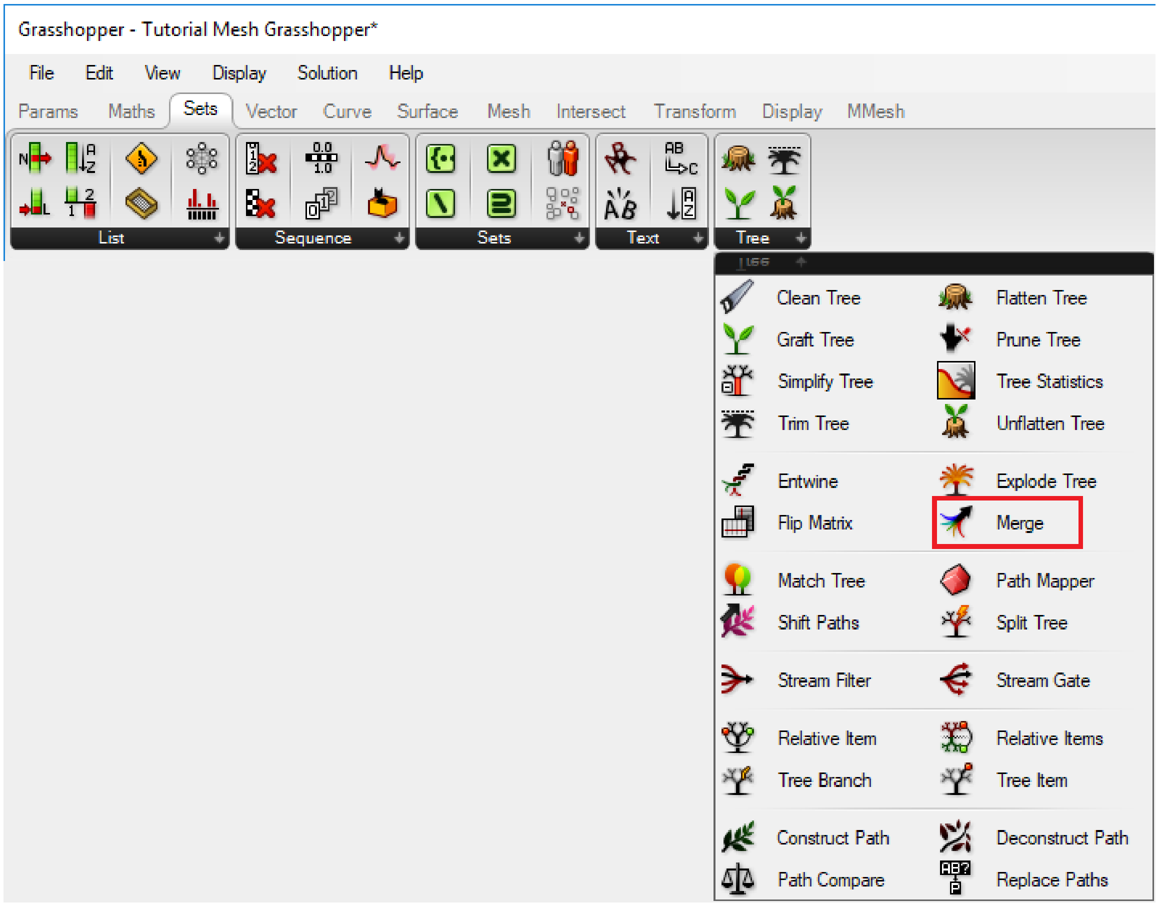


Figure 1.6. The *"Merge"* component in *Grasshopper*

After adding all of the necessary components to the canvas, connect them together following the example in Figure 1.7. 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 1.7 can be used as an example of how to arrange and connect the various components.

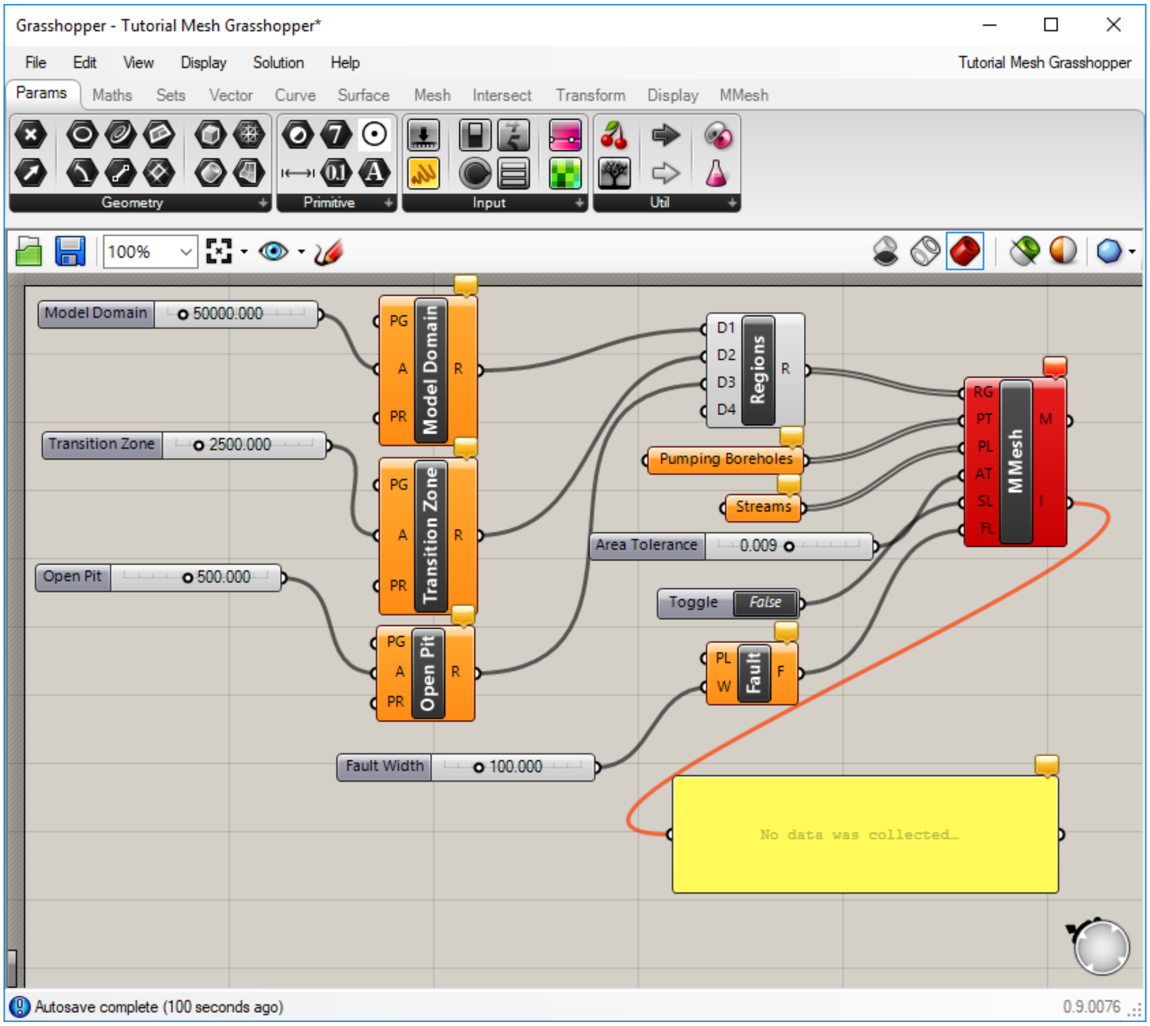


Figure 1.7. Connected *Grasshopper* components and parameters

Once the components are properly connected, it is time to define parameters for each component. For each number slider, add a name and define the range. In this tutorial demonstration, the number sliders have been named *"Model Domain," "Transition Zone," "Open Pit," "Area Tolerance," and "Fault Width."* The ranges for these number sliders are the following, respectively: 25,000–250,000, 1,000–10,000, 100–1,000, 0.001–0.1, and 25–250. To define the initial value, specify the ranges, and name the number sliders, right-click on each number slider and select *"Edit"* in the drop-down list. In the "*Slider*" window, you can enter a *"Name"* and specify the *"Min"* and *"Max"* and set the current value. After completing the changes, click *"OK"* to save the changes and close the *"Slider"* window. The other components can be renamed by right-clicking on the component and entering a new name where the current name appears in the drop-down menu. At this point, it may be advisable to save the *Grasshopper* document so as not to lose any work should anything happen.

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 "s*elpt"* 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.

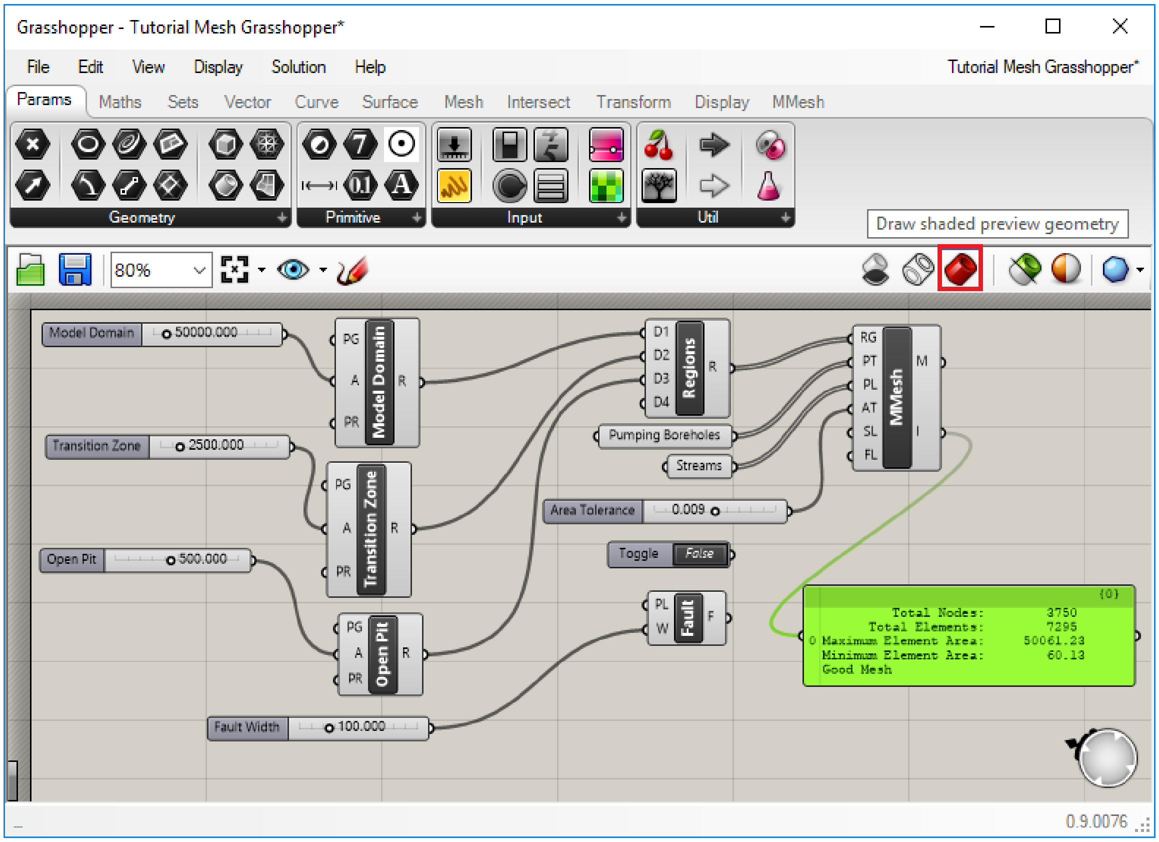


Figure 1.8. The output from the *Grasshopper* algorithm

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 1.8. 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 1.9 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.

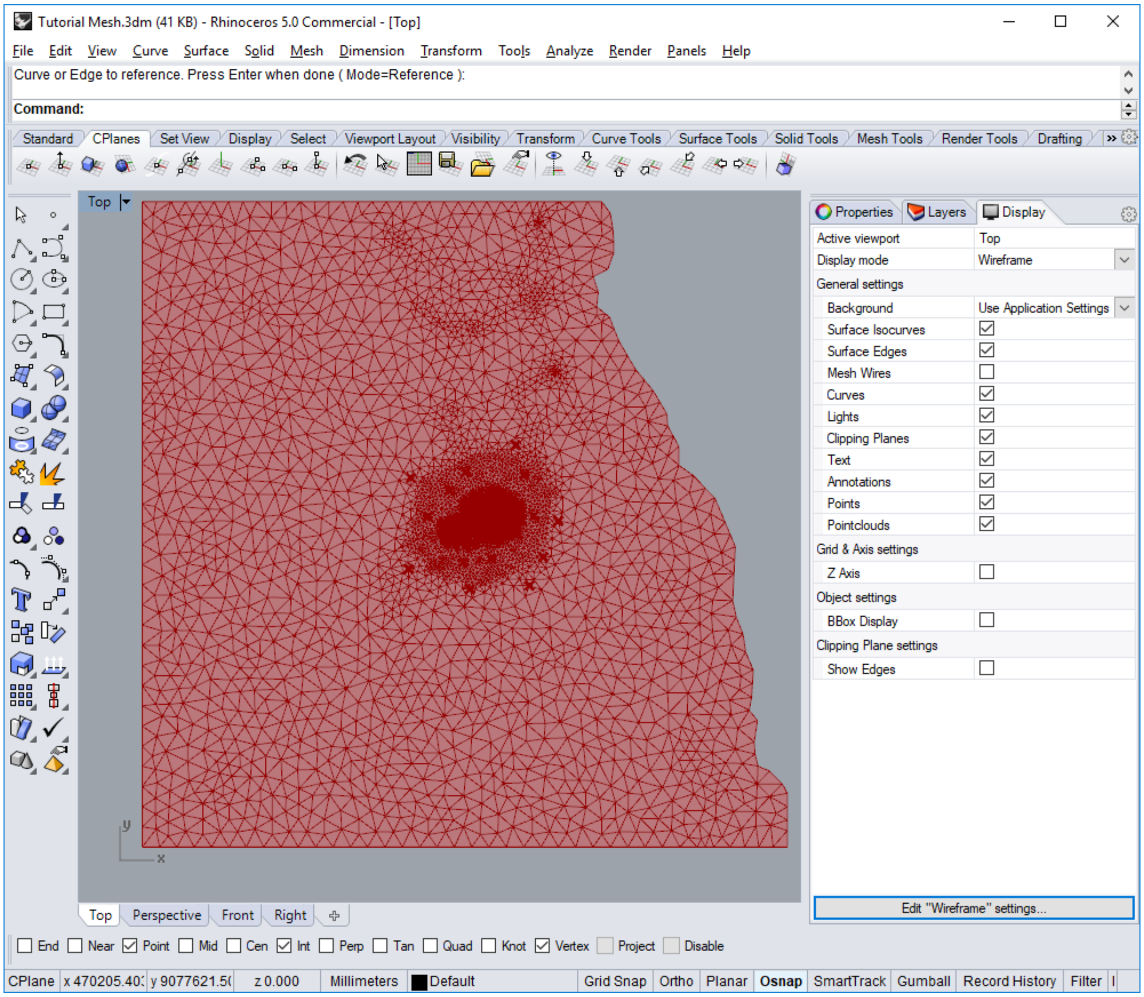


Figure 1.9. The *Grasshopper*-generated mesh

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 1.10.

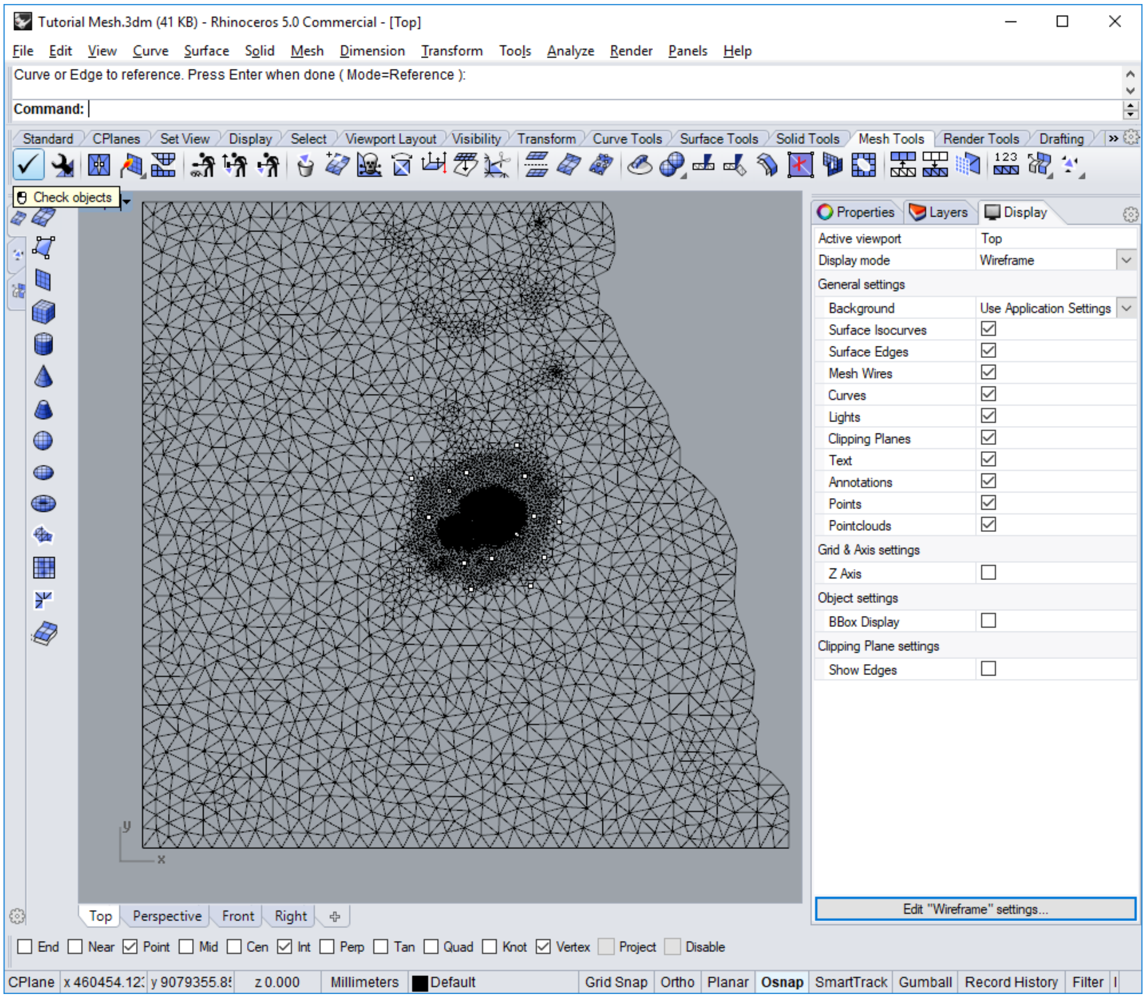


Figure 1.10. The completed mesh in *Rhino*

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 1.10. 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."*

1.3. Building the model in *MINEDW*

Now that the mesh has been created, the mesh is ready to be imported into ***MINEDW***. Open a new ***MINEDW*** window and click "*File*" on the Main Menu banner, and then select "*Import.*" The "*Import File*" dialog box appears (Figure 1.11). 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.*" **<u>IMPORTANT</u>**: No mesh will appear in ***MINEDW*** at this time; a few more steps must be taken before the mesh is displayed.

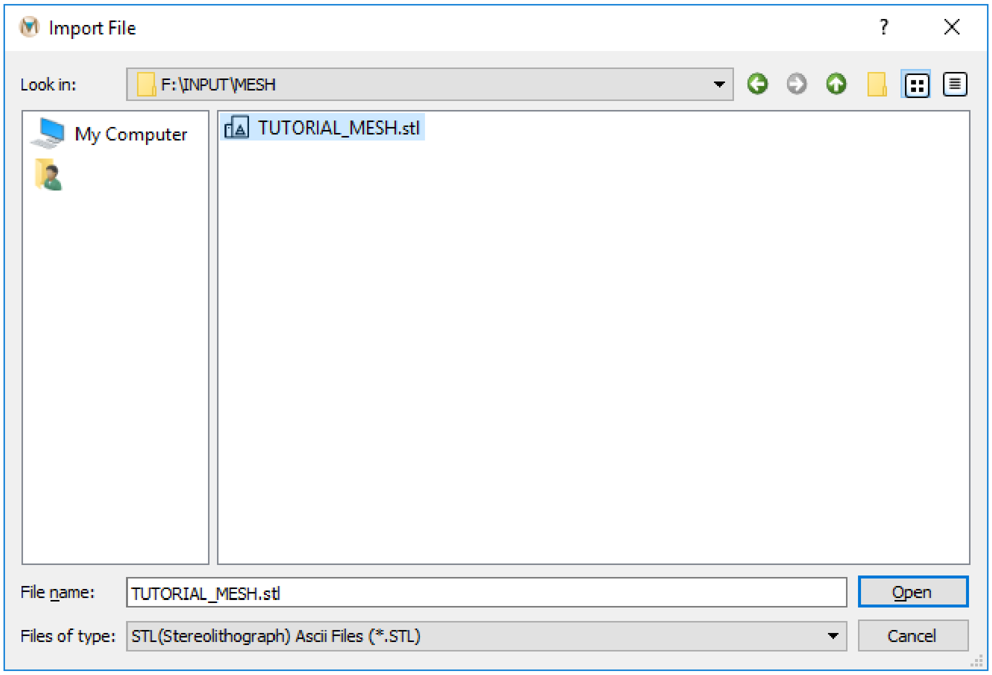


Figure 1.11. The "*Import File*" dialog box

After importing the mesh, save the ***MINEDW*** project file. Select "*File*">"Save as" and, using the *"Create a MINEDW Project File"* dialog box that opens, 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, "*SS-TUTORIAL,"* and then click on *"Save"* to create the project file and close the dialog box.

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 1.9 appears. Make sure that all the fields contain the values shown in Figure 1.12. 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 x 10^{-10}, while the outer closure criterion is likely to be greater than 1 x 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.

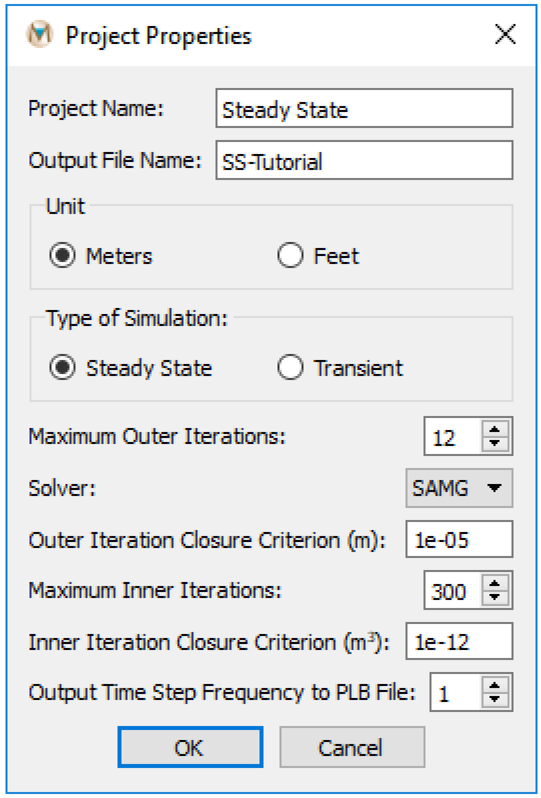


Figure 1.12. The "*Project Properties*" dialog box

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 1.13) appears. Set the "*Maximum Time Steps*" to 100 or greater. Enter "31" for 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 at Time Step*" combo box is set to 1. Click "*OK*" to save the changes and close the *"Set Up Time Step"* dialog box.

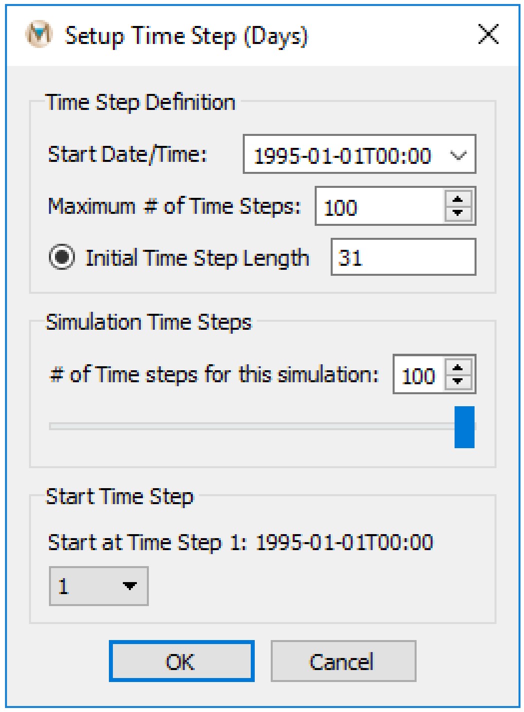


Figure 1.13. The "*Set Up Time Step*" dialog box

Step 4. Importing Ground-Surface Elevation

To incorporate the ground-surface elevation in the model, select the "*List*" tab in the "*Control Panel*" pane on the right-hand side of the ***MINEDW*** interface. Expand the "*Node*" item by double-clicking the "*Node*" item or clicking on the small triangle next to "*Node.*" Next, double-click "*2D Contour*" as shown in Figure 1.14. The mesh will be displayed in the plot pane.

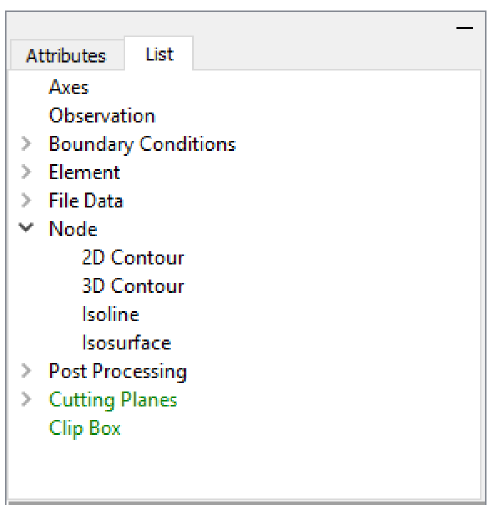


Figure 1.14. The "*Control Panel*" with activated "*List*" tab and "*Node*" plot item

From the "*Control Panel*" pane on the right-hand side of the ***MINEDW*** interface, select the "*Attributes*" tab. Make sure that the *"Color By"* attribute is set to "*Elevation*" and that the

"*Layer*" attribute is set to 1 (Figure 1.15). Next, click the "*Select*" tool on the Main Menu banner.

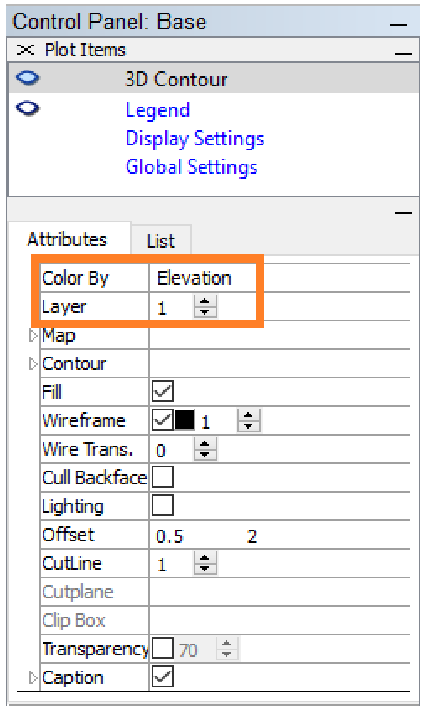


Figure 1.15. The "*Control Panel*" pane showing the "*Attributes*" tab with "*Layer*" set to "1"

In the ***MINEDW*** View Pane, use the cursor to click and drag a box around the whole domain to select all of the nodes in Layer 1. Press the [Enter] key to open the "*Assign Properties for Nodes*" dialog box shown in Figure 1.16. 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 1.17 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.

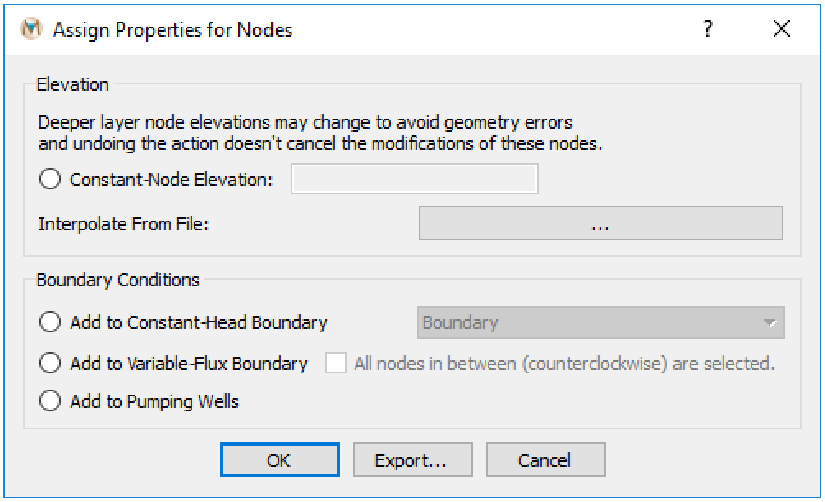


Figure 1.16. The "*Assign Properties for Nodes*" operations dialog box

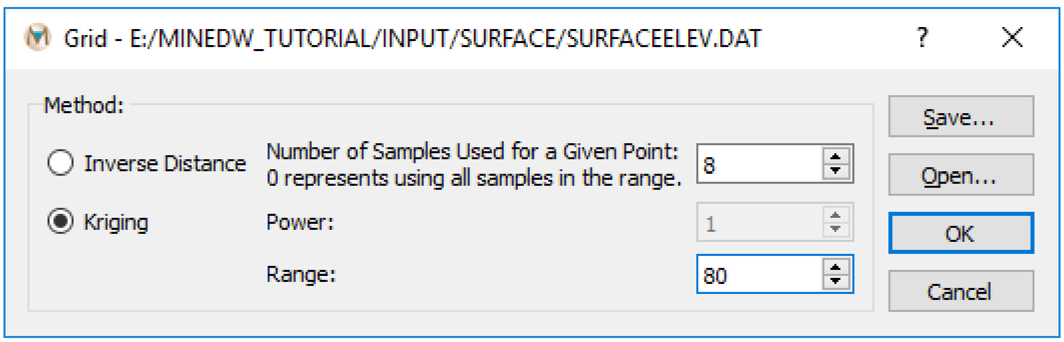


Figure 1.17. The "*Grid*" dialog box and example values for surface interpolation

View the ground-surface elevation as contours by clicking on the "*Attributes*" tab in the "*Control 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 1.18.

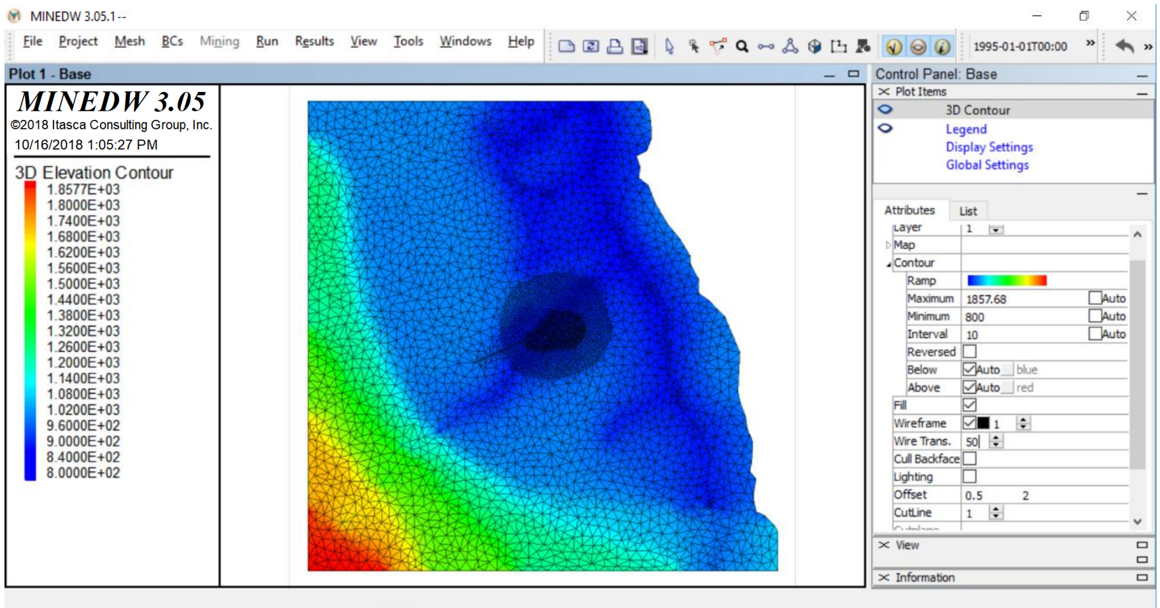


Figure 1.18. ***MINEDW*** **color flood of topography**

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 1.19) 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 1.20). 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 (Figure 1.20). 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.

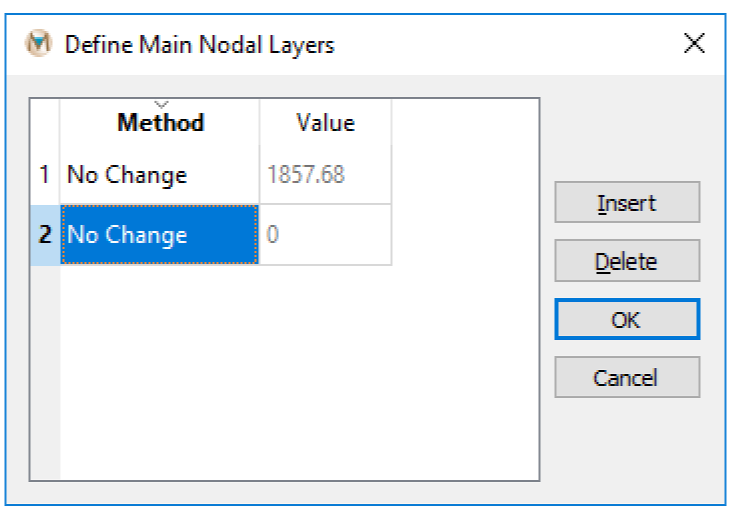


Figure 1.19. The "*Define Main Nodal Layers*" dialog box

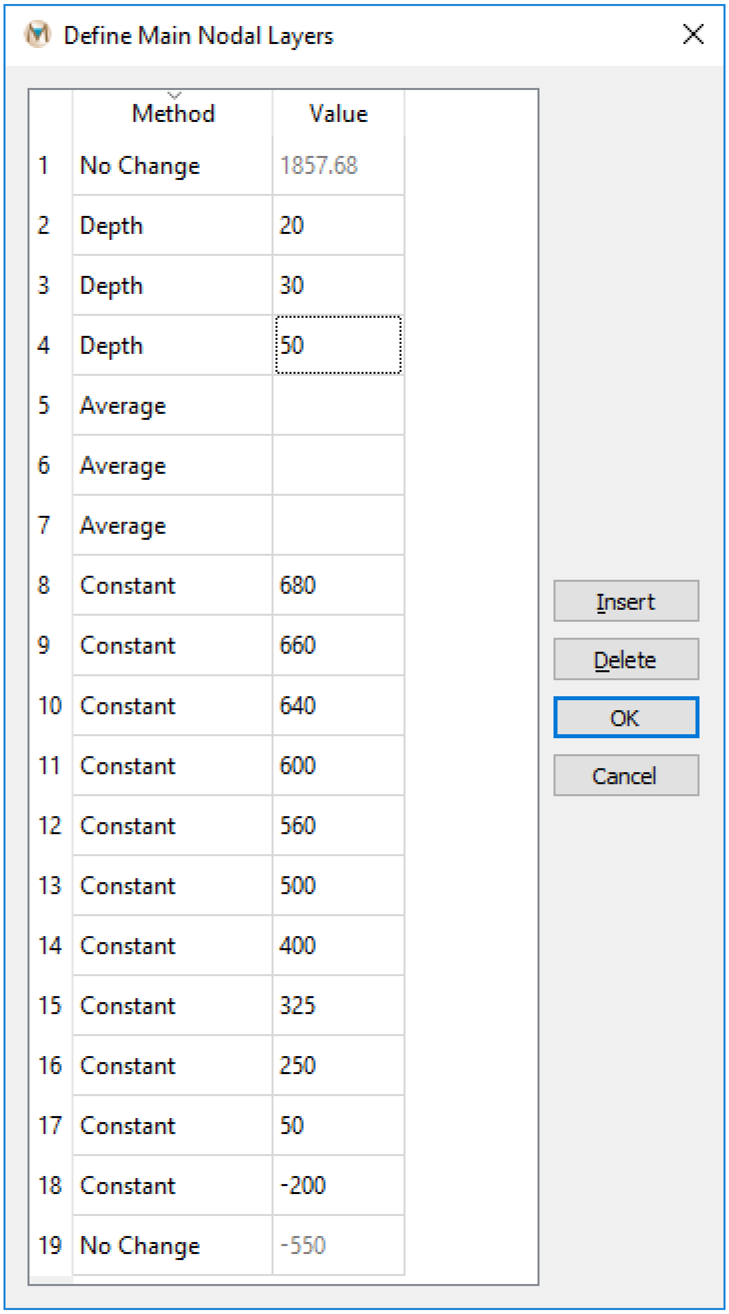
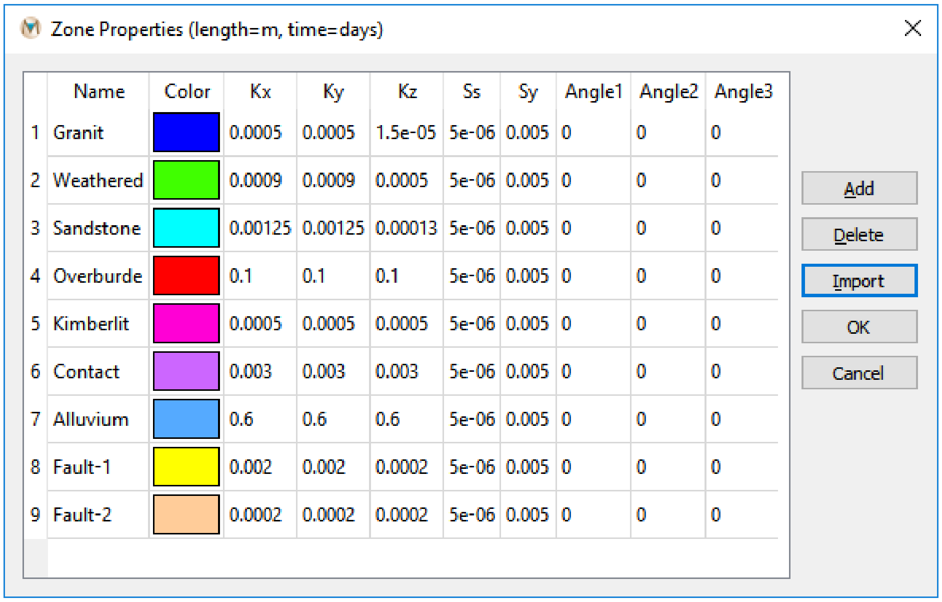


Figure 1.20. The "*Define Main Nodal Layers*" dialog box showing different methods available

Step 6. Defining Zone Properties

To define the properties of a hydraulic zone, click "*Project*" on the Main Menu banner and then select "*Zone Properties.*" The "*Zone Properties*" dialog box (Figure 1.21) appears. To add a new zone, click the "*Add*" button. Enter the zones and related hydraulic properties as listed in Figure 1.21 and then click "*OK.*"



Zone Properties (length=m, time=days)

	Name	Color	Kx	Ky	Kz	Ss	Sy	Angle1	Angle2	Angle3
1	Granit		0.0005	0.0005	1.5e-05	5e-06	0.005	0	0	0
2	Weathered		0.0009	0.0009	0.0005	5e-06	0.005	0	0	0
3	Sandstone		0.00125	0.00125	0.00013	5e-06	0.005	0	0	0
4	Overburde		0.1	0.1	0.1	5e-06	0.005	0	0	0
5	Kimberlit		0.0005	0.0005	0.0005	5e-06	0.005	0	0	0
6	Contact		0.003	0.003	0.003	5e-06	0.005	0	0	0
7	Alluvium		0.6	0.6	0.6	5e-06	0.005	0	0	0
8	Fault-1		0.002	0.002	0.0002	5e-06	0.005	0	0	0
9	Fault-2		0.0002	0.0002	0.0002	5e-06	0.005	0	0	0

Figure 1.21. The "Zone Properties" dialog box

Step 7. Assigning Hydraulic Zones to Elements

From the ***MINEDW*** main window, select the "*List*" tab in the "*Control Panel*" pane. Expand the "*Element*" item and double-click "*3D Element*," as shown in Figure 1.22. 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).

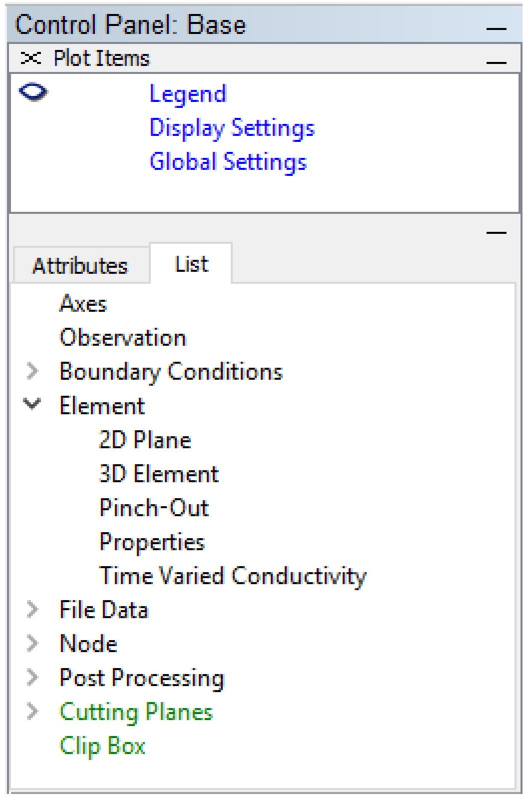


Figure 1.22. The "*Control Panel*" with the *"List"* tab and *"Element"* plot item activated

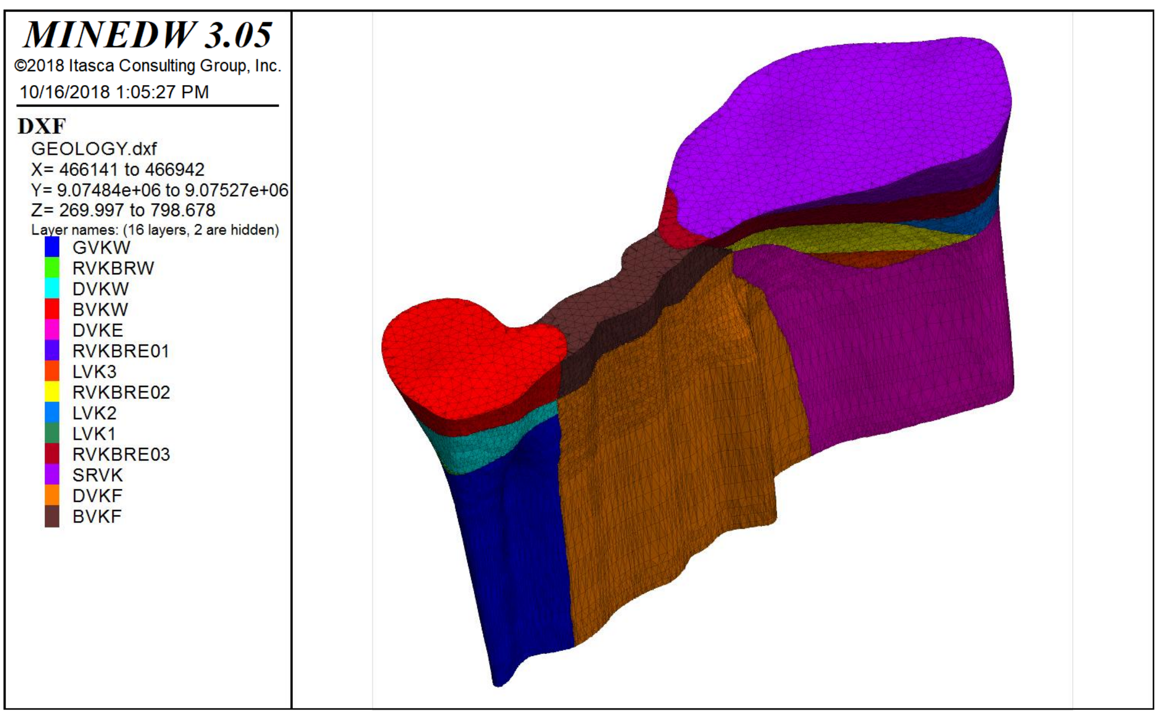


Figure 1.23. A wireframe format of a geological block model

In ***MINEDW***, the assignment of hydraulic zones can be done using a 3-D .DXF with a wireframe format of a geological block model (Figure 1.23). 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 1.24 under the "*Control Panel*" pane is set to 1.

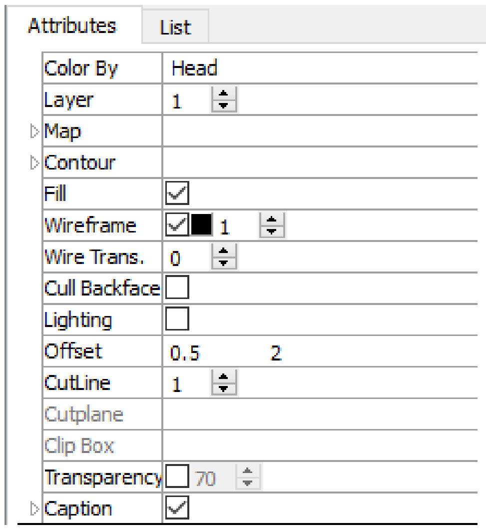


Figure 1.24. The "*Attributes*" tab with "*Layer*" set to 1

Select all the elements on the first layer by using the cursor to click and drag a selection box as shown in Figure 1.25, then press the [Enter] key.

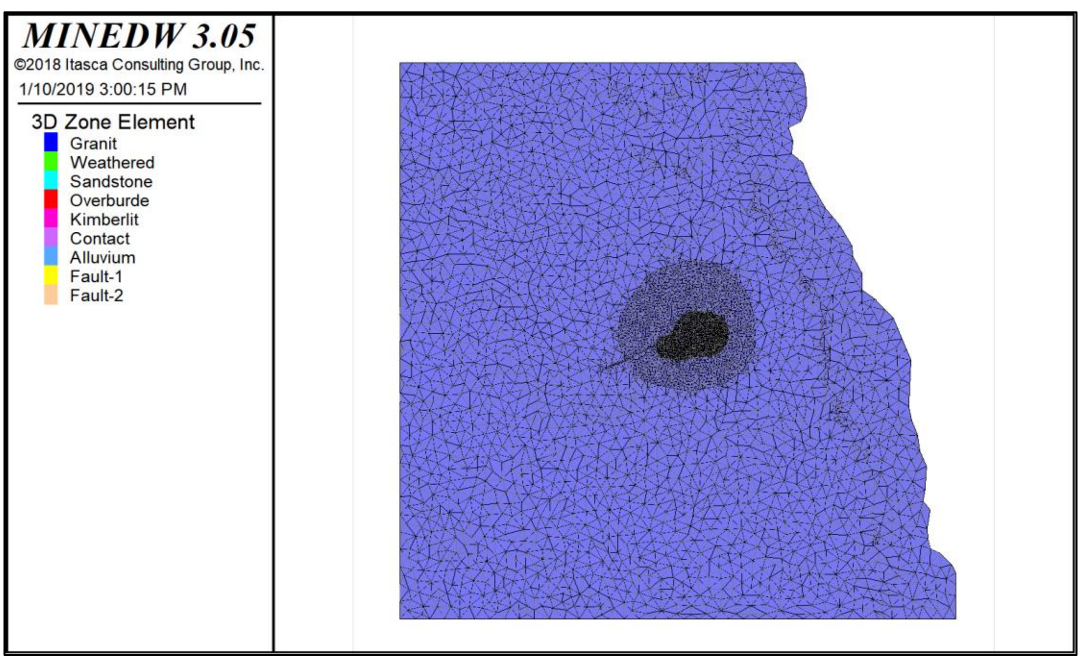


Figure 1.25. Selection of first layer of elements

The “*Select Geological Zone:*” dialog box shown in Figure 1.26 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.*”

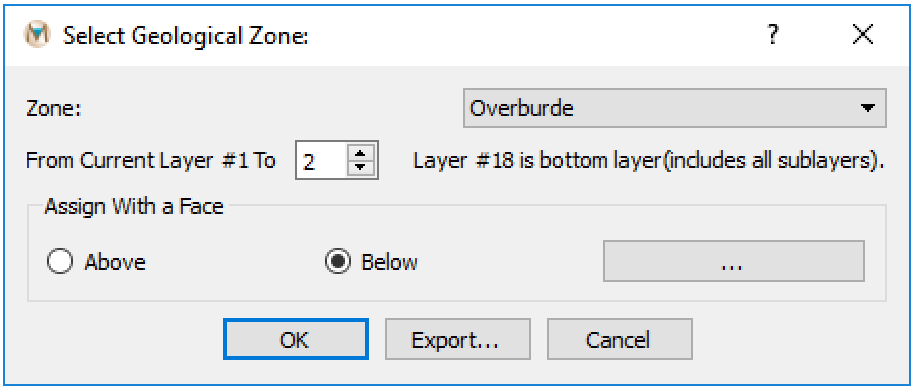


Figure 1.26. The “*Select Geological Zone:*” operations dialog box

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 Geological Zone:*” 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 Geological Zone:*” 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 an ESRI shapefile. The shapefile was created by using the buffer tool in ArcMap and the River.BLN files used in the Mesh Generator. To select elements in the river channel, click the “*List”* tab and then click the arrow next to “*File Data.”* Double-click *“ESRI Shape File”* and then on the *“Attributes”* tab click the plus sign. Browse to the *“STREAM_ALLUVIUM.SHP”* file and select it, then click “*Open.”* If you do not see the polygon buffer representing the river, ensure the *“Project to Surface”* box is checked. Make sure that “*Layer*” under the “*Attributes*” tab under the “*Control Panel*” pane is set to 1.

Next, click the *“Select With Overlay”* tool and click the “*Polygon”* radio button, then click “*OK.”* The elements that form the river channel should now be selected; press [Enter]. Change the “*Zone”* combo box to “*Alluvium”* and change the “*To Layer”* value to 3 (Figure 1.27), then click “*OK.”*

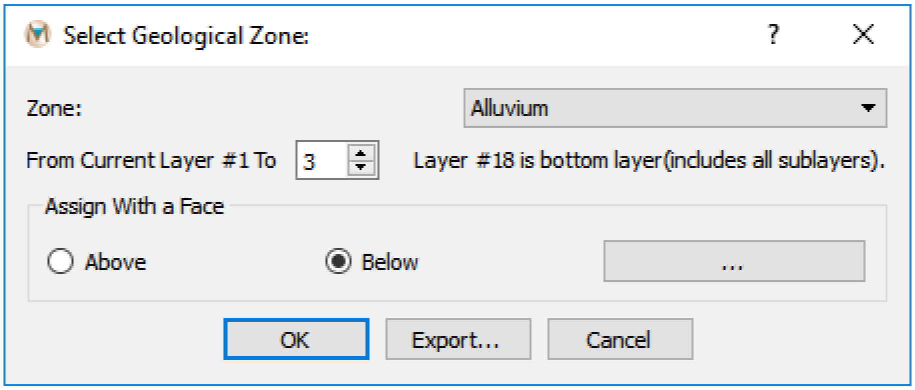


Figure 1.27. The "*Select Geological Zone*" dialog box

To assign the "*Kimberlite*" and *"Contact"* zones using a 3-D .DXF, add a *"DXF"* plot item from the "*Control Panel*" pane by choosing "*List*" then "*File Data.*" Expand "*File Data*" by clicking on the small triangle next to "*File Data.*" Under "*File Data,*" double-click on "*DXF.*" The "*Attributes*" tab will appear (Figure 1.28). Click on the "+" next to "*File.*" The "*Select DXF data file*" window will appear. Select the "*GEOLOGY.DXF*" file. All of the geologic units are present in this 3-D .DXF file. In order to view just the geologic units of interest, select the *"Attributes"* tab, then click the triangle next to *"Layers."* Uncheck the boxes next to *"OVB"* and *"GNS."* The remaining geologic units will be used to represent the contact zone and the kimberlite pipe.

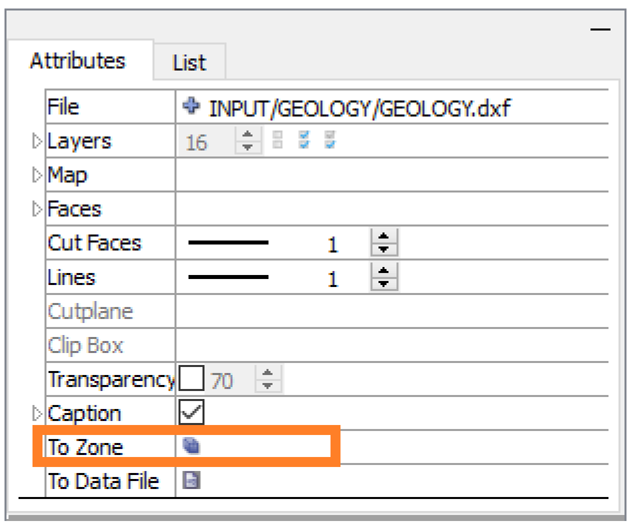


Figure 1.28. The "*Attributes*" tab

To define the contact zone, click on the *"To Zone"* button (Figure 1.28) and then select *"Contact"* from the combo box (Figure 1.29). Then check "*Face*" in the box next to *"Layer Thickness,"* enter 100, then click *"OK."* The process may take some time, but the area will be assigned to the contact zone. **Note:** The contact zone may need to be adjusted manually by selecting elements with the *"Select"* tool. To assign elements with "*Kimberlite*" properties, repeat the steps above but select *"Kimberlite"* instead of *"Contact"* and check "*Volume*" and "*Whole.*" The volume representing the kimberlite pipe will need to be adjusted manually by adjusting element properties individually; see Figure 1.30 for a reference of what the final kimberlite and contact zones look like.

Figure 1.30 shows the two fault zones that intersect the kimberlite pipe. The fault zones are assigned using the "*Select with Overlay"* tool. To do this, click on the *"List"* tab in the "*Control Panel"* pane, click *"File Data,"* and then double-click "*BLN."* Next, click on the button next to *"File"* on the *"Attributes"* tab and navigate to the "*FAULT.BLN*" file. The "*FAULT.BLN*" should be projected to the surface automatically, making it visible. If it is not visible when opened, check the *"Project to Surface"* box on the attributes tab. If the "*RIVER.SHP"* file is still visible in the View Pane or *"Plot Items"* pane, remove it before proceeding. Make sure that "*Layer*" under the "*Attributes*" tab under the "*Control Panel*" pane is set to 1. Next, click on the "*Select*" tool if it is not already selected and click the "*Select with Overlay"* button. Then check the button next to *"Polyline"* and click *"OK."* Fault-zone elements should now be selected. Press the [Enter] key to bring up the *"Select Geological Zone:"* dialog box. In the box next to *"From Current Layer #1 to,"* enter "18," which is the bottom, and select "*Fault 1,"* then click *"OK."* The section of the fault that intersects the contact and kimberlite zones will need to be selected by hand and assigned to *"Fault 2"* (Figure 1.30).

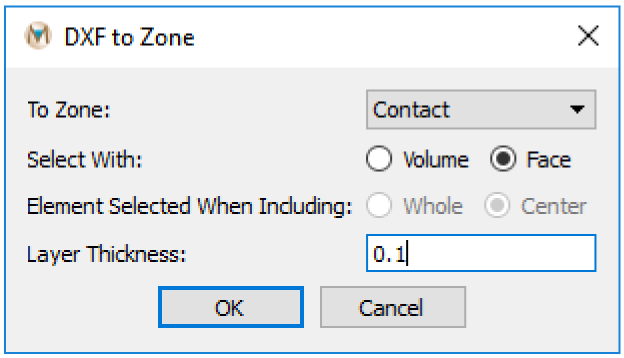


Figure 1.29. The "*DXF to Zone*" dialog box

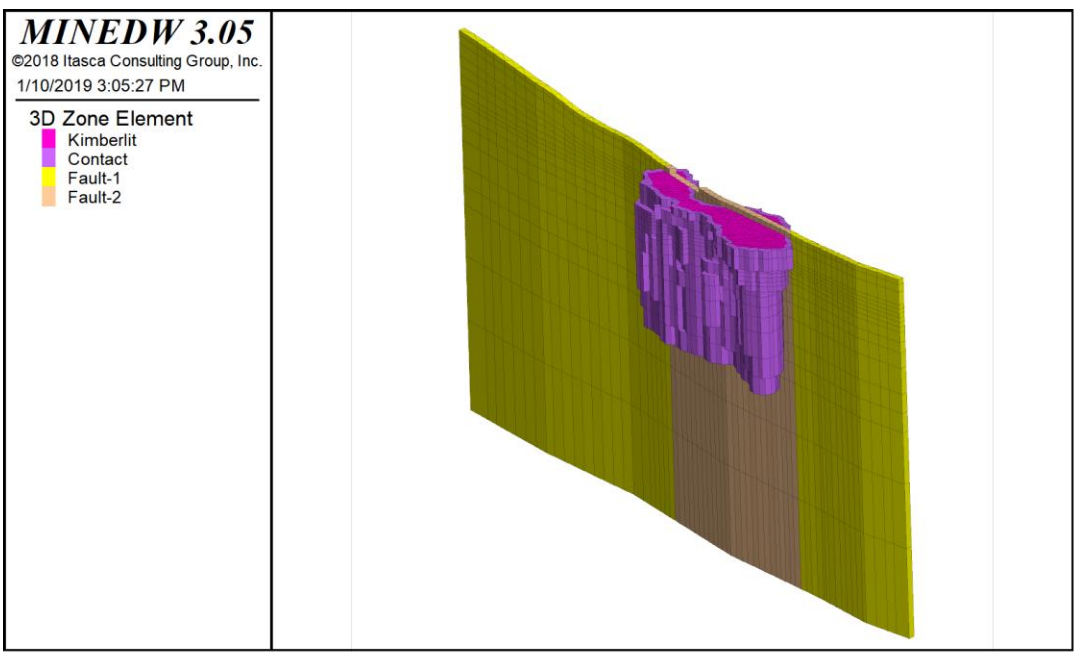


Figure 1.30. The completed contact zone, kimberlite pipe, and fault zones

For a detailed explanation of how to cut cross sections, please refer to the ***MINEDW*** User Manual section 9.12. Briefly, to create a cross section, add a "*3D Element*" plot item to the View Pane, then select the "*Create Plane…"* tool on the menu banner. Select two points on the *"3D Element"* plot item, and the cross section will appear. Rotate the cross section manually or click the "*Snap View"* button on the *"Attributes"* tab of the *"Plane"* plot item to better orient the cross section. To create a cross section similar to the one displayed in Figure 1.31, click on the "*3D Element*" plot item, then select its "*Attributes*" tab and locate the "*Cutplane*" attribute. Check the boxes next to "O*n*" and "*Front.*"

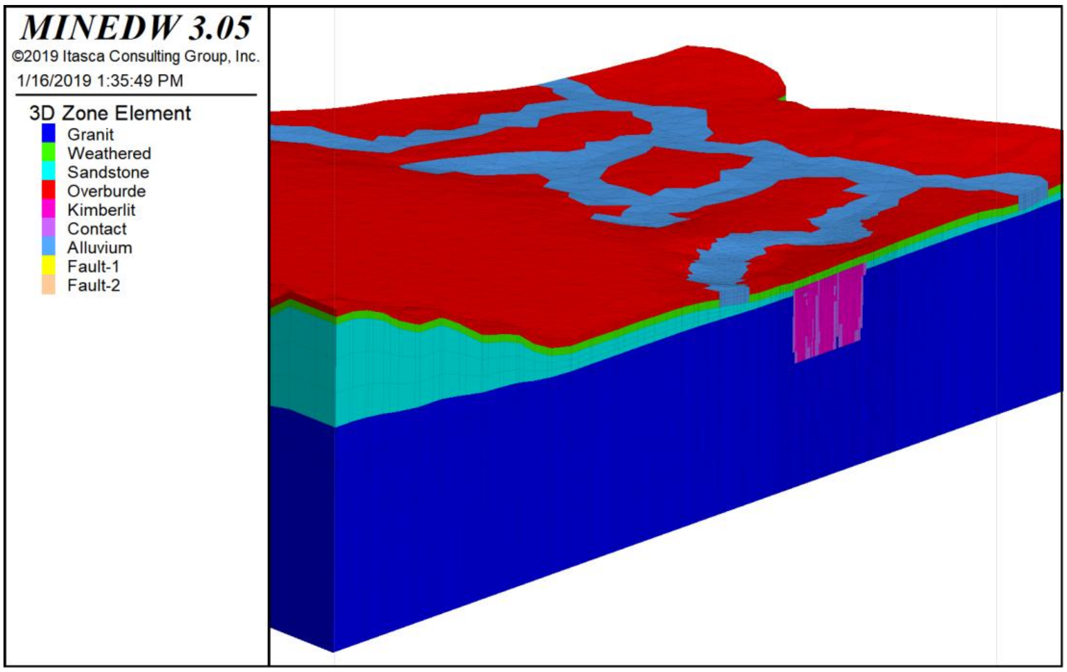


Figure 1.31. Cross section of the geological units in the model

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 1.32).

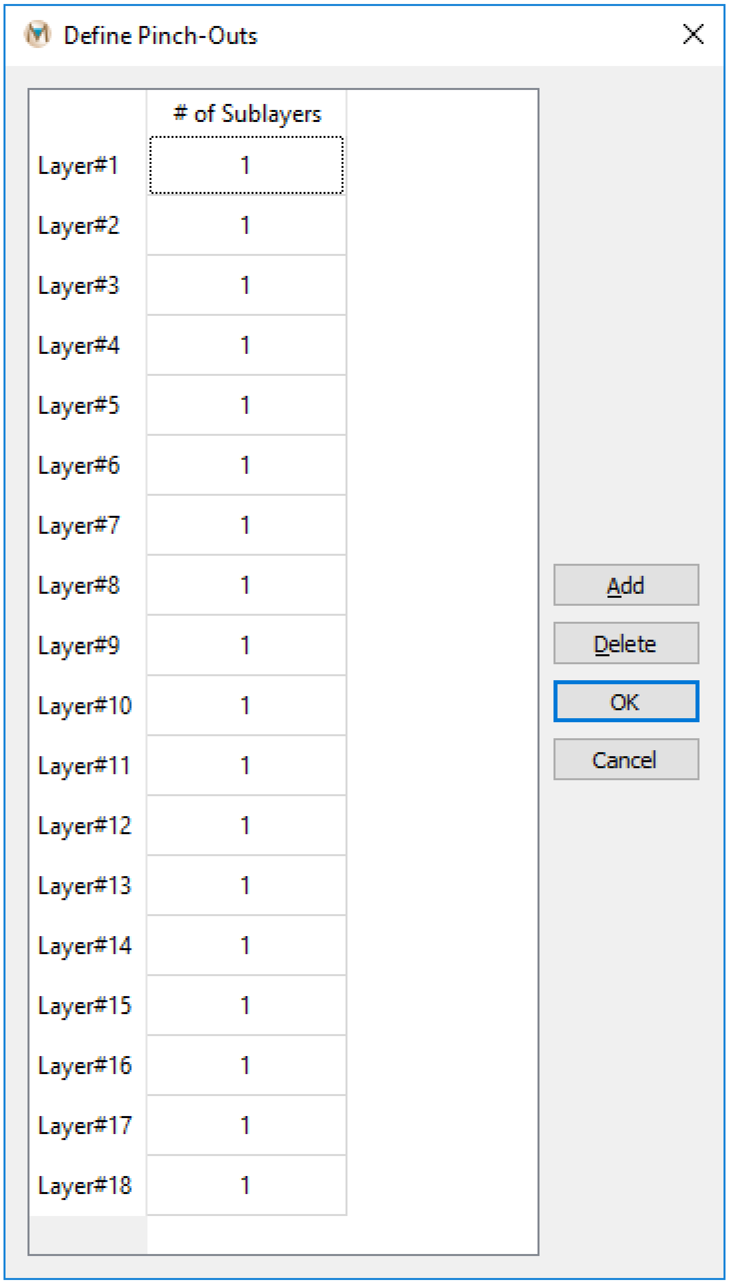


Figure 1.32. The "*Define Pinch-Outs*" dialog box

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 1.33 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 1.33). 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).

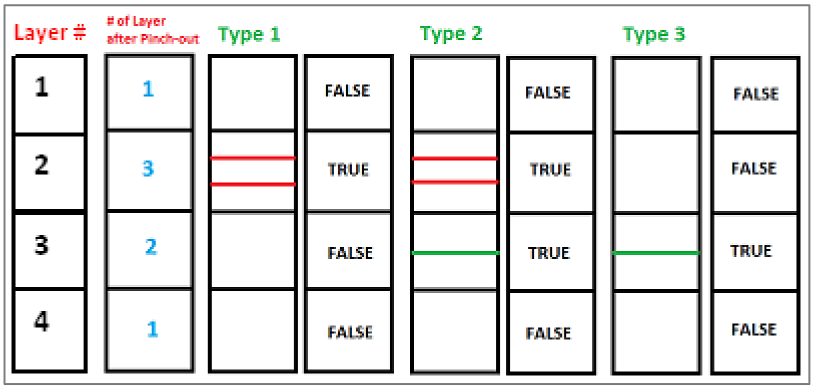


Figure 1.33. Schematic explanation of pinch-outs

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 1.34), then click "*OK.*"

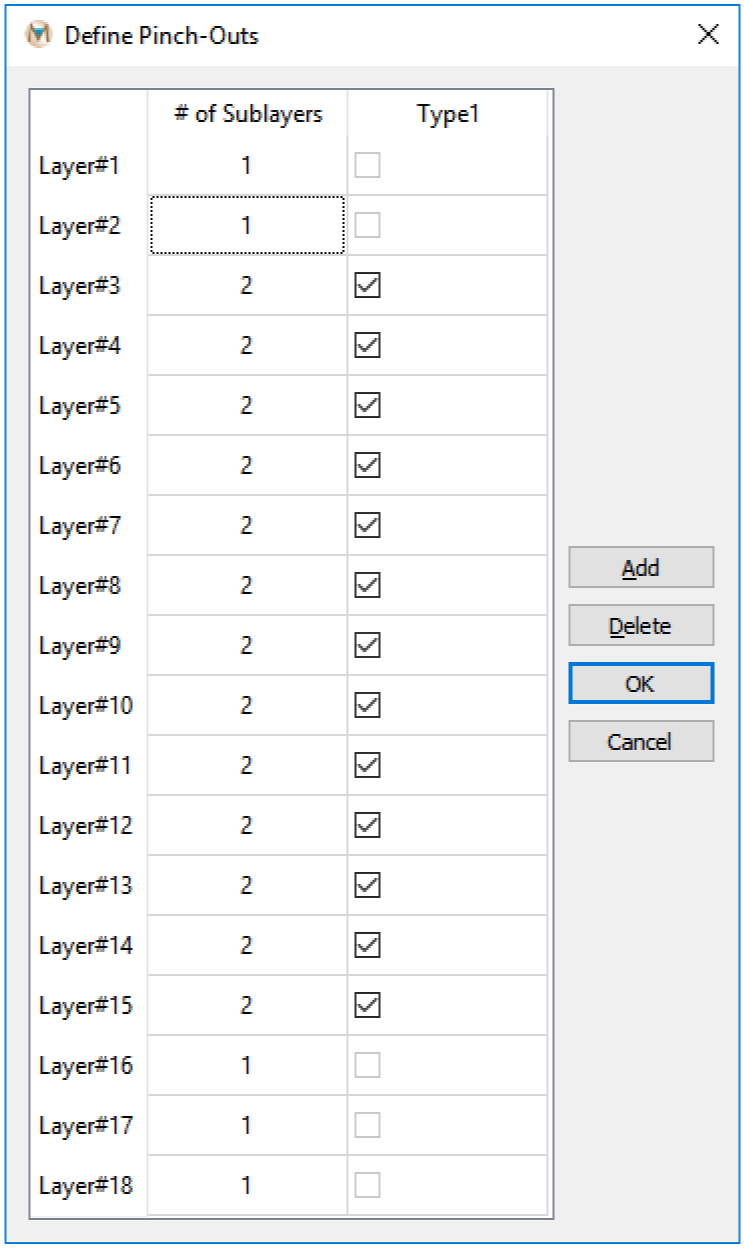


Figure 1.34. The "*Define Pinch-Outs*" dialog box

To add pinch-outs to a selected area, select the "*List*" tab in the "*Control Panel*" pane. Expand the "*Element*" item, and double-click "*Pinch-Out*," as shown in Figure 1.35.

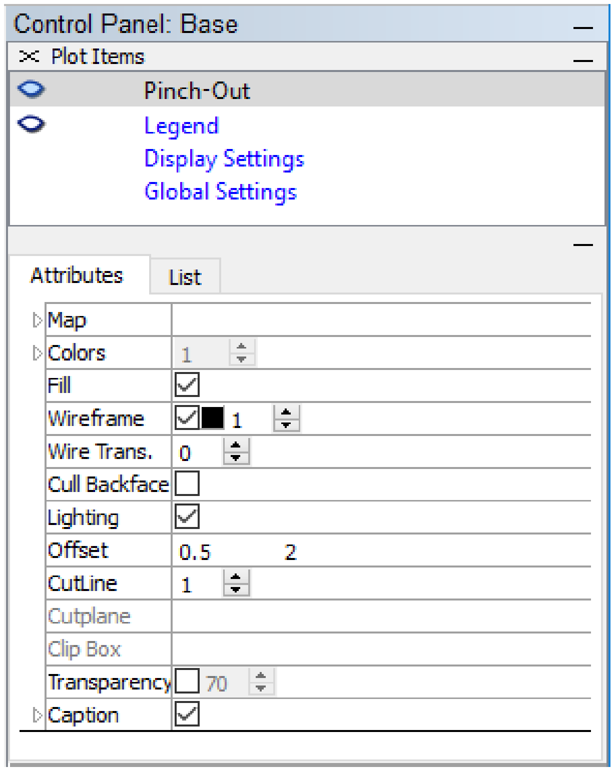


Figure 1.35. The "*Pinch-Out*" plot item "*Attributes*"

From the main window, click the "*Select*" button on the Main Menu banner. Select the nodes that define the elements where the "*Type 1*" pinch-out will be created by clicking the "*Select with Polygon*" button and selecting the nodes as shown in Figure 1.36.

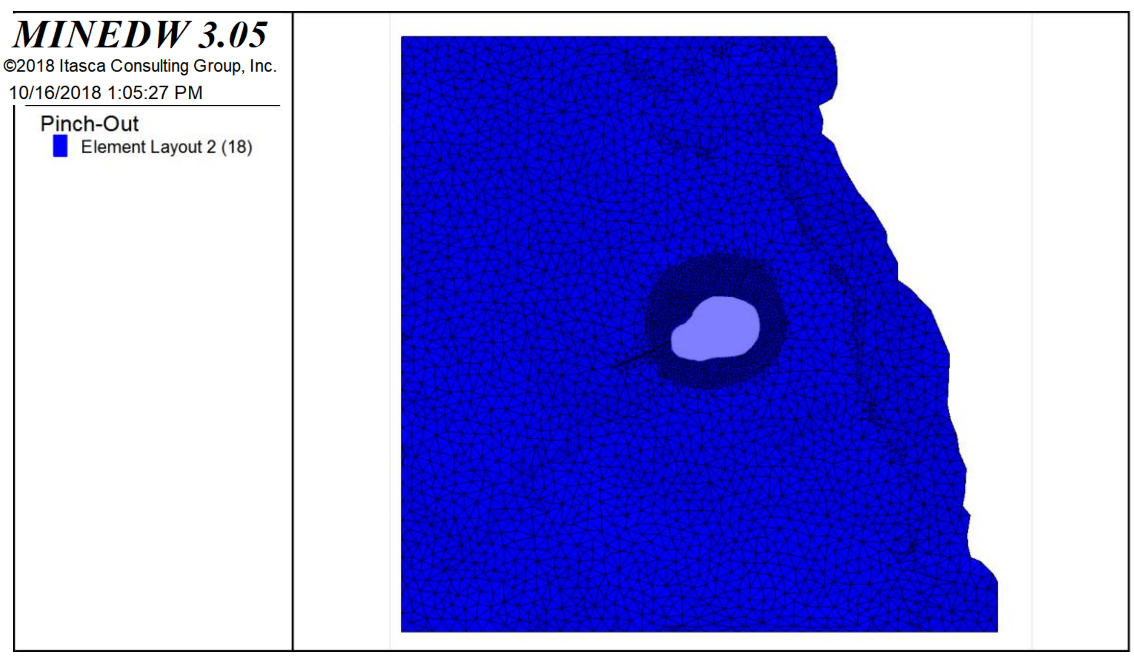


Figure 1.36. The View Pane with the selected nodes for pinch-out Type 1

Press the [Enter] key. The "*Select Pinch-Out Type*" dialog box appears. Select "*Type 1*" (Figure 1.37) and click "*OK.*"

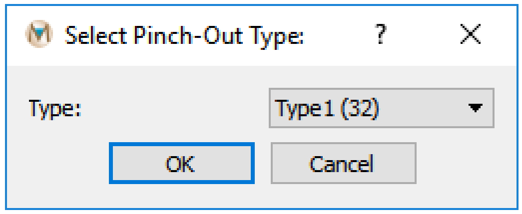


Figure 1.37. The "*Select Pinch-Out Type*" dialog box showing "*Type 1*" selected

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 1.38.

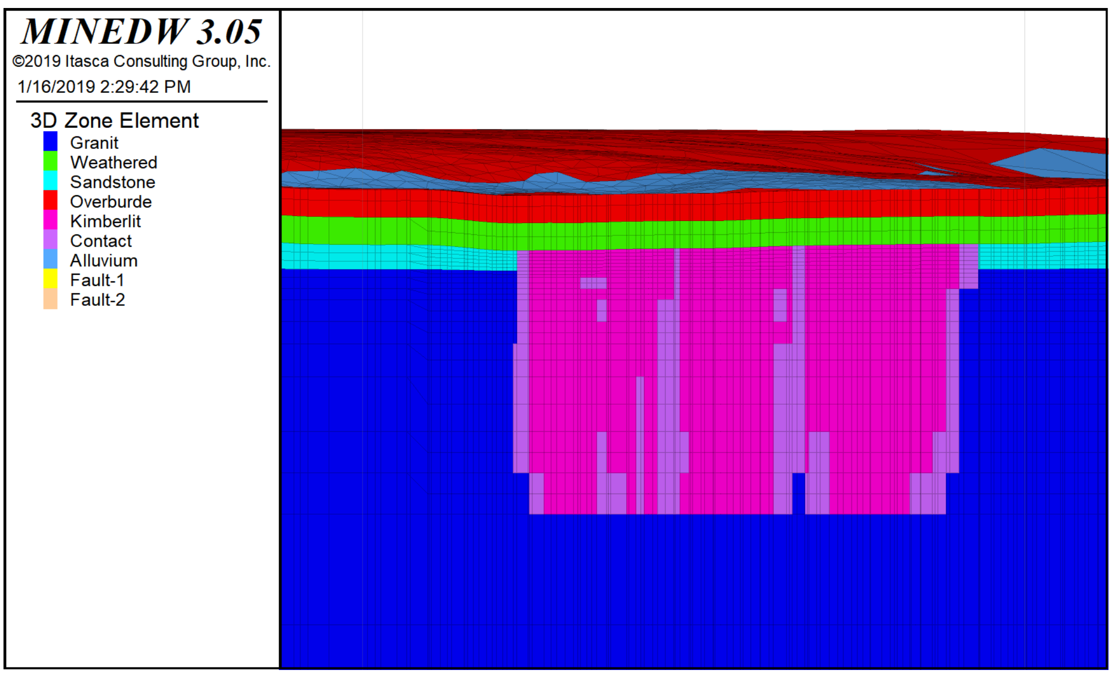


Figure 1.38. The View Pane with the cross-sectional view with pinch-out Type 1

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 1.39.

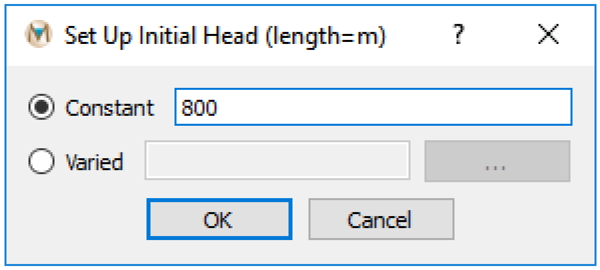


Figure 1.39. The "*Set Up Initial Head*" dialog box

Step 10. Assigning the Boundary Conditions

To assign boundary conditions, select the "*List*" tab in the "*Control Panel*" pane. Expand the "*Node*" item, and then double-click "*3D Contour*" (Figure 1.40).

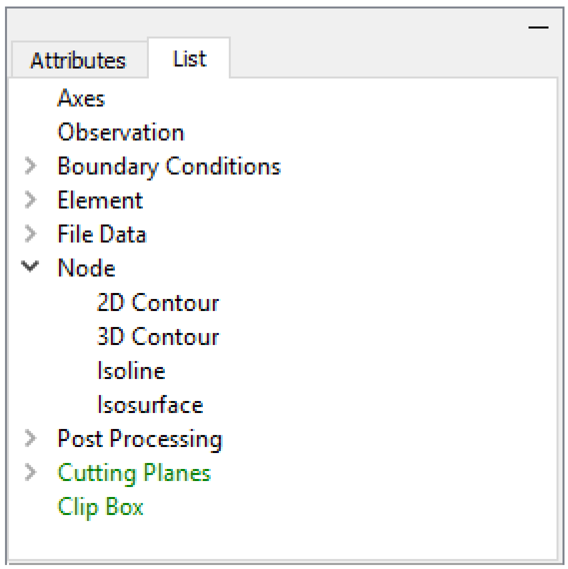


Figure 1.40. The "*Control Panel*" pane with the "*Node*" item expanded to show the "*3D Contour*" option

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.

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 1.41). To change the name of the group, click in the drop-down box where the group name appears and change the text (Figure 1.41). 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.

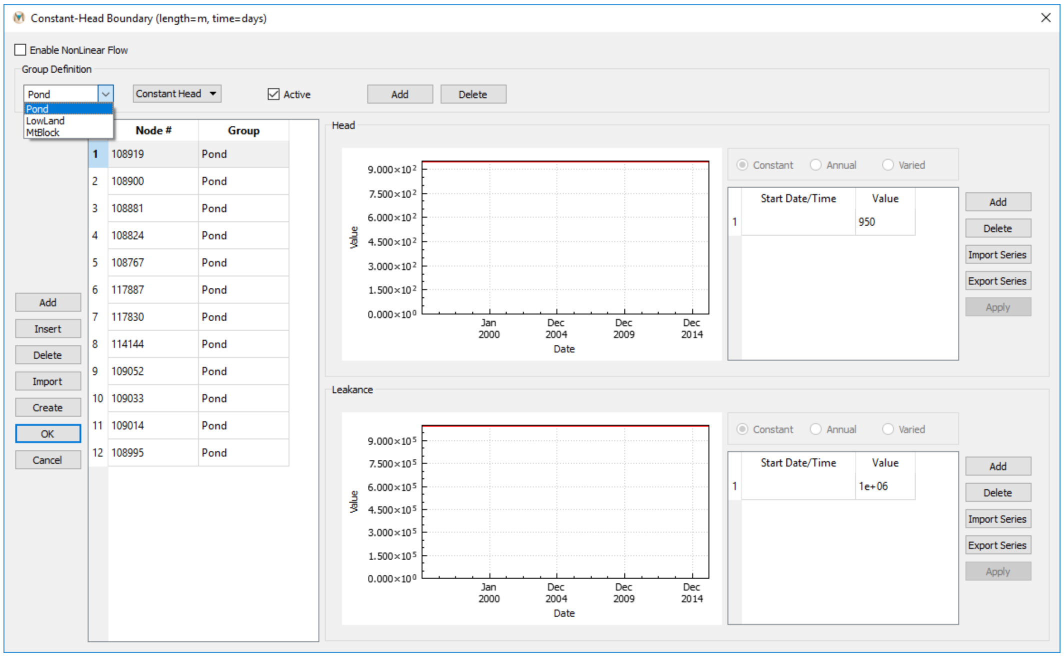


Figure 1.41. The "*Define Group for Constant Heads & Drain Nodes*" dialog box

To add constant-head boundary conditions to the mesh, select the *"3D Contour"* plot item from the *"Plot Items"* list in the *"Control Panel"* pane. Ensure that the *"Layer"* attribute under the *"Attributes"* tab is set to 5, then click the "*Select*" button on the Main Menu banner. Using the "*Select*" tool, select the nodes along the southwest perimeter as shown in Figure 1.42 and then press [Enter]. In the "*Assign Properties for Nodes*" dialog box that appears, select "*Add to Constant Head Boundary*" (Figure 1.43). 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.

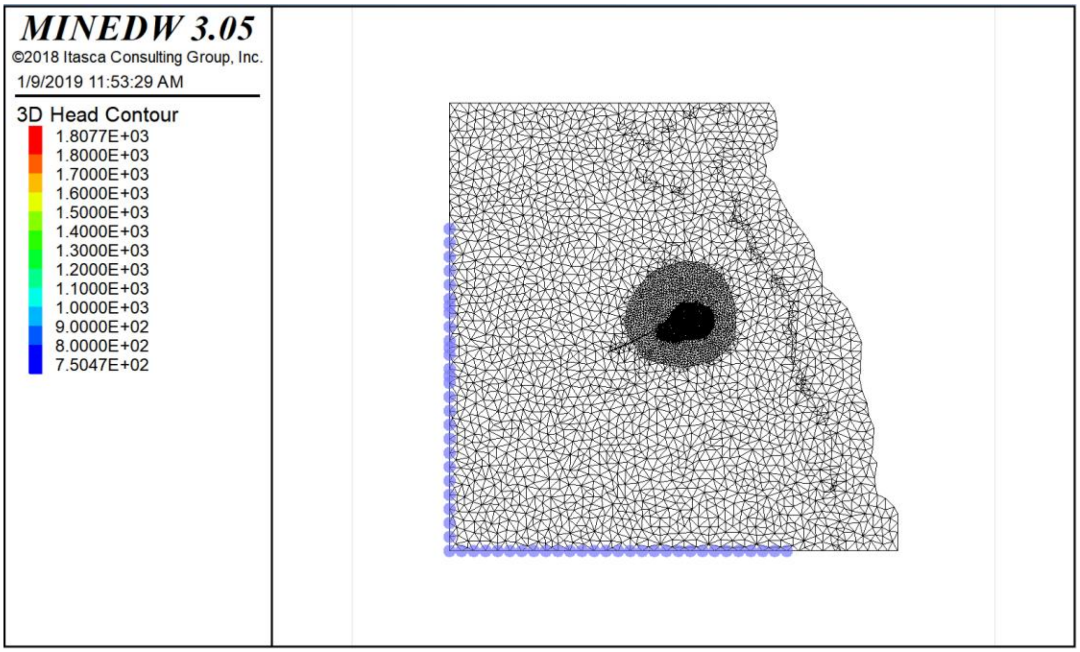


Figure 1.42. The View Pane with the selected constant head boundary nodes

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.

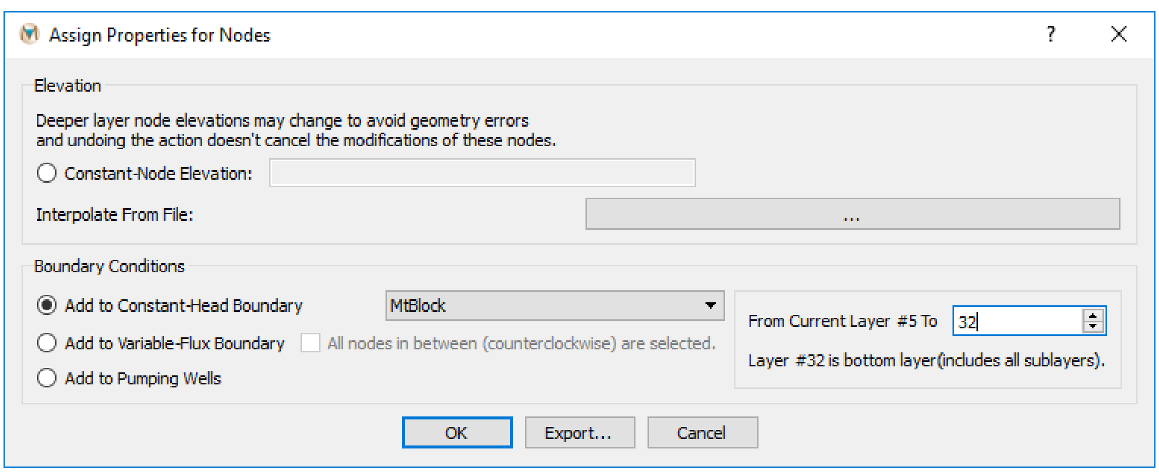


Figure 1.43. The "*Assign Properties for Nodes*" dialog box

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 *"nodel.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 the 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 chead 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 1.45). 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.*"

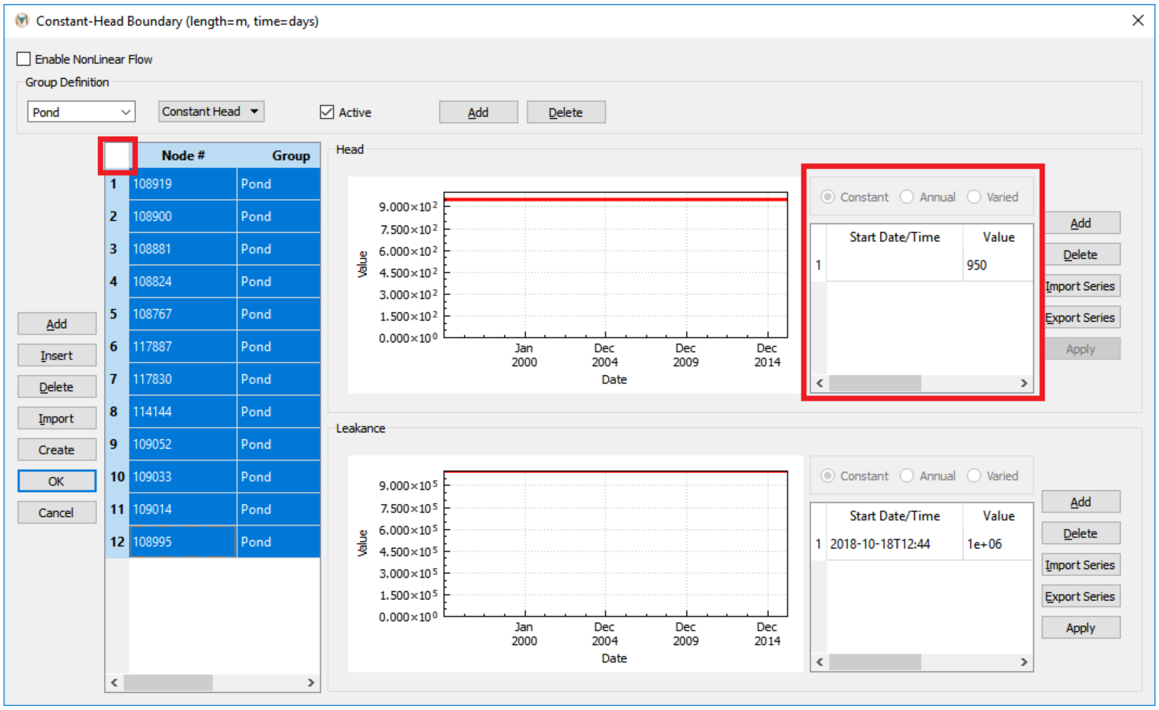


Figure 1.44. The "*Constant-Head Boundary*" dialog box

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 1.44. 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 1.44). 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.

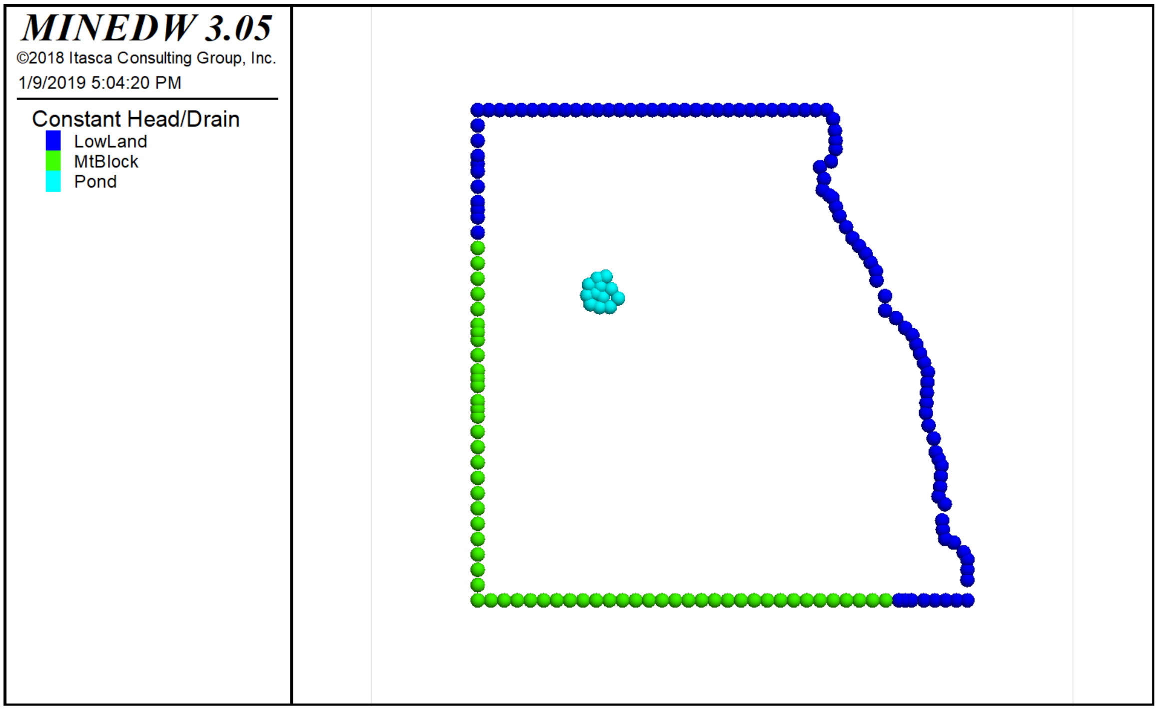


Figure 1.45. The completed constant-head boundary conditions

Figure 1.45 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.

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 1.46 with the completed fields.

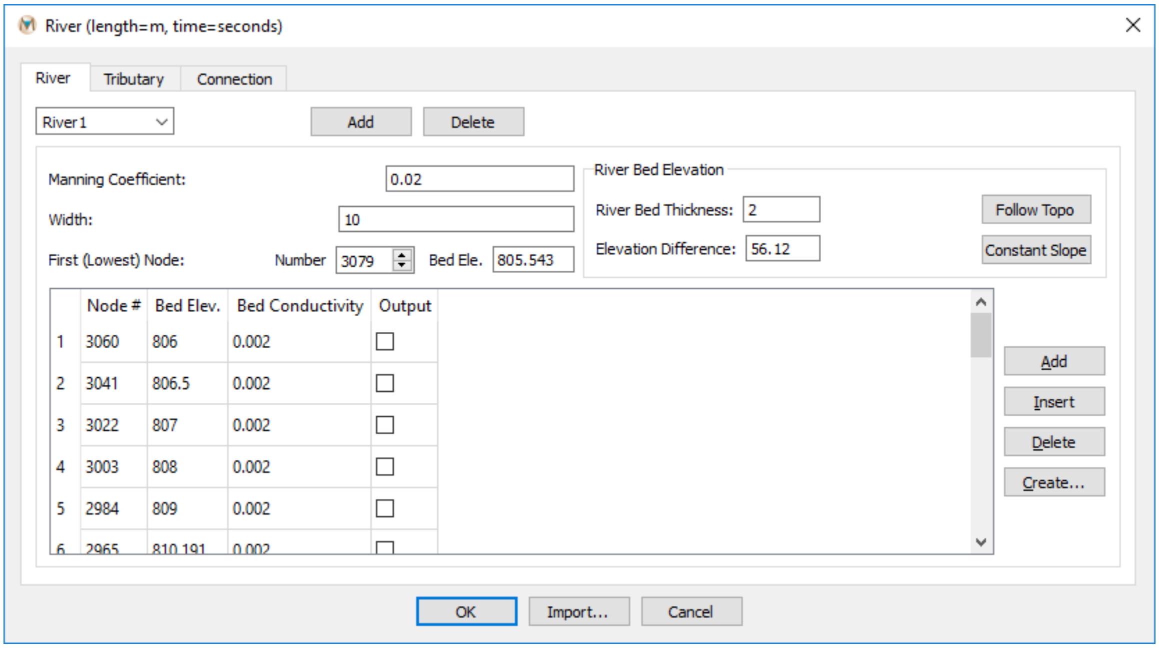


Figure 1.46. Completed river segment 1 and simulation parameters

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 1.47 shows the *"Connection"* tab with the correctly defined connections for the tutorial.

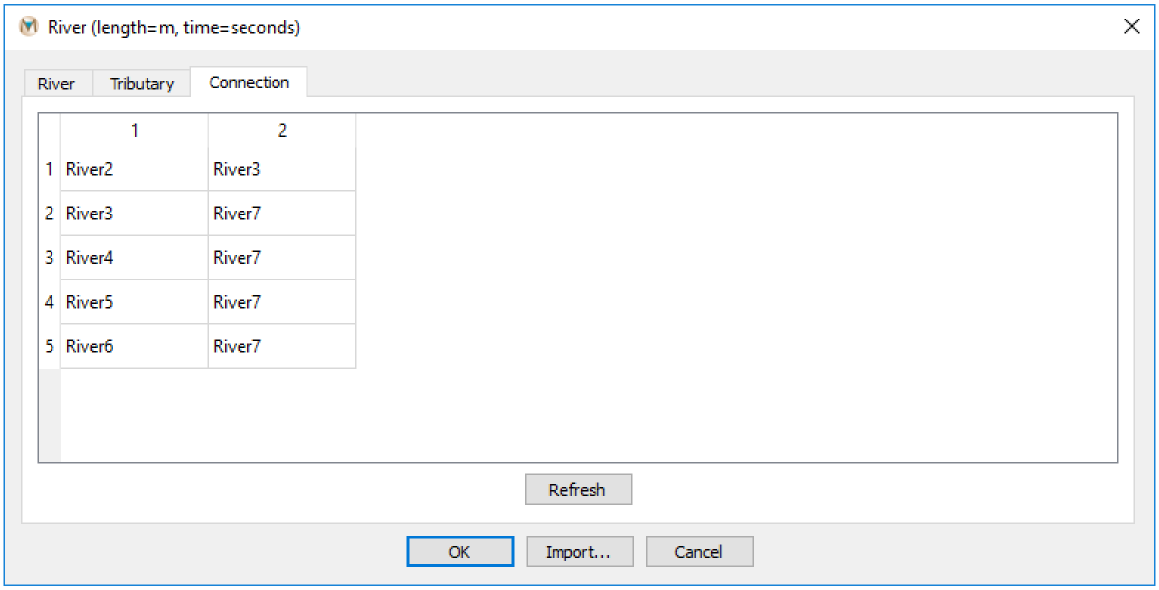


Figure 1.47. The *"Connection"* tab with connections between river reaches defined

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 1.48 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.

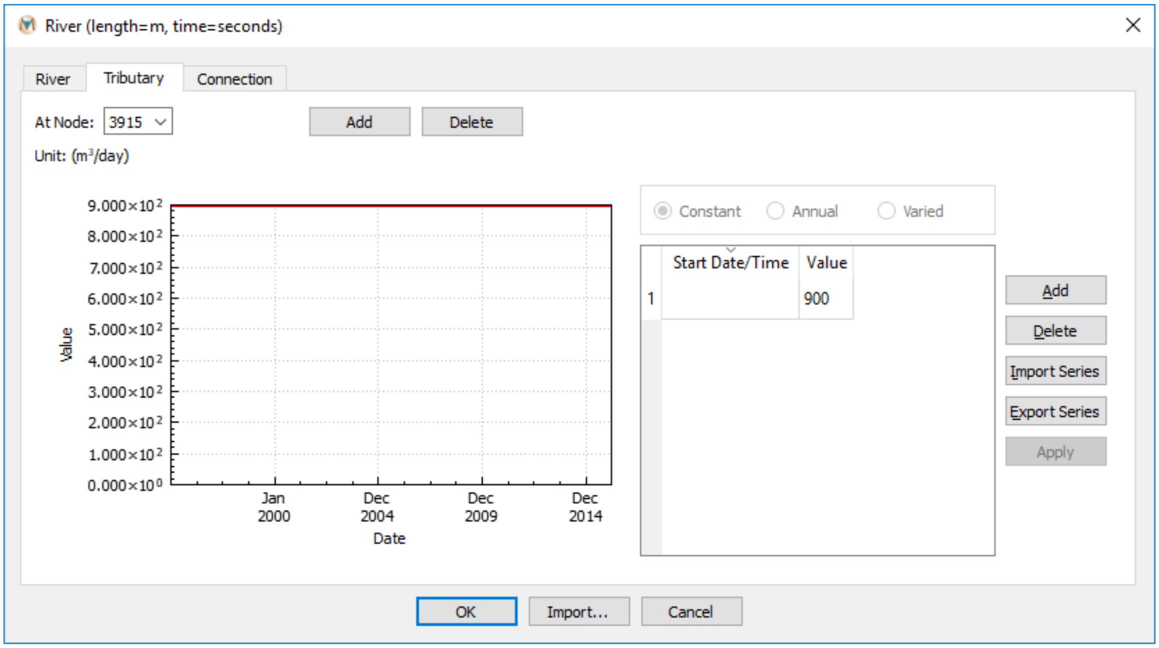


Figure 1.48. The "*Tributary*" tab showing the tributary for River 5

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 1.49 shows the completed river network and two tributaries that were added to the model.

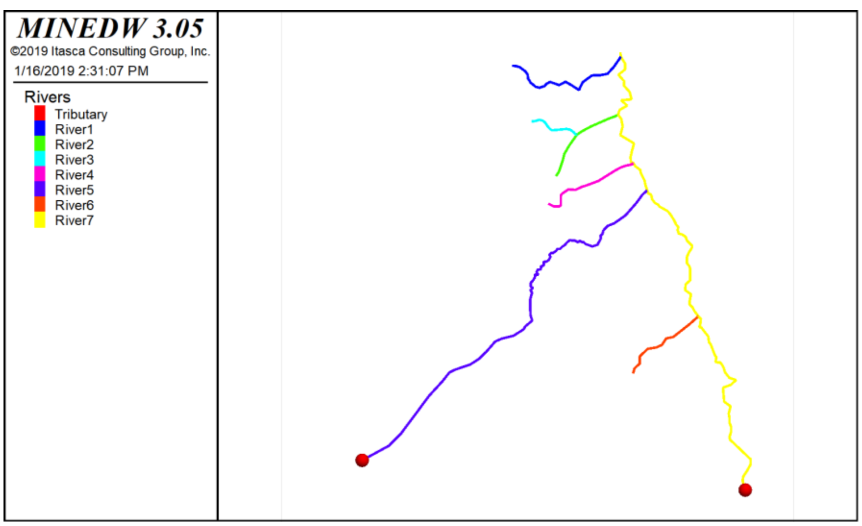


Figure 1.49. The completed river network

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.

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 1.50). To create recharge zones, click "*BCs*" on the Main Menu banner, then select "*Recharge.*" The "*Recharge*" dialog box (Figure 1.50) 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 1.50).

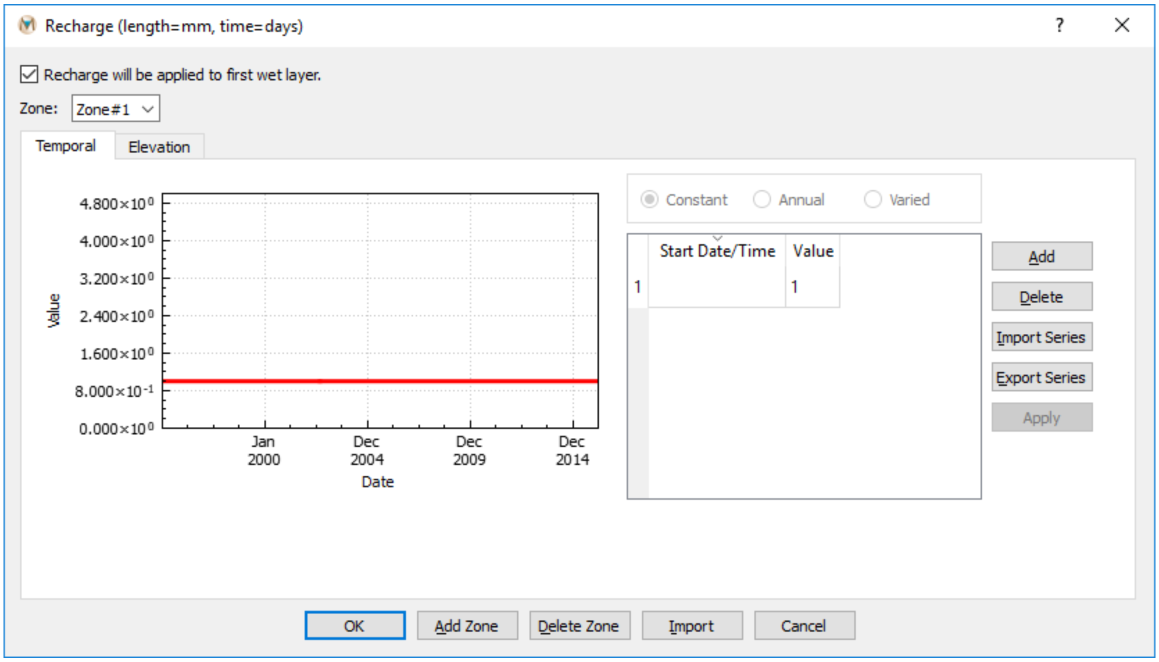


Figure 1.50. The "*Recharge*" dialog box showing Zone #1

Next, click on the *"Elevation"* tab and define the parameters for the elevation-based recharge equation (Figure 1.51). 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 1.51) 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 (Figures 1.50 and 1.51), 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."*

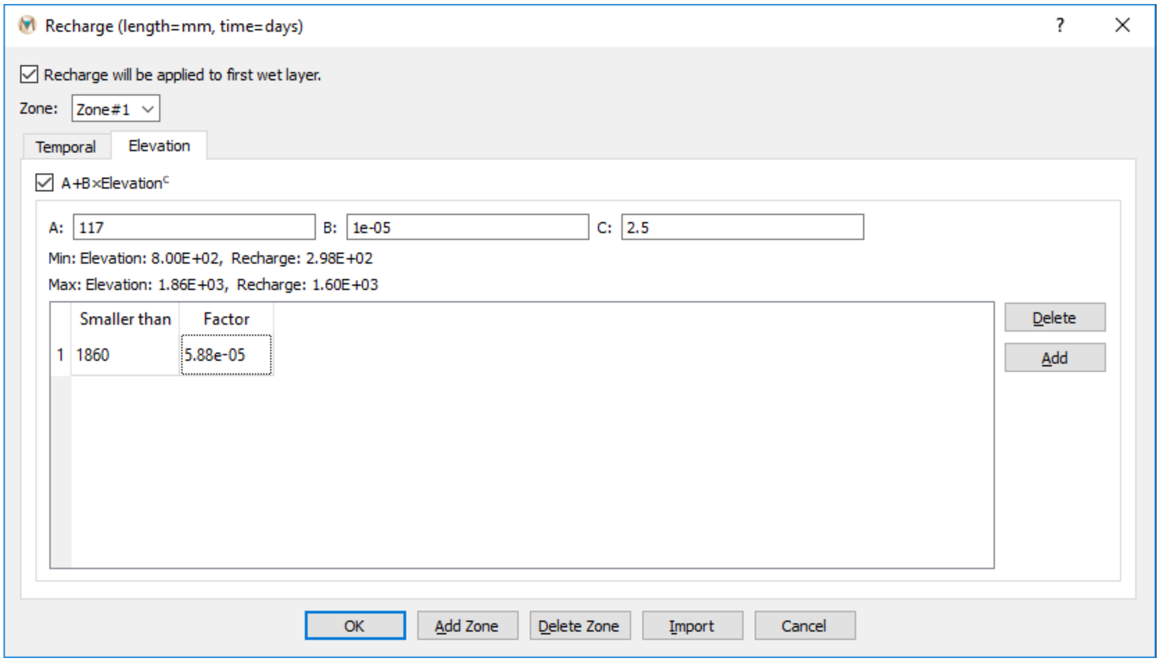


Figure 1.51. Parameters for orographically controlled recharge

To assign recharge zones to the model domain, select the "*List*" tab in the "*Control Panel*" pane. Expand the "*Element*" item and double-click "*2D Plane*" (Figure 1.52).

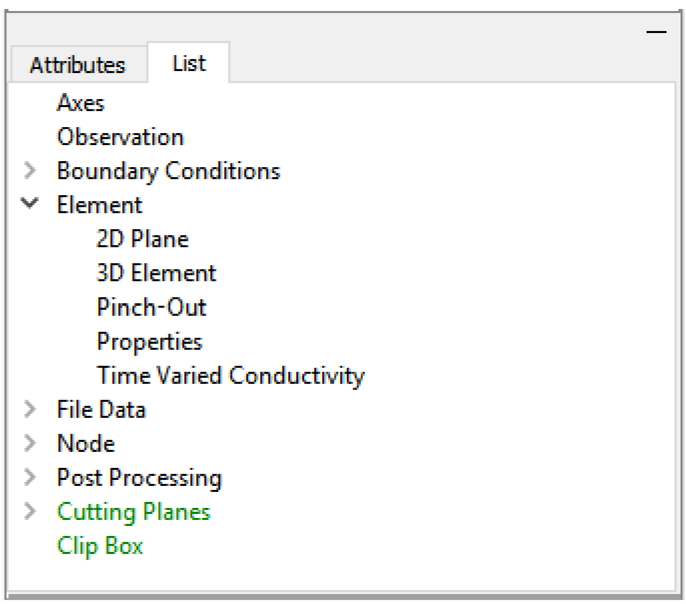


Figure 1.52. The location of the *"2D Plane"* plot item

While the 2-D plane is selected in the "*Control Panel*" pane, select the "*Attributes*" tab and select "*Recharge*" from the "*Color By*" drop-down menu (Figure 1.53).

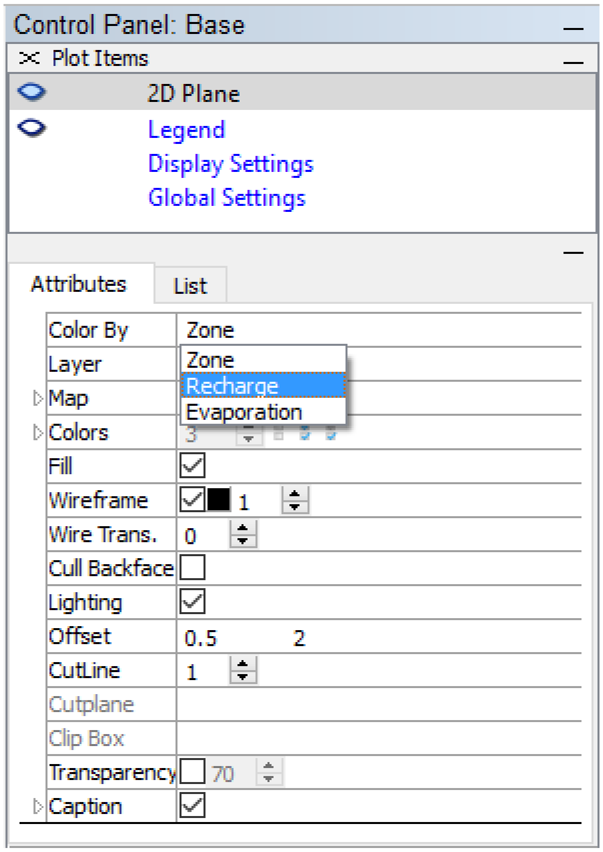


Figure 1.53. The *"Attributes"* of the *"2D Plane"* plot item

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 1.54). Make sure that Zone #1 is selected (Figure 1.54) and then click "*OK.*"

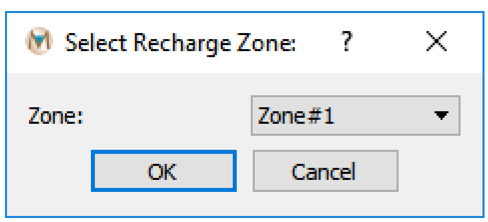


Figure 1.54. The "*Select Recharge Zone*" dialog box

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 1.55.

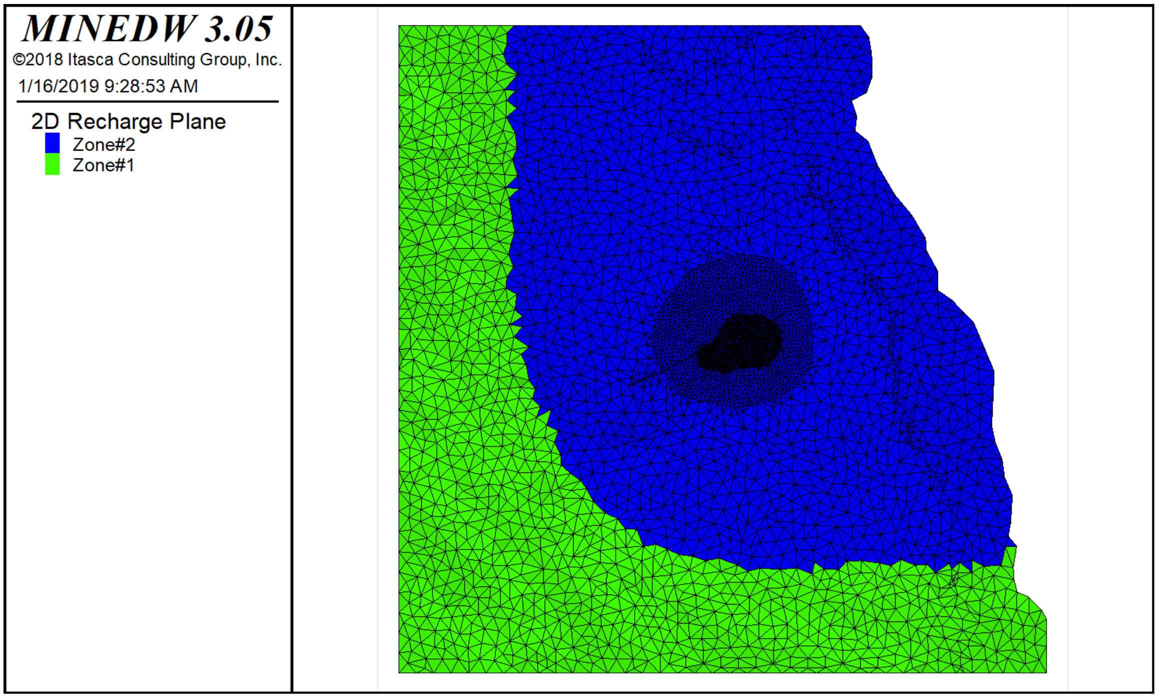


Figure 1.55. Assignment of recharge zones to the model

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 1.56). 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.

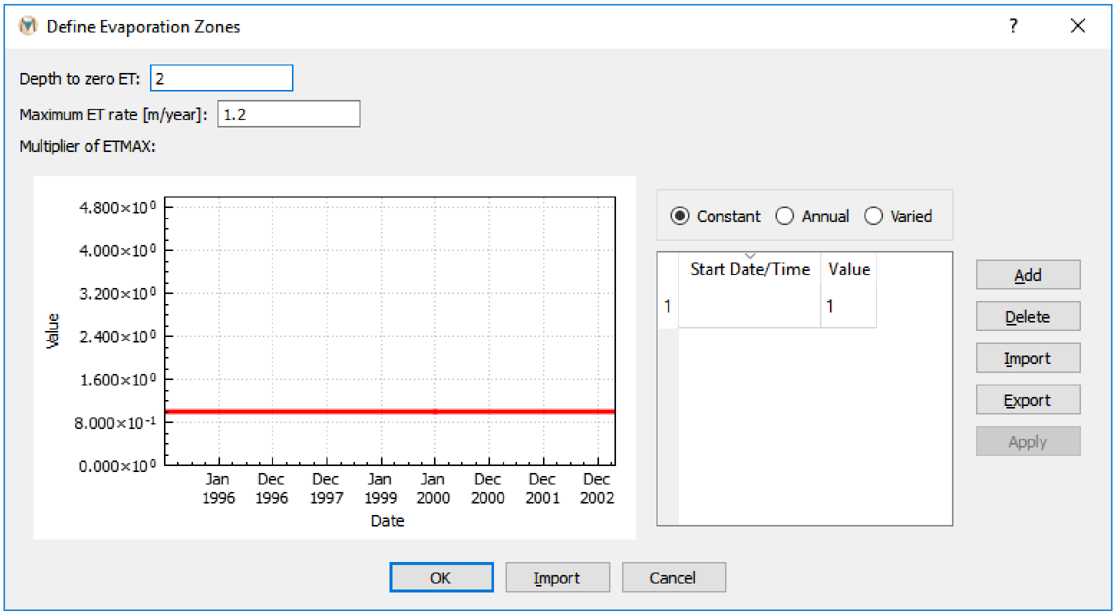


Figure 1.56. The "*Define Evaporation Zones*" dialog box

Now that the evaporation zone has been created, it needs to be assigned to the mesh. To assign evaporation to the mesh, add a "*2D Plane*" plot item and select "*Evaporation*" as the "*Color By*" attribute. Next, select all the elements that are not adjacent to the *"River"* or "*Pond*" boundary conditions. It may be easiest to use the *"Select With Polygon"* tool and select elements that do not border the *"River"* or "*Constant Head*" boundary conditions (Figure 1.57). When the appropriate elements have been selected, press [Enter] to open the *"Select Evaporation Zone"* dialog box. Select *"Evaporation"* in the drop-down box and then click *"OK"* to finalize the evaporation zone.

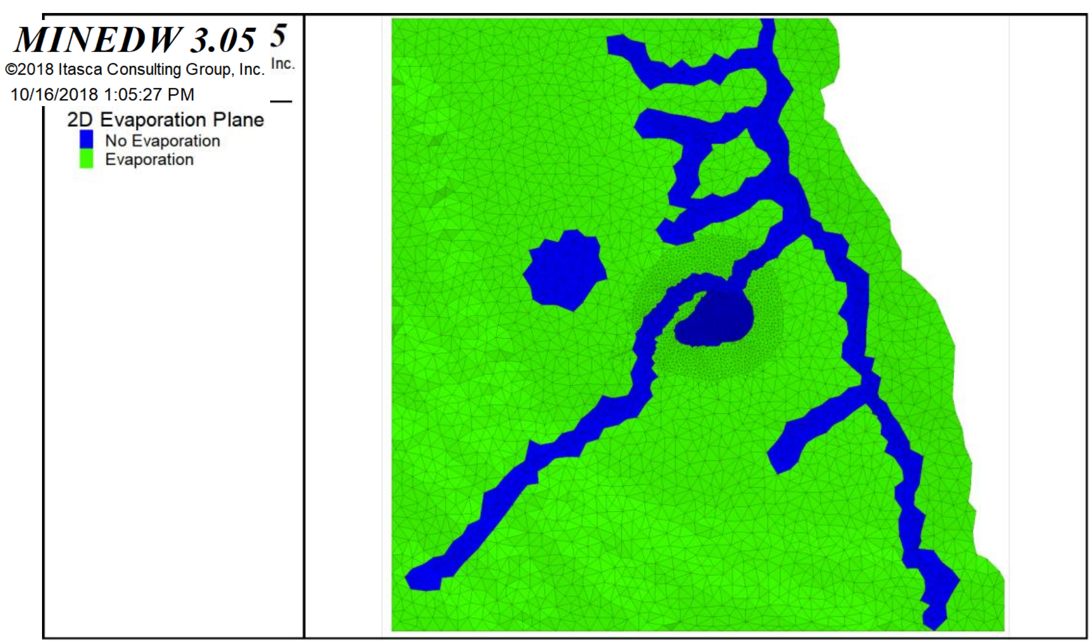


Figure 1.57. The assigned evaporation zones

Step 14. Saving the Project

After completing the model setup, click "*File*" on the Main Menu banner. Select "*Save Project*" to save the changes to the project.

1.4. 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.

1.5. 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.

Water-Table Contour

To plot the water-table contours, select the "*List*" tab from the "*Control Panel*" pane, expand the "*Node*" item, and then double-click "*2D Contour*" (Figure 1.58).

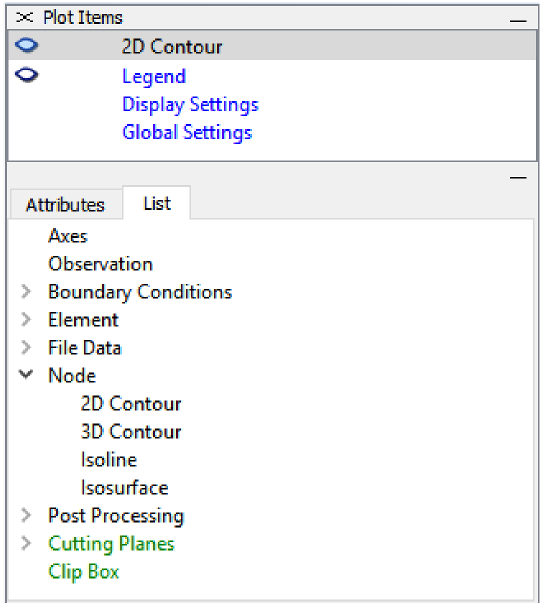


Figure 1.58. The "*2D Contour*" plot item

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 1.59. For the steady-state simulation, view the last time step (time step 100 in this tutorial).

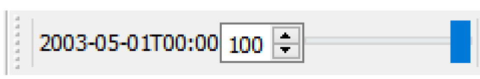


Figure 1.59. Time step slider

To export the water-table contour as a bitmap, select "*Export Base*" from the "*File*" item on the Main Menu banner and choose "*Bitmap.*" The exported bitmap is provided in Figure 1.60.

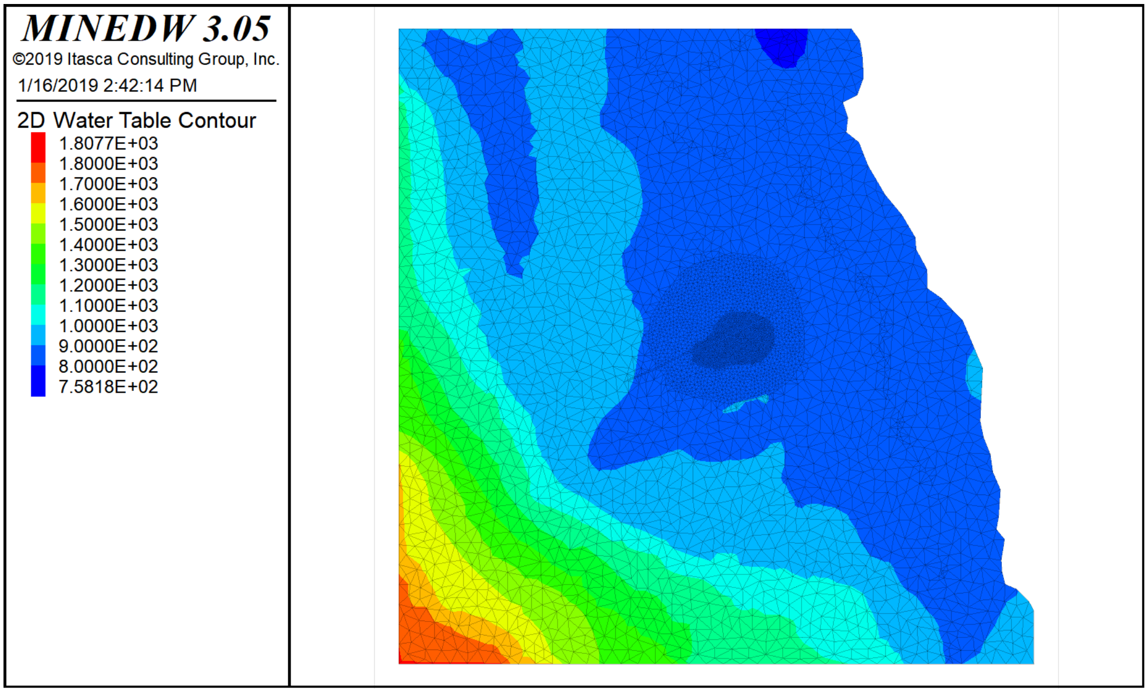


Figure 1.60. Water-table contours from steady-state simulations

2-D or 3-D Head Contour

To create a 2-D or 3-D contour plot of the head distribution, select the *"Node"* item under *"List"* on the *"Control Panel"* pane and double-click on either *"2D Contour"* or *"3D Contour."* 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 1.61 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, double-click on *"Plane"* located under *"Cutting Planes."* Enter the following values for the *"Plane"* on the *"Attribute"* tab: *Origin* = (466236, 9.07489e+6, 250), *Normal* = (-0.429008, 0.903301, 0) *Dip/DD* = (90, -25.4046). 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 *"Cutplane"* 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.

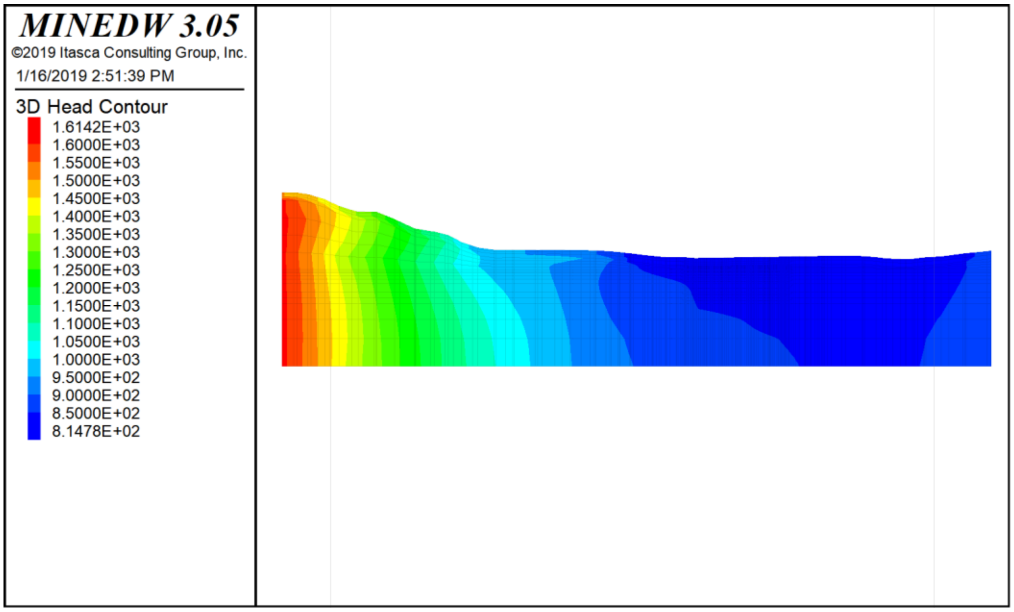


Figure 1.61. A 3-D cross section of head exported from *MINEDW*

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 1.62, select *"3D Element"* (found under *"Element"*) from the *"List"* tab in the *"Control Panel"* pane. Next, add a *"2D Contour"* (under *"Node"*) and select *"Water Table"* from the *"Color By"* drop-down menu on the *"Attributes"* tab. Finally, create a cross section using the *"Create Plane with two selected points on the plot"* tool. Colors for both the *"3D Element"* and *"2D Contour"* plot items can be modified by expanding the *"Contour"* attribute and choosing a different color ramp.

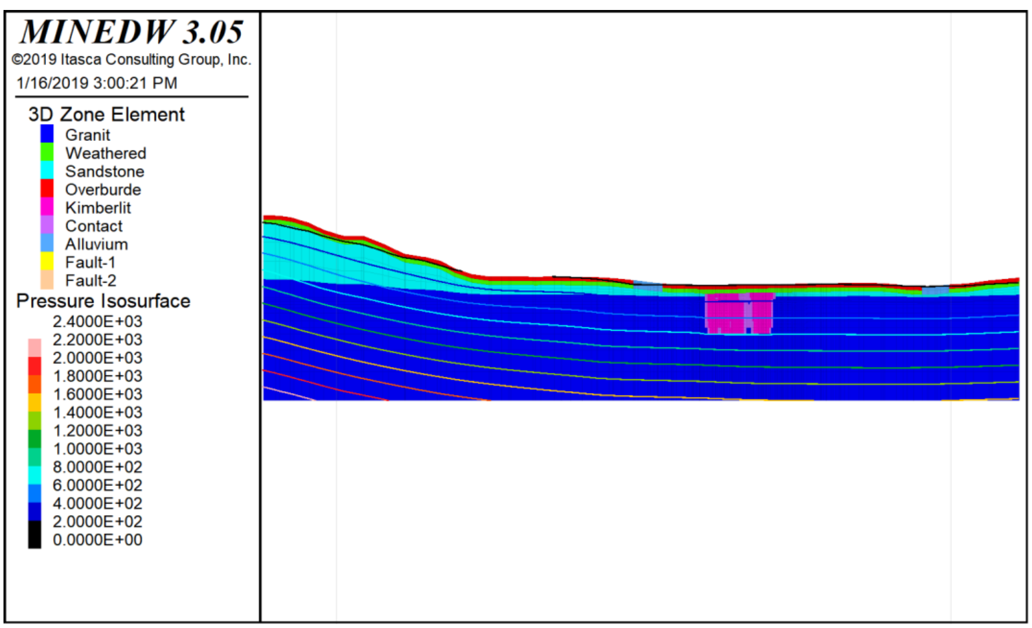


Figure 1.62. A cross section of the geology and water table

Tutorial 2. Open Pit, Zone of Relaxation, Pumping Wells, and Pit Lake

2.1 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.

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 2.1.

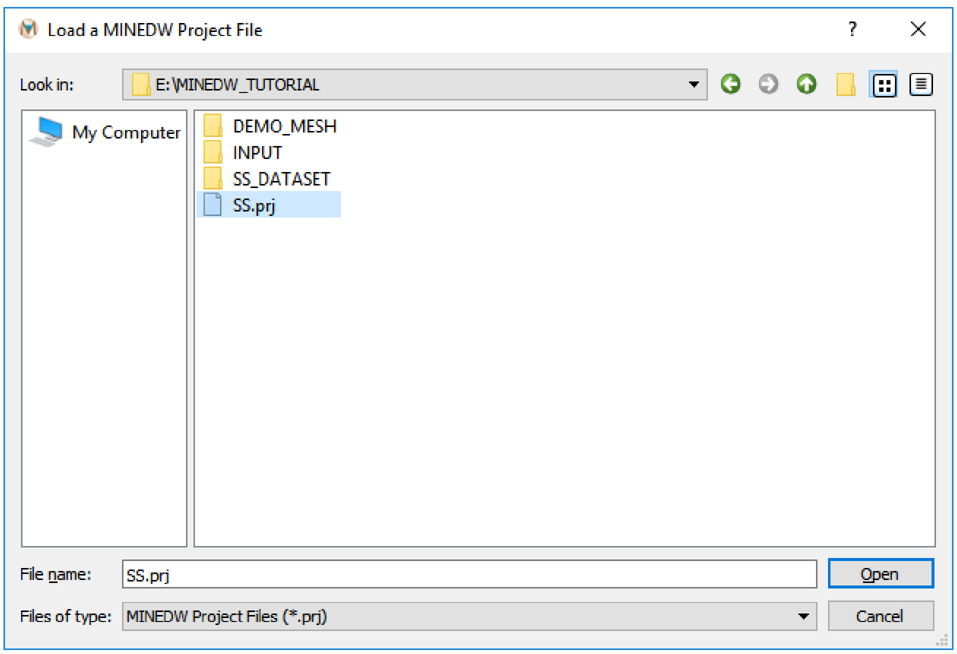


Figure 2.1. The "*Load a MINEDW Project File*" dialog box

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.

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 2.2 appears. Make sure that all the fields shown in Figure 2.2 contain the values shown in Figure 2.2. For transient simulations, select the "*Transient*" option and then click "*OK.*"

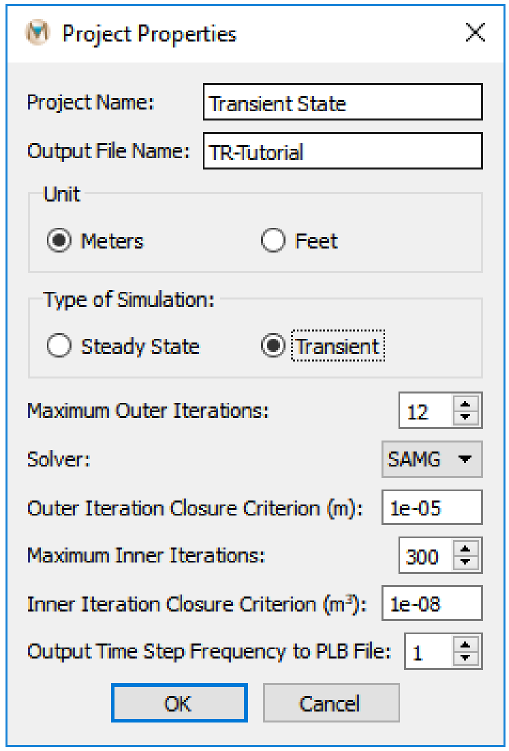


Figure 2.2. The "*Project Properties*" dialog box

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 2.3) appears. Set the "*Maximum Time Steps*" to 300. The first time step is 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.*"

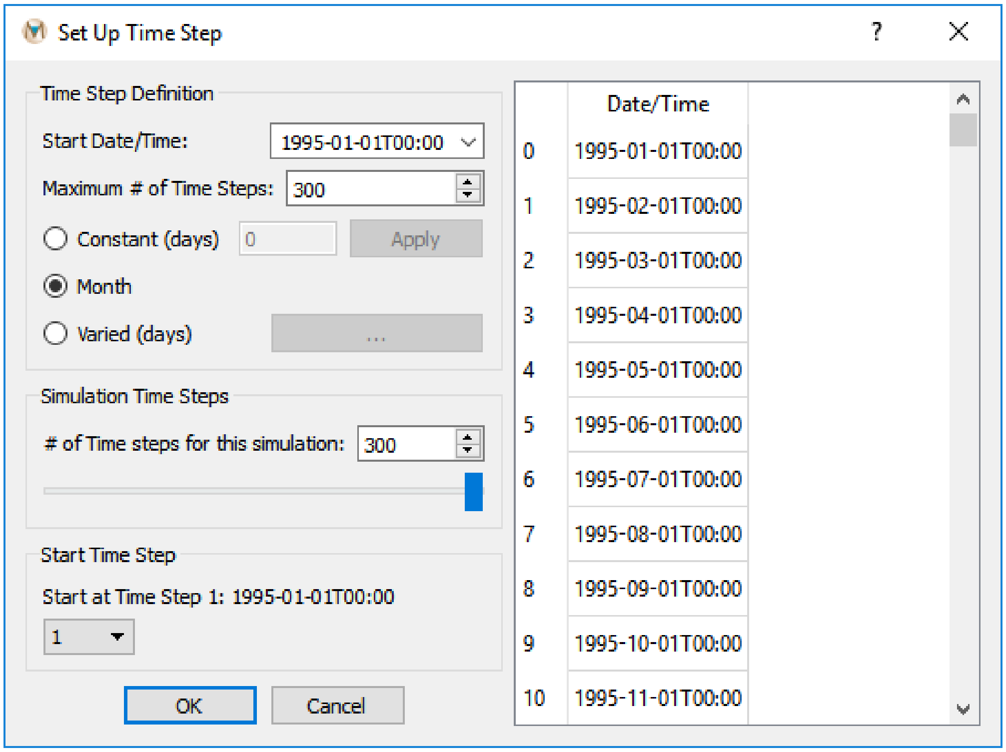


Figure 2.3. The "*Set Up Time Step*" dialog box

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 2.4, then click "*OK.*"

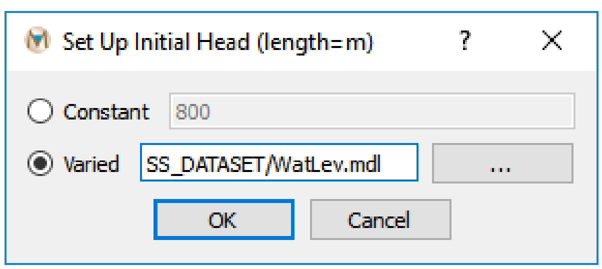


Figure 2.4. The "*Set Up Initial Head*" dialog box

Step 4. 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 "*INPUT/OPENPIT*" file directory.

On the "*List*" tab, double-click on "*File data,*" then double-click on "*DXF*." The "*Attributes*" tab will appear. To select the .DXF file for Year 1, select the "+" next to "*File*" as shown in the red box in Figure 2.5, browse to the file, select it, and click "*Open*." Next. select the icon next to "*To Data File*" at the bottom of the "*Attributes*" tab to save the information from the .DXF file as a data ("*.DAT") file. 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.

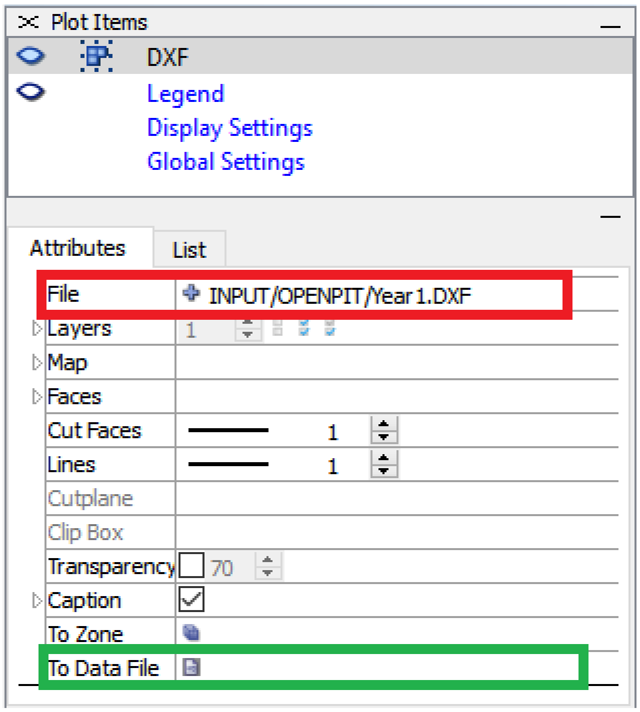


Figure 2.5. Reading a .DXF file and saving as a data file

To create the mining plan, on the Main Menu banner, click "*Mining,*" then select "*Create Mining Plan*." The "*Create Mining Plan*" dialog box (Figure 2.6) 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 2.6. 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 2.6). 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 Rang."* 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.

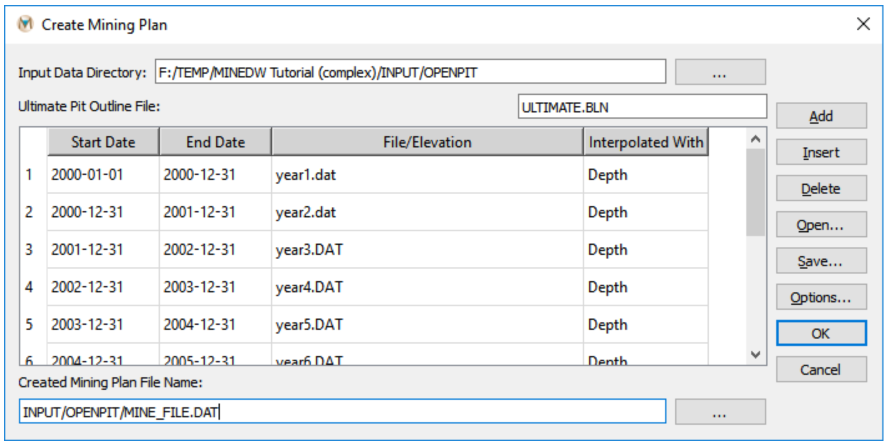


Figure 2.6. The "*Create Mining Plan*" dialog box

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 2.7).

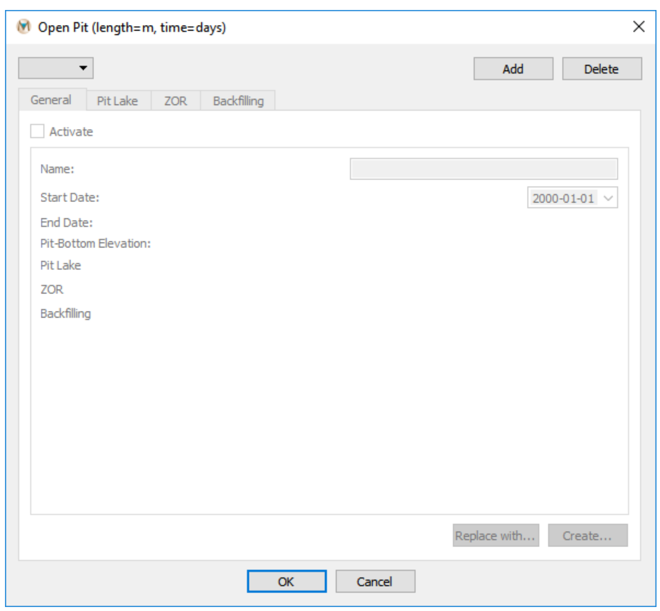


Figure 2.7. The "*Open Pit*" dialog box

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*." 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.

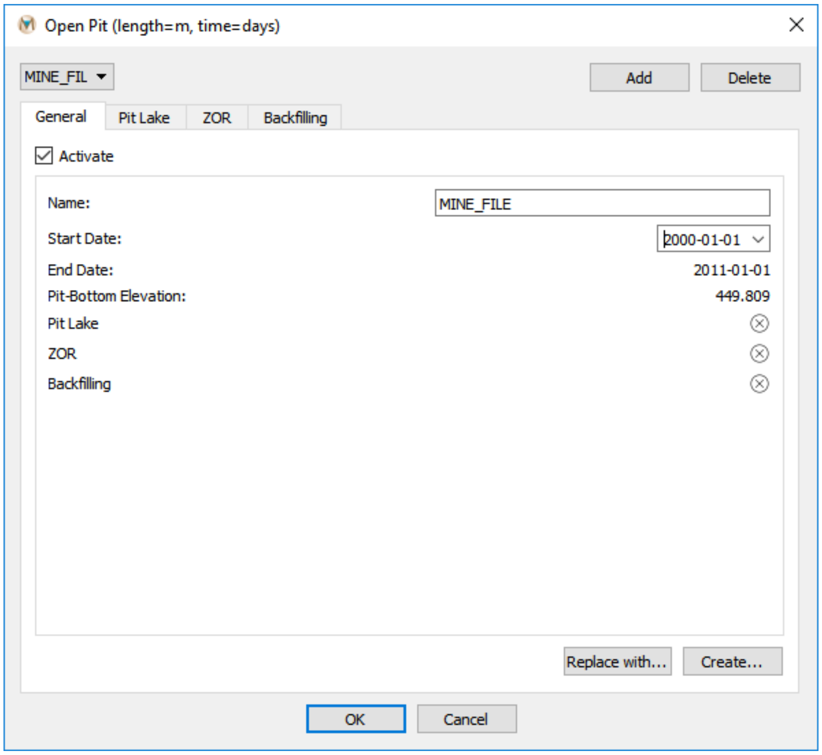


Figure 2.8. The "*Open Pit*" dialog box showing the "*General*" tab

Step 5. 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 2.8) 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 2.10).

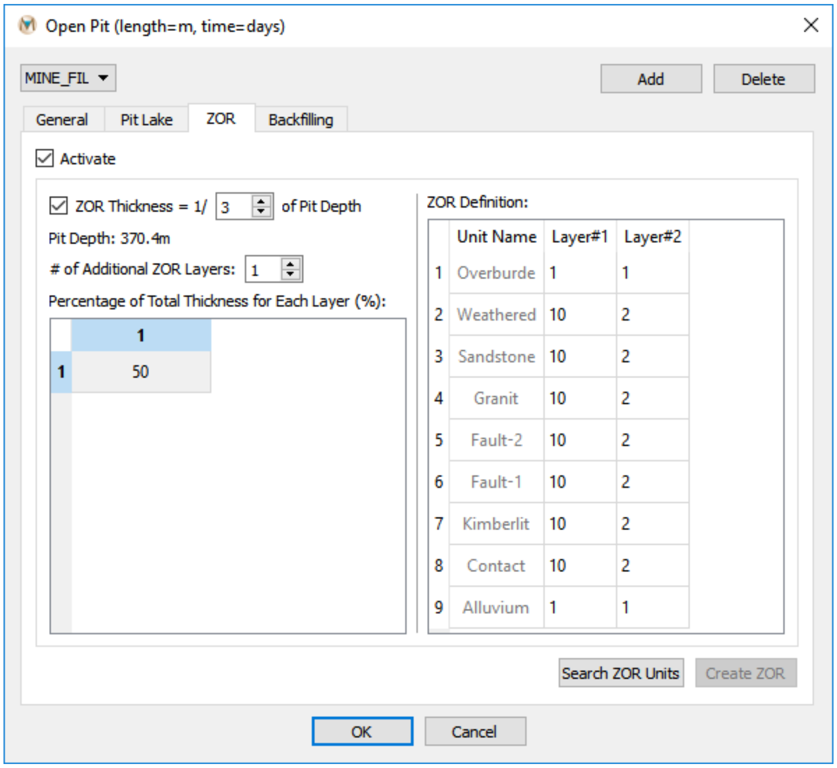


Figure 2.9. The "*Open Pit*" dialog box showing the "*ZOR*" tab

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 2.9). 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 2.10 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.

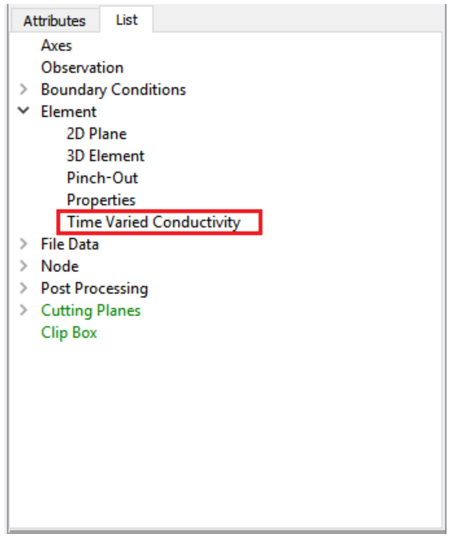


Figure 2.10. The *"Time Varied Conductivity"* plot item

Once a ZOR has been created, it can be visualized by selecting the "*List*" tab in the "*Control Panel*" pane and then selecting the "*Time Varied Conductivity*" plot item under the "*Element*" group (Figure 2.10). The ZOR that was created is shown in Figure 2.11.

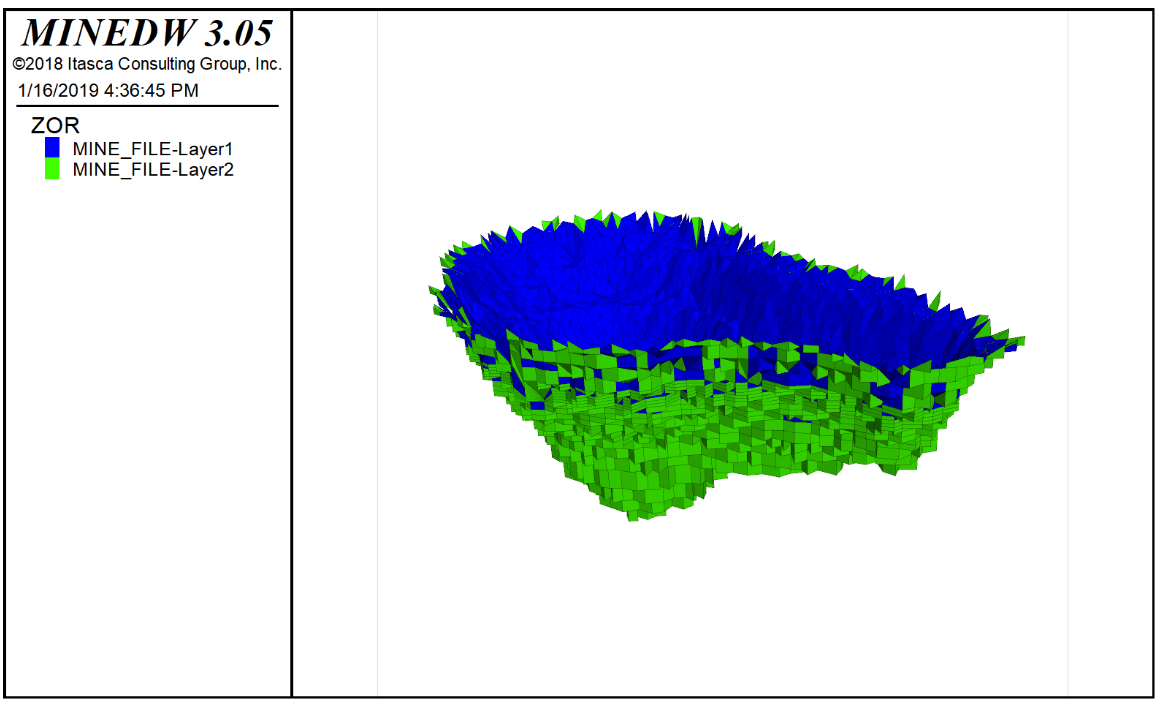


Figure 2.11. ZOR defined in the model

Step 6. Open-Pit Backfilling

Backfilling may be used in some mining applications to prevent the formation of a pit lake or to dispose of tailings after mining has ceased. To add backfilling, select the *"Backfilling"* tab (Figure 2.12). Check the *"Activate"* box and define the *"Implement Date"* for backfilling. The *"Implement Date"* of backfilling must be before the *"Start Date"* of pit-lake formation. ***MINEDW*** does not support the backfilling of an existing pit lake and will automatically adjust the *"Start Date"* or *"Implement Date"* for pit-lake formation or backfilling operations to prevent this. Backfilling operations are simulated in one time step rather than progressive backfilling over multiple time steps. The backfill elevation can be defined as constant by checking the *"Constant Elevation"* option and entering a value. Otherwise, if more detailed information is available for the backfill, *"Varied"* can be selected. For this tutorial, check the *"Constant Elevation"* radio button and enter the elevation of the final backfill (Figure 2.12). Finally, the hydraulic parameters for the backfill can be selected from the drop-down list labeled *"Backfill Unit"* (Figure 2.12). The entire backfilled zone will have the same hydraulic properties and cannot be subdivided. When the *"Backfilling"* parameters are completely defined, click *"OK"* in the *"Open Pit"* dialog box to ensure the changes that were made are saved.

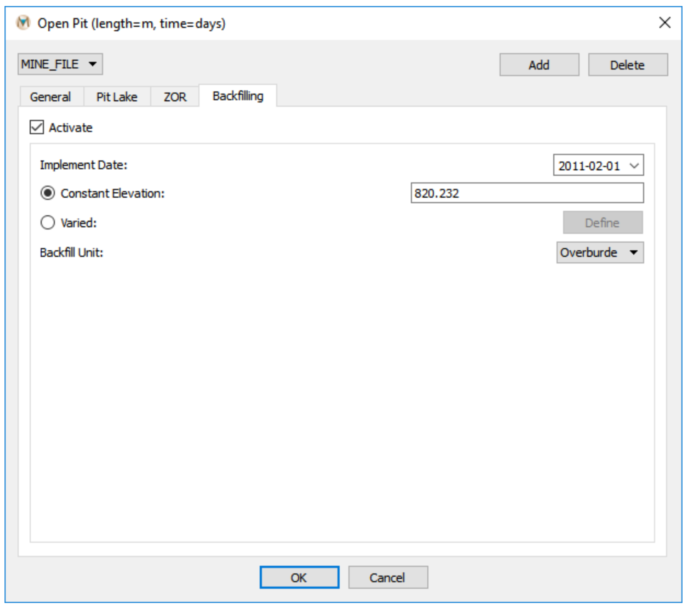


Figure 2.12. The "*Backfilling*" tab

Step 7. Creating a Pit Lake

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 2.8). 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 2.8). 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 2.13) 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.

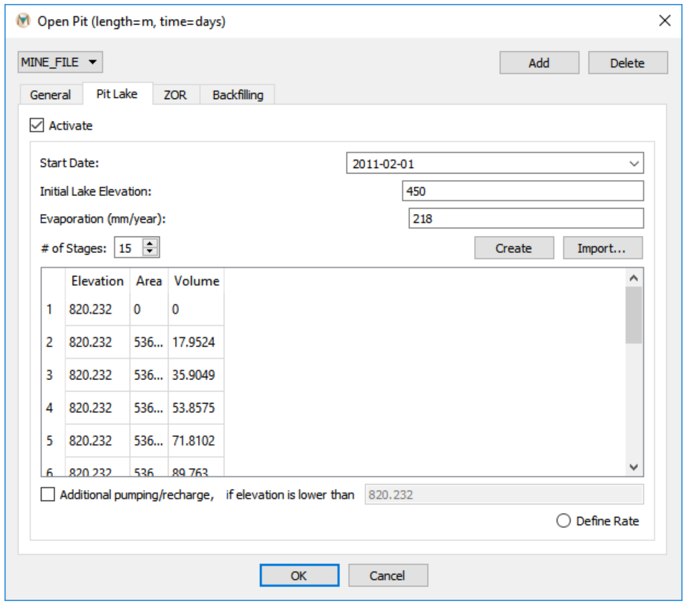


Figure 2.13. The "*Pit Lake*" tab

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 "*Control Panel*" pane, select the "*List*" tab, expand the "*Node*" item, and then double-click "*3D Contour.*" Next, toggle to layer 27 using the toggle buttons next to the "*Layer*" attribute on the "*Attributes"* tab (Figure 2.14). 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.

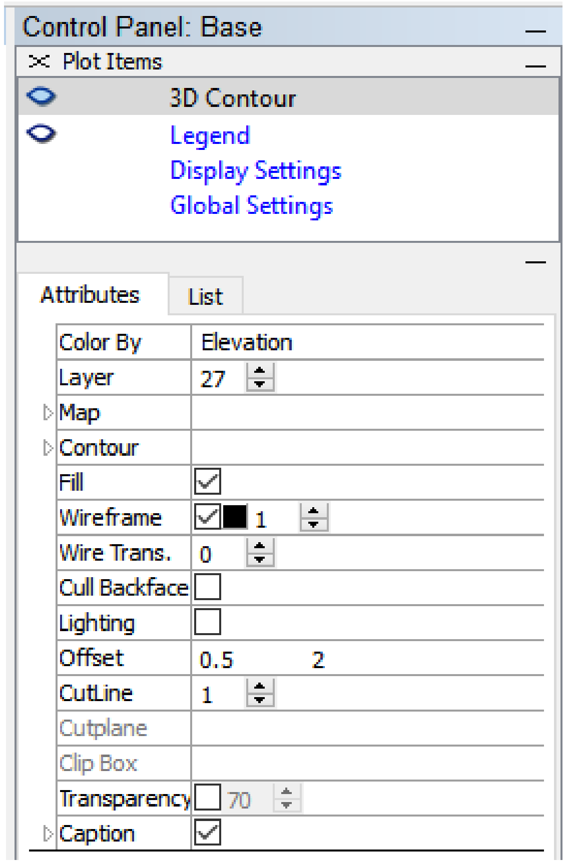


Figure 2.14. The "*Control Panel*" pane with the "*Attributes*" tab selected to show the correct layer for pumping wells

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 *"File Data"* item in the *"Control Panel"* to the View Pane (Figure 2.15). 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.Dat"* file in the *"INPUT/MESH_INPUT/DAT"* directory.

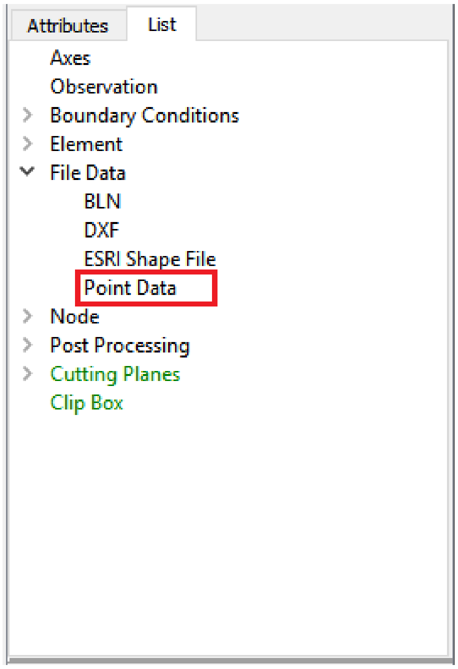


Figure 2.15. The *"Point Data"* plot item

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 2.16), 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.

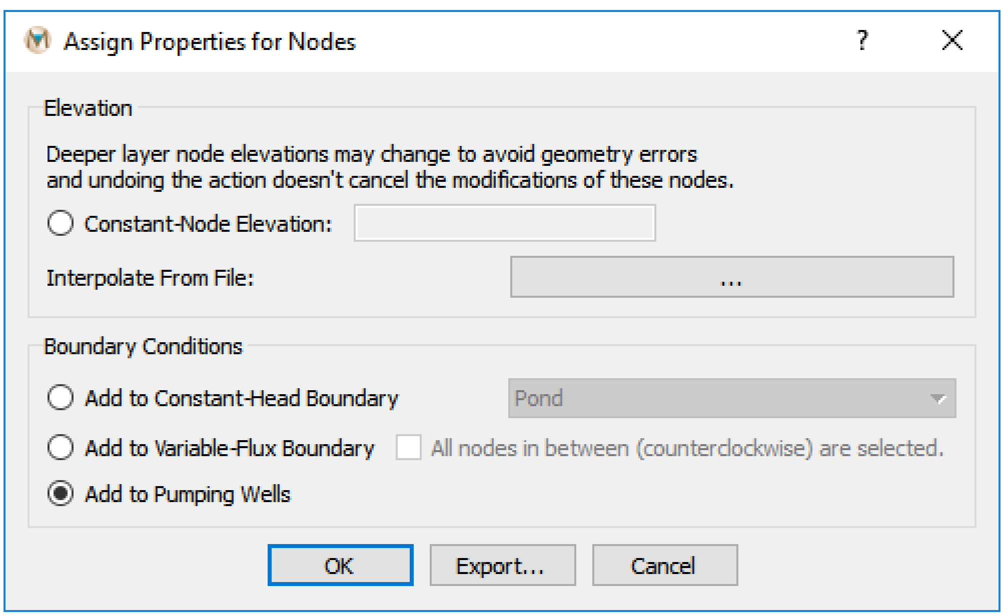


Figure 2.16. The "*Assign Properties for Nodes*" dialog box showing the "*Add to Pumping Wells*" option selected

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 2.17).

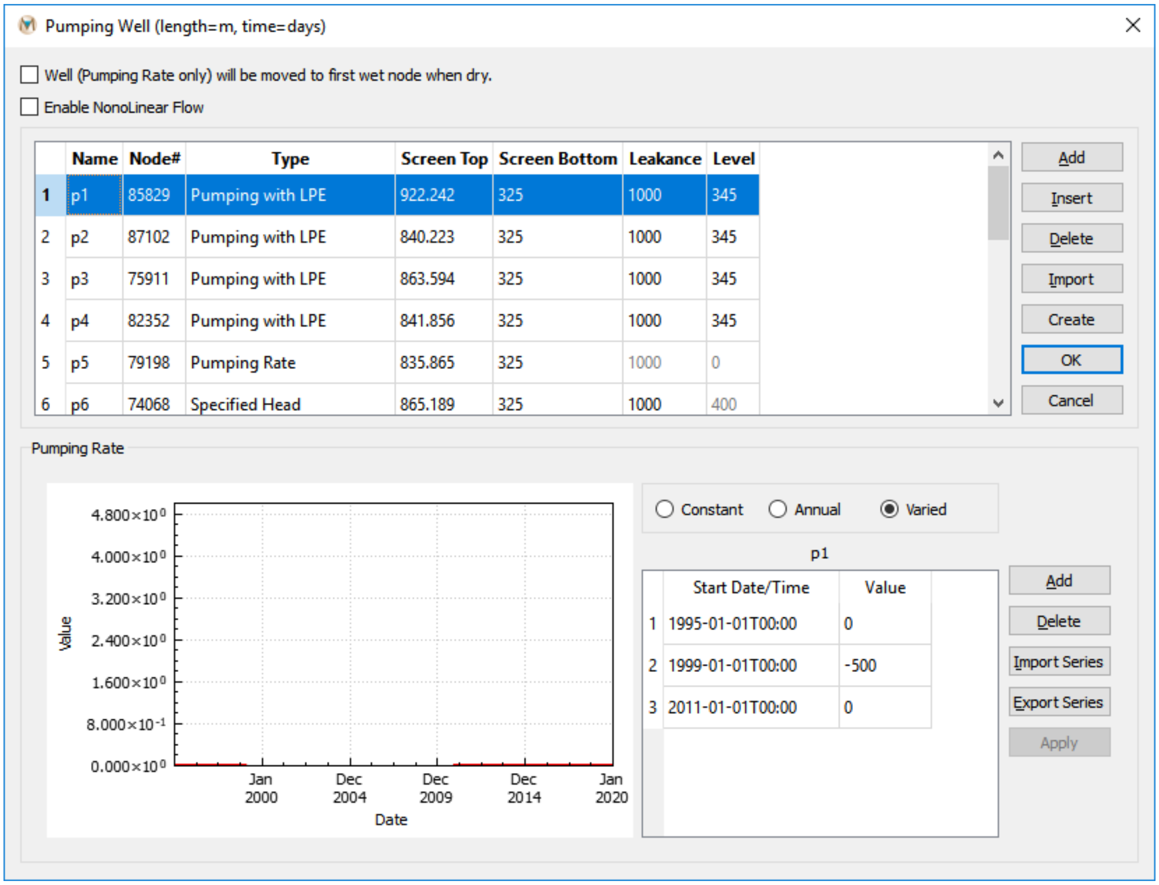


Figure 2.17. The "*Edit Pumping Wells*" dialog box

Set the "*Type*" as "*Pumping with LPE*" for the first several pumping wells (Figure 2.16 and Figure 2.17). 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 2.17). If the pumping rate changes with time, go to the lower right window shown in Figure 2.17 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.

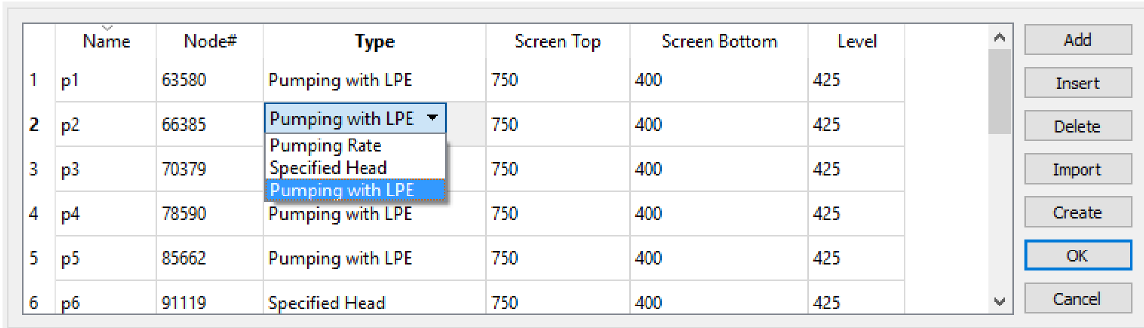


Figure 2.18. The "*Edit Pumping Wells*" dialog box showing the pumping well types available in *MINEDW*

For "*Pumping Rate*" wells, select "*Pumping Rate*" under "*Type*" (Figure 2.18) 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.

"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.

Step 9. Defining the Boundary Conditions

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 2.19.

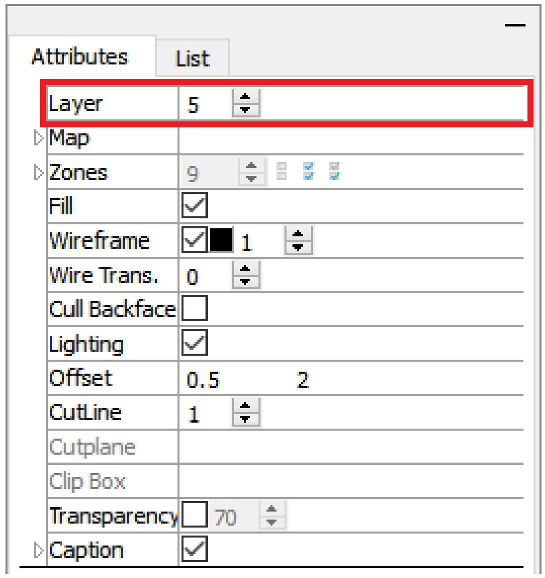


Figure 2.19. The "*Attributes*" tab

From the ***MINEDW*** main menu, click the "*Select*" button on the Main Menu banner.

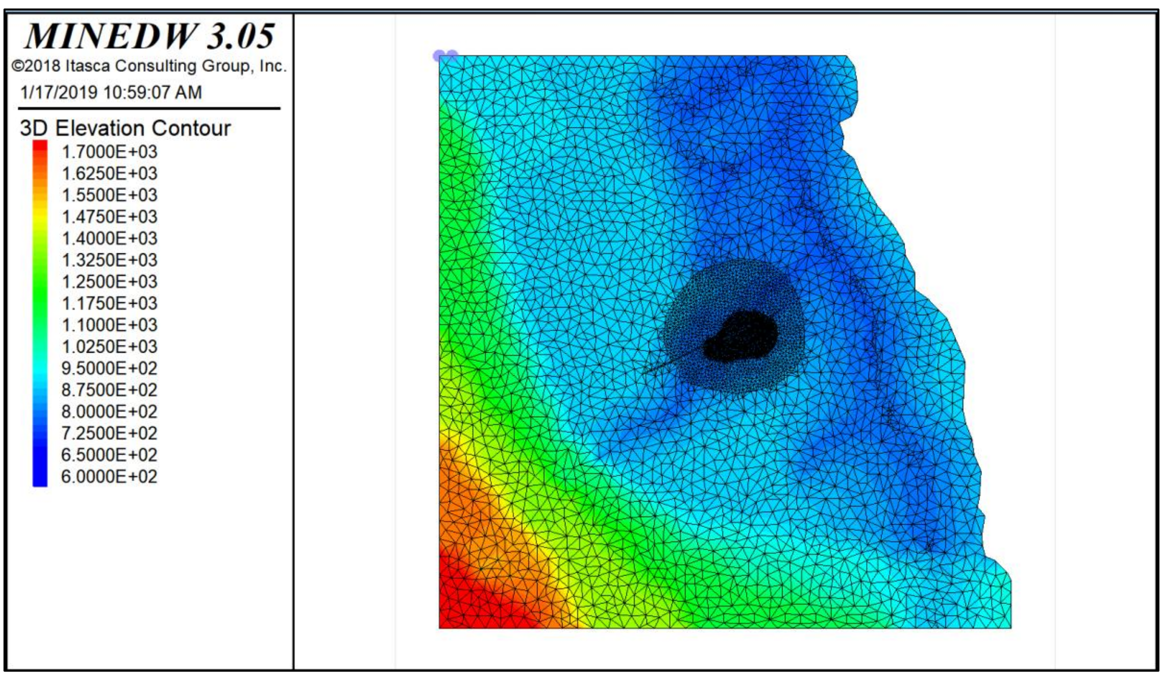


Figure 2.20. The View Pane showing the first and last node of the variable-flux boundary

Select the beginning node (node at top left corner) and ending node (the next node to the right) (Figure 2.20). 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 2.21). Select "*Add to Variable-Flux Boundary*" and check the "*All nodes in between (counterclockwise)*" box, as shown in Figure 2.21 below. Enter "*32*" or whatever layer is the bottom layer in the "*From Current Layer #1 To*" box on the right.

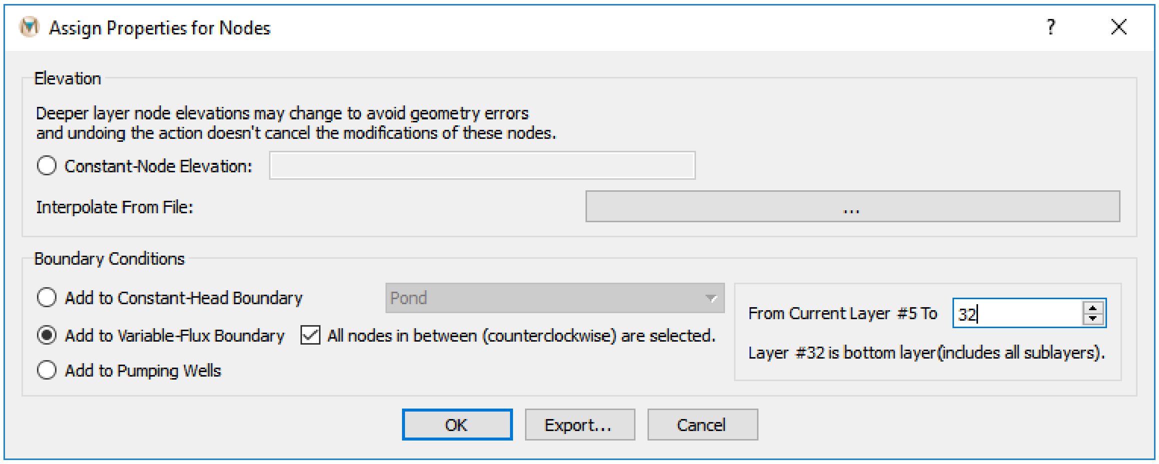


Figure 2.21. The "*Assign Properties for Nodes*" dialog box showing the "*Add to Variable-Flux Boundary*" option selected

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 2.22 shows the *"Variable-Flux Boundary"* dialog box with the imported flow rates at the bottom of the dialog box.

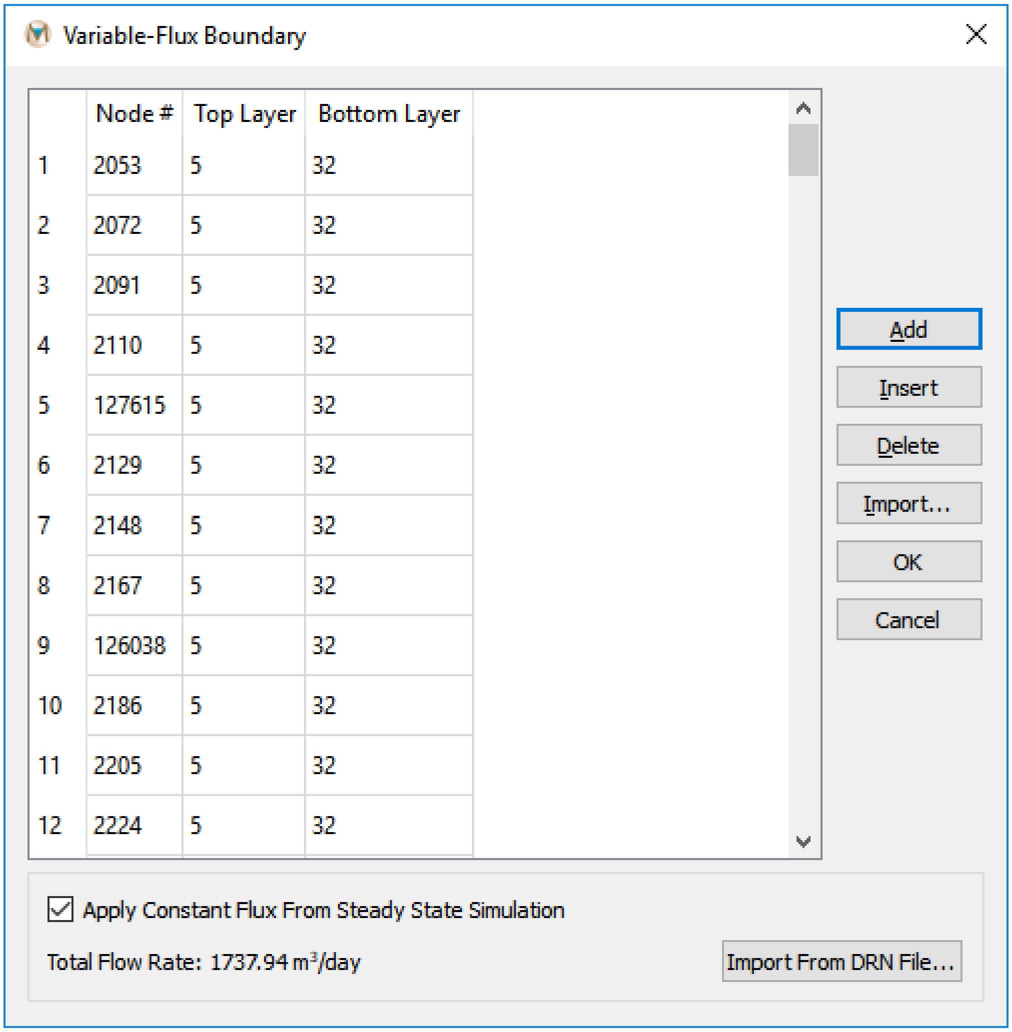


Figure 2.22. Imported flow rates

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.

2.2 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 2.23).

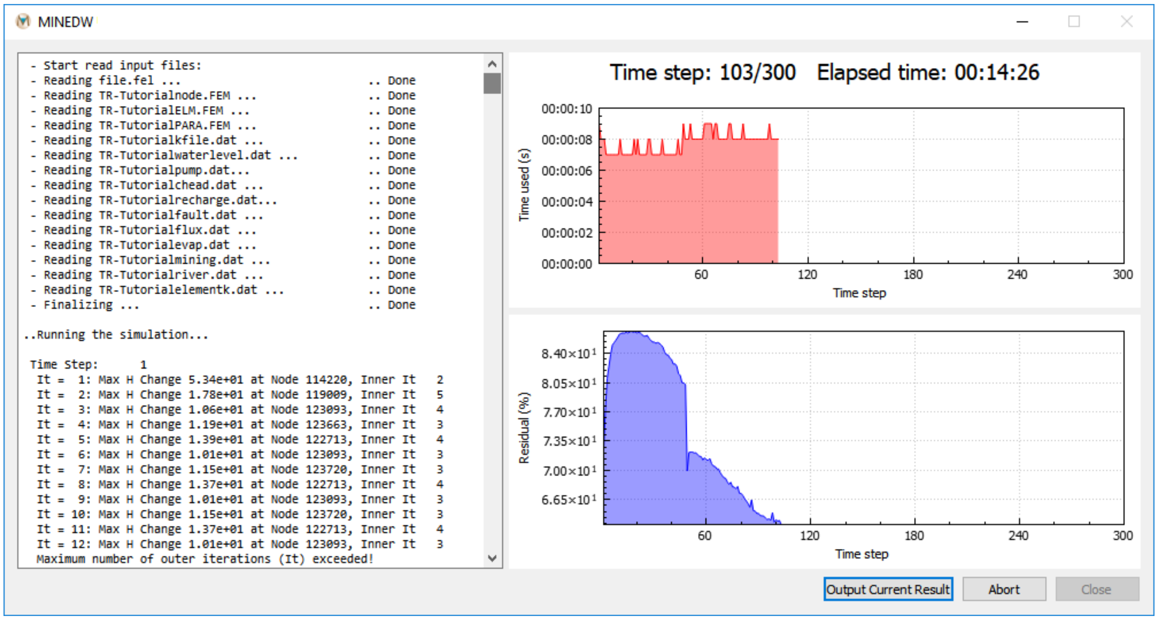


Figure 2.23. Screen display of output from the model run

2.3 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 2.24) to the desired time step.

Figure 2.24. Time-step slider

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 2.25). 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 2.26). 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 2.26) 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.

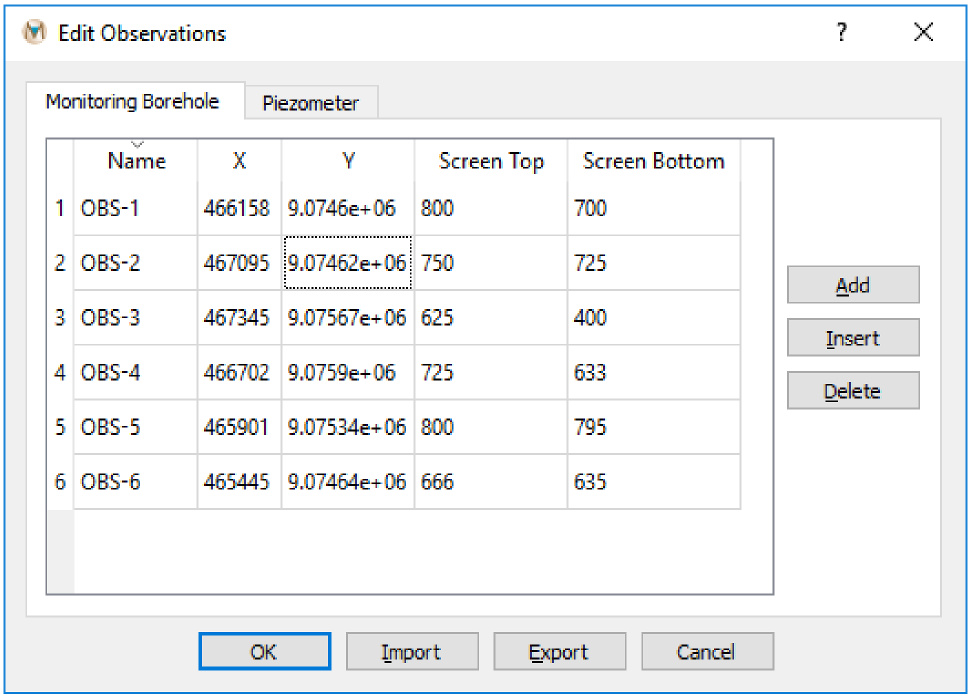


Figure 2.25. The "*Edit Observations*" dialog box

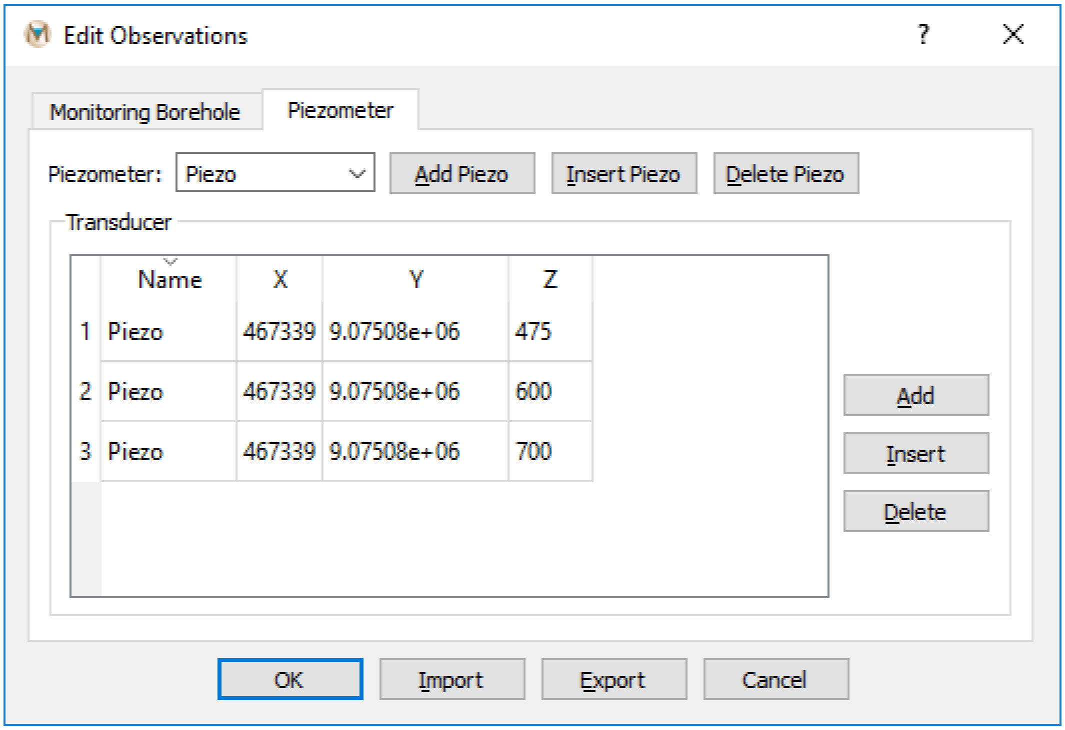


Figure 2.26. The "*Edit Observations*" dialog box with the "*Piezometer*" tab selected

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.

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 2.27). 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 2.27.

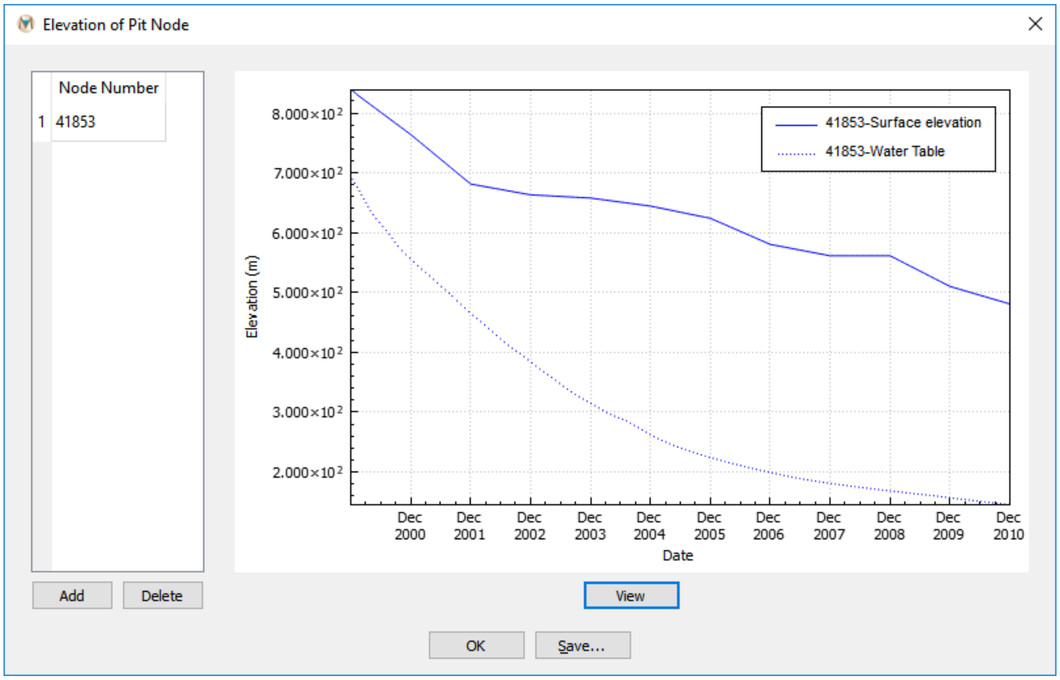


Figure 2.27. "*Elevation of Pit Node*" dialog box

If multiple nodes need to be plotted, click "*Add*" and insert the additional *"Node Numbers."* Then click "*View*" again (Figure 2.28). 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."*

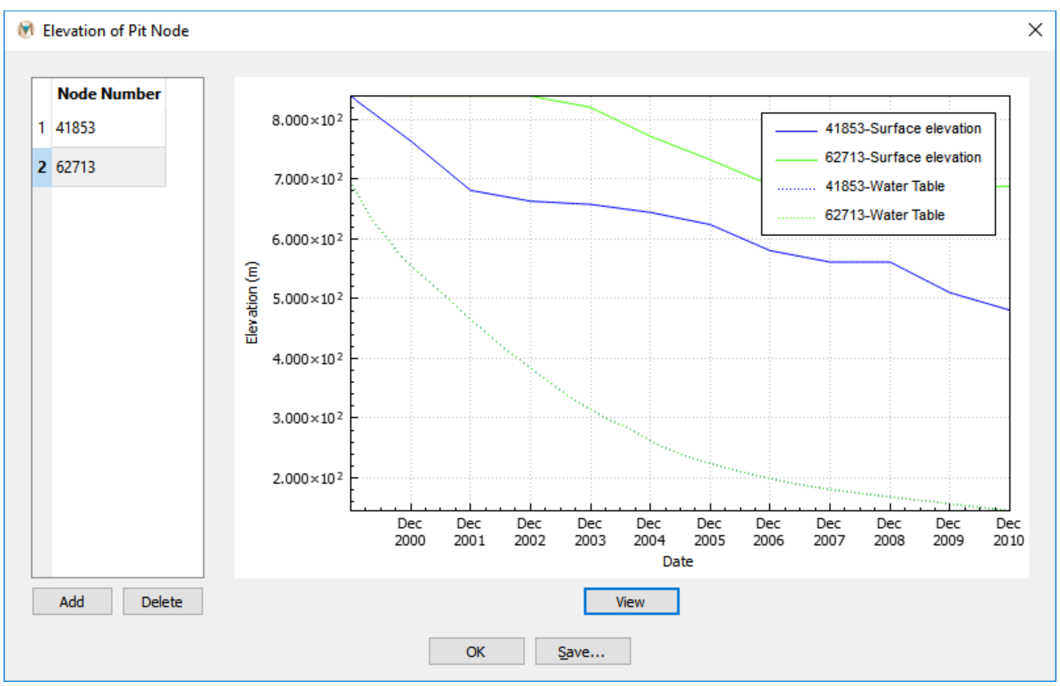


Figure 2.28 "*Elevation of Pit Node*" dialog box

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 2.29). 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.

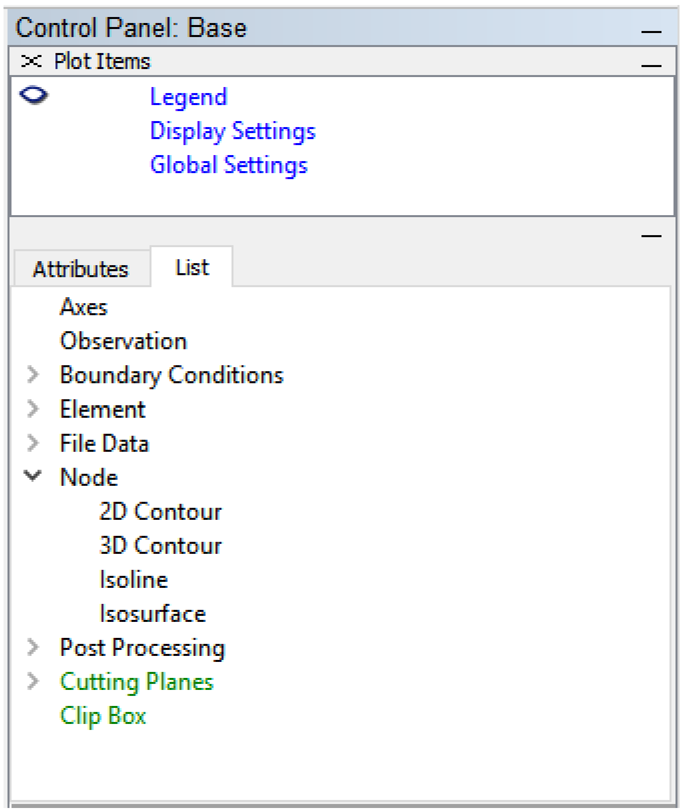


Figure 2.29. The "*Control Panel*" pane with the "*Node*" item expanded to show the "*3D Contour*" option

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 2.30.

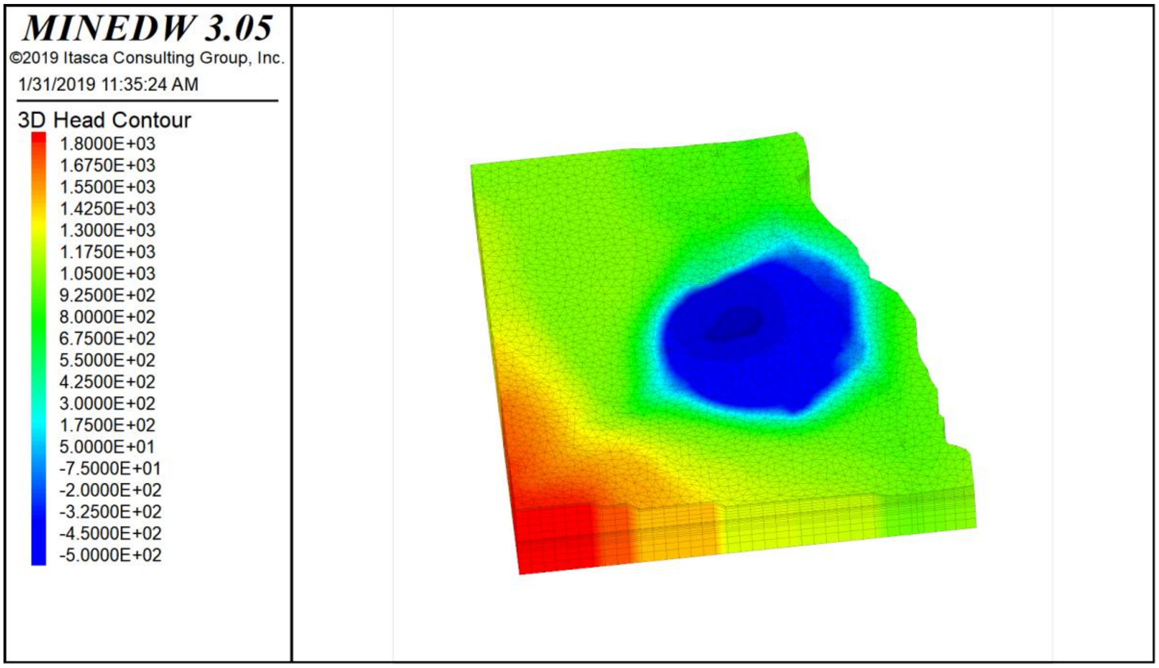


Figure 2.30. Screen display of head distribution in 3-D

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 2.31).

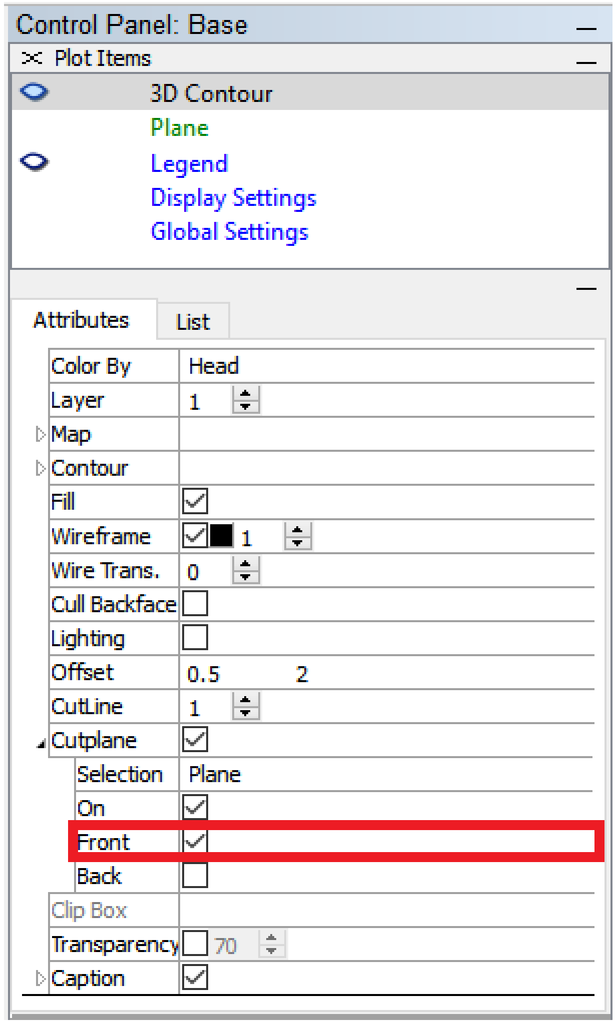


Figure 2.31. The "*Control Panel/Plot Items*" pane with the "*3D Contour*" item showing the "*Attributes*" tab

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 2.32.

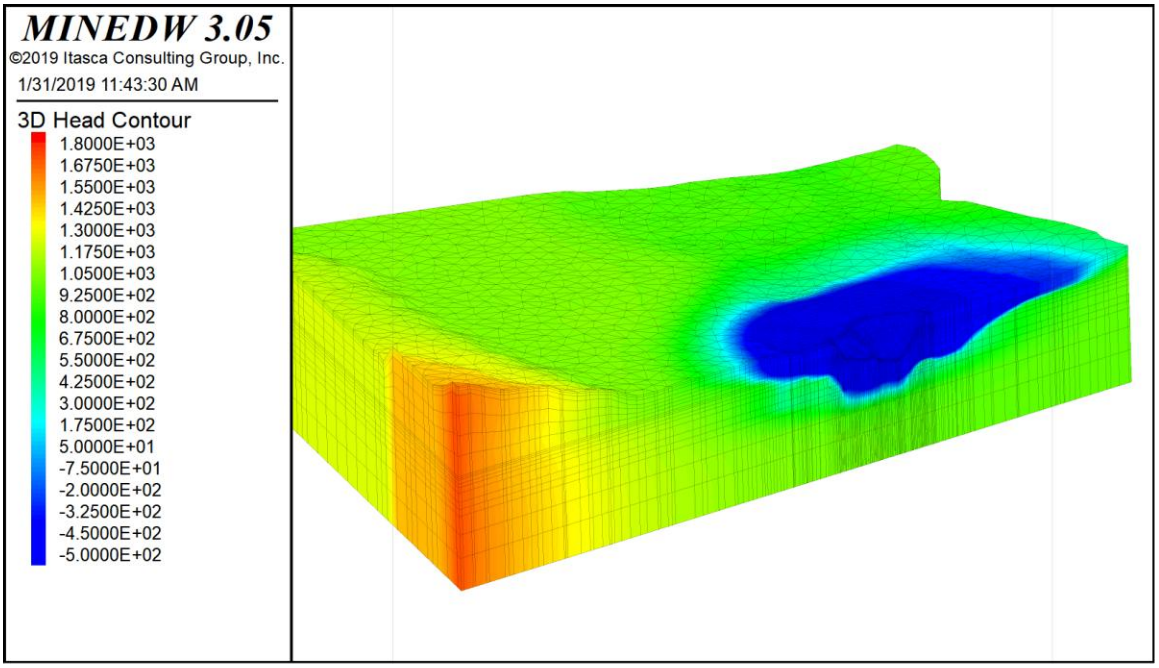


Figure 2.32. Screen display of output from the model run

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 2.33).

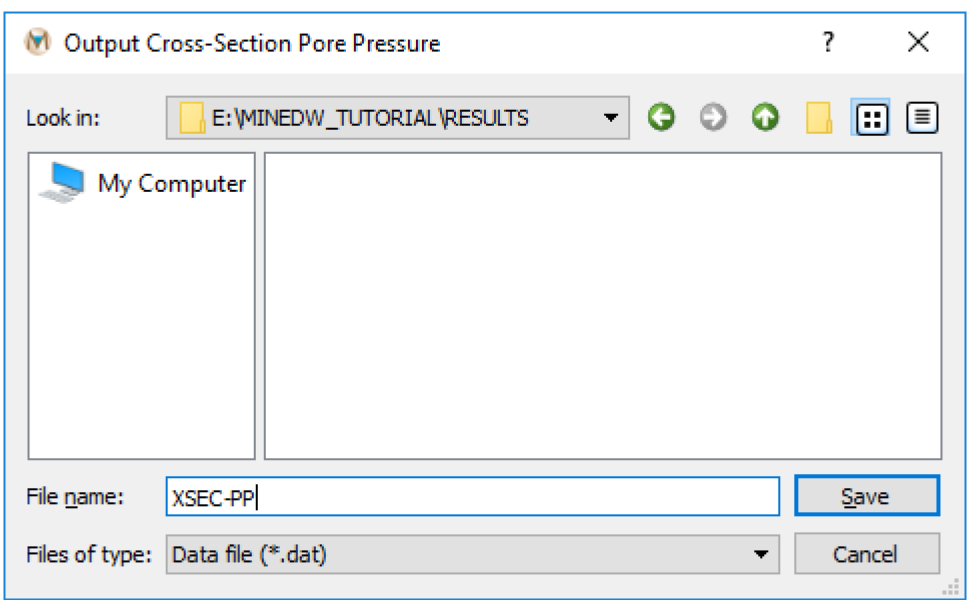


Figure 2.33. The "*Output Cross-Section Pore Pressure*" dialog box

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 2.34), 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.

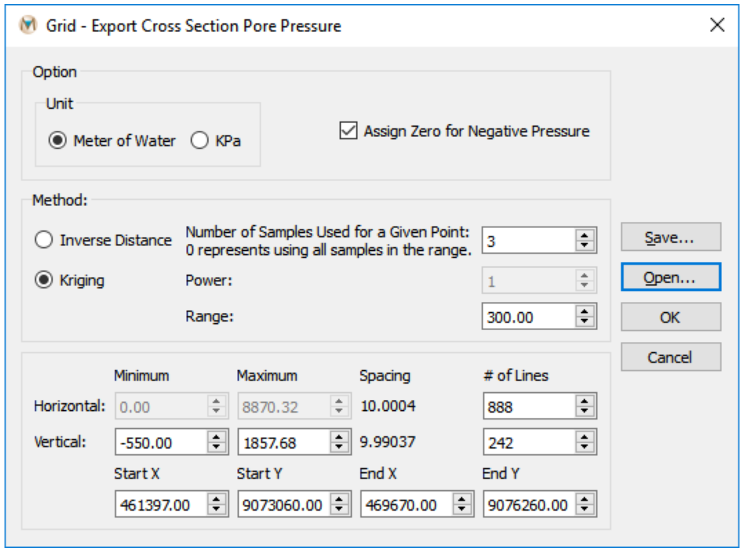


Figure 2.34. The "*Grid – Export Cross Section Pore Pressure*" dialog box

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 2.35).

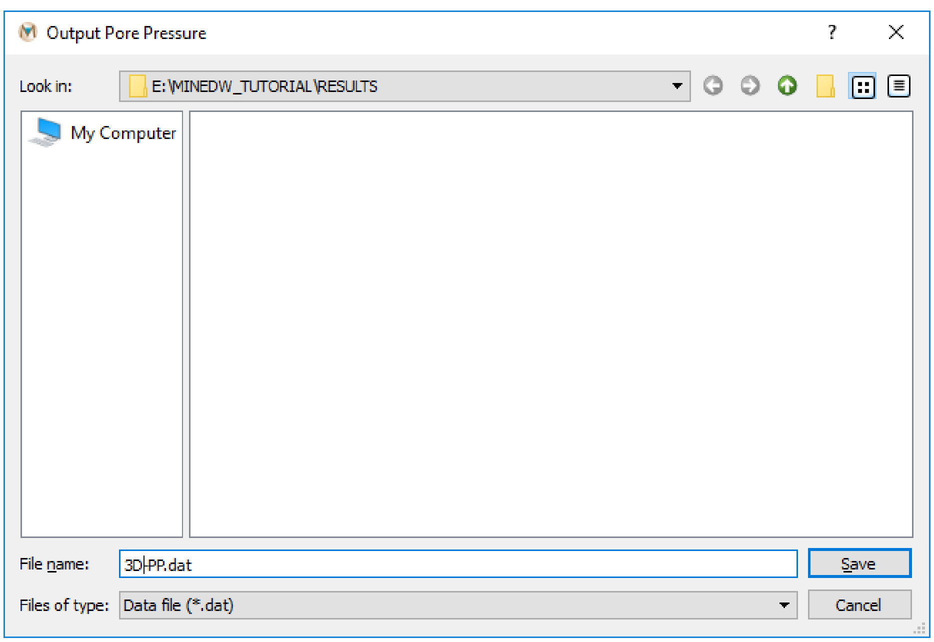


Figure 2.35. The "*Output Pore Pressure*" dialog box

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 2.36), then click "*OK.*"

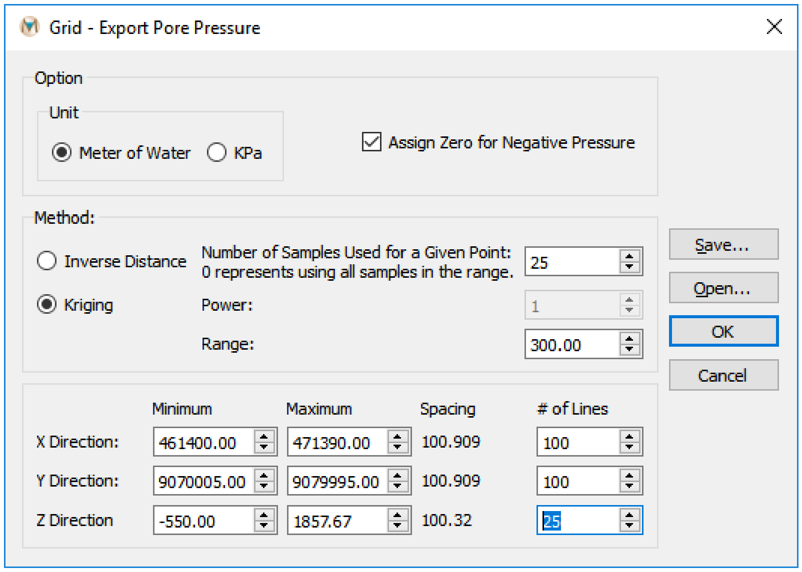


Figure 2.36. The "*Grid – Export Pore Pressure*" dialog box

The interpolated pore pressures are output as a "*.DAT" file and can be transformed and plotted using other commercial visualization software, such as *EVS*© or Golden Software's *Voxler*.

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 2.37). Select the "*.SEP" file from the folder where ***MINEDW*** was executed. Select "*Geological Units*" to determine the contributions of each geological unit to seepage.

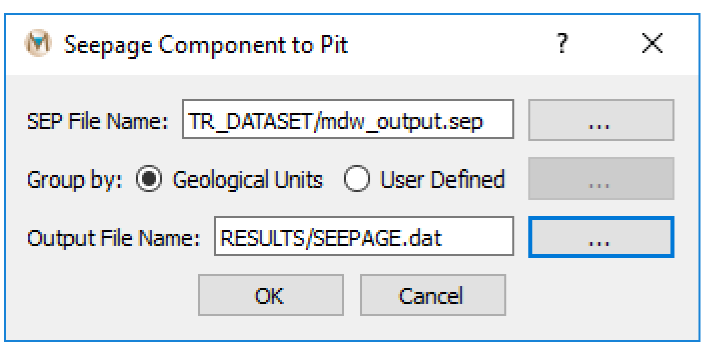


Figure 2.37 The "*Seepage Component to Pit*" dialog box

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 2.38). 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.

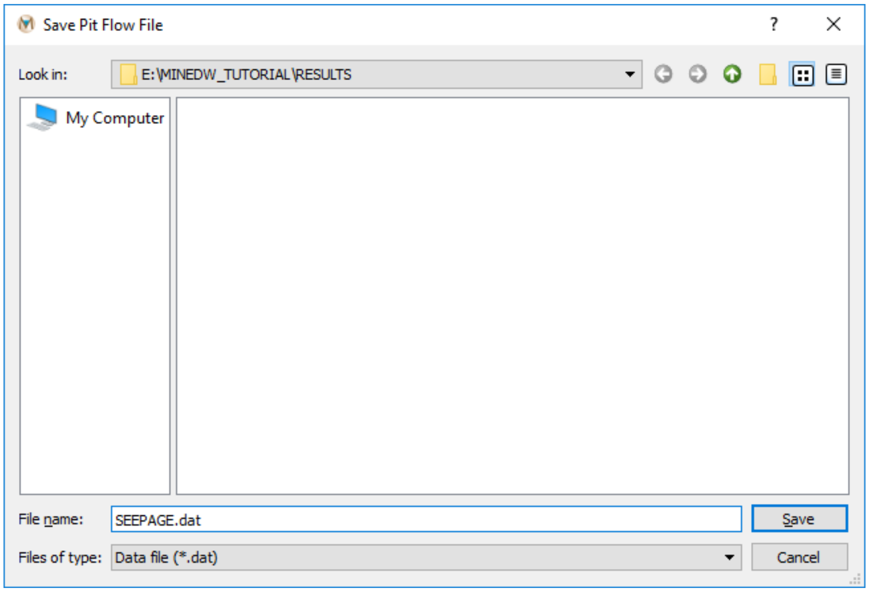


Figure 2.38. The "*Save Pit Flow File*" dialog box

The "SEEPAGE.DAT" file can be imported to Microsoft® Excel or other plotting software to create time-series plots. Total seepage vs. time (Figure 2.39) and contributions from each geological unit to the seepage through time (Figure 2.40) can be plotted.

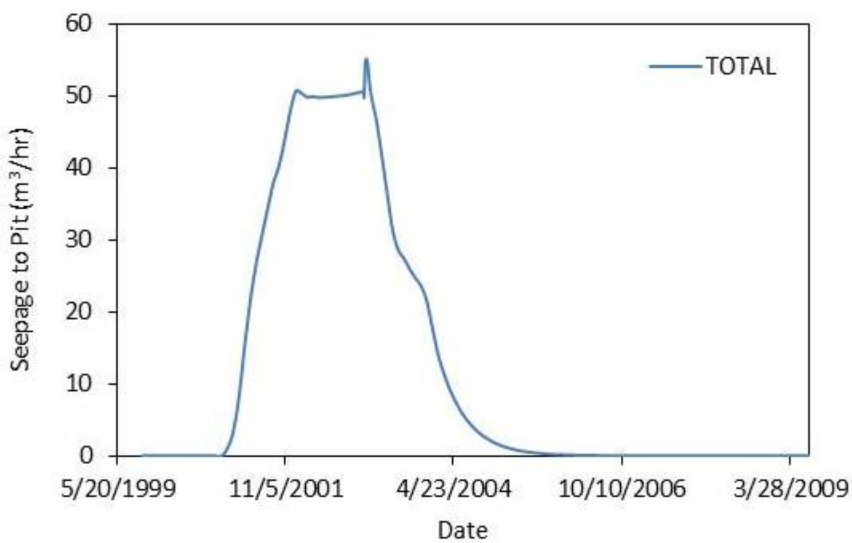


Figure 2.39. Total seepage to the pit through time

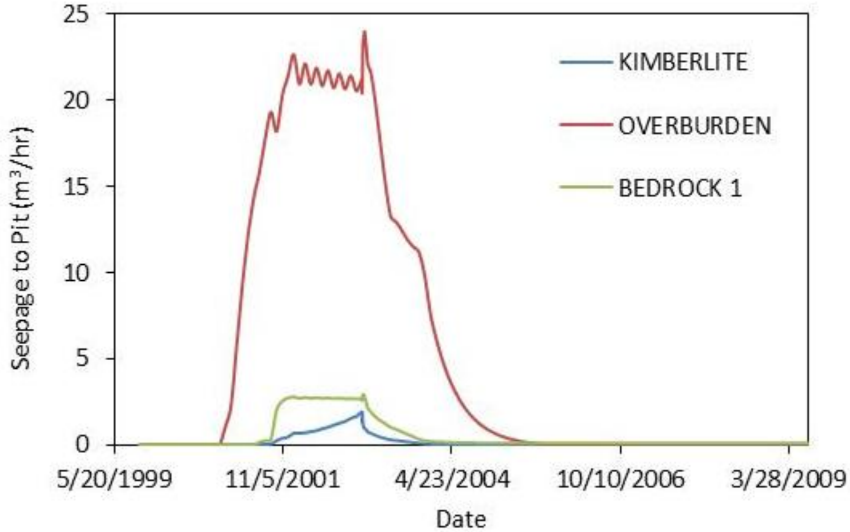


Figure 2.40. Contributions from each geological unit of seepage through time

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 2.41 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.

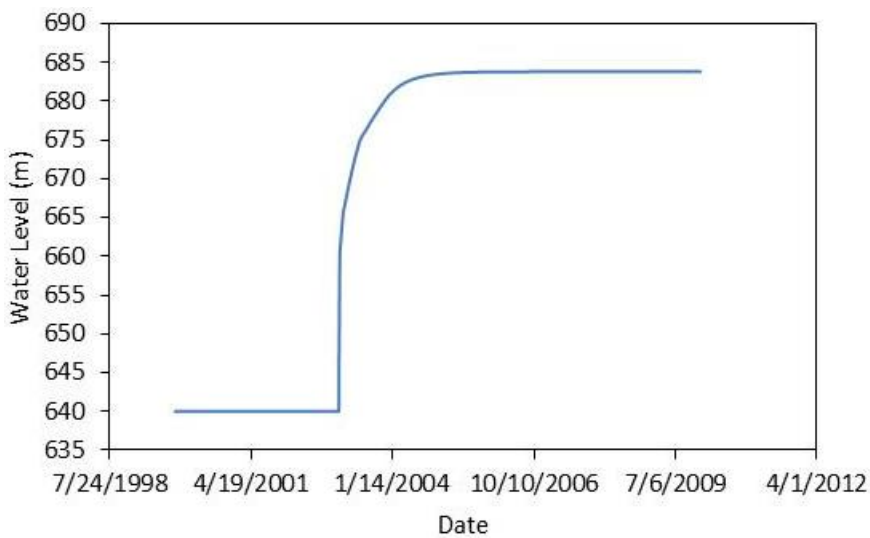


Figure 2.41. Pit-lake level changes through time

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.

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 2.42. 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 2.42. 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.

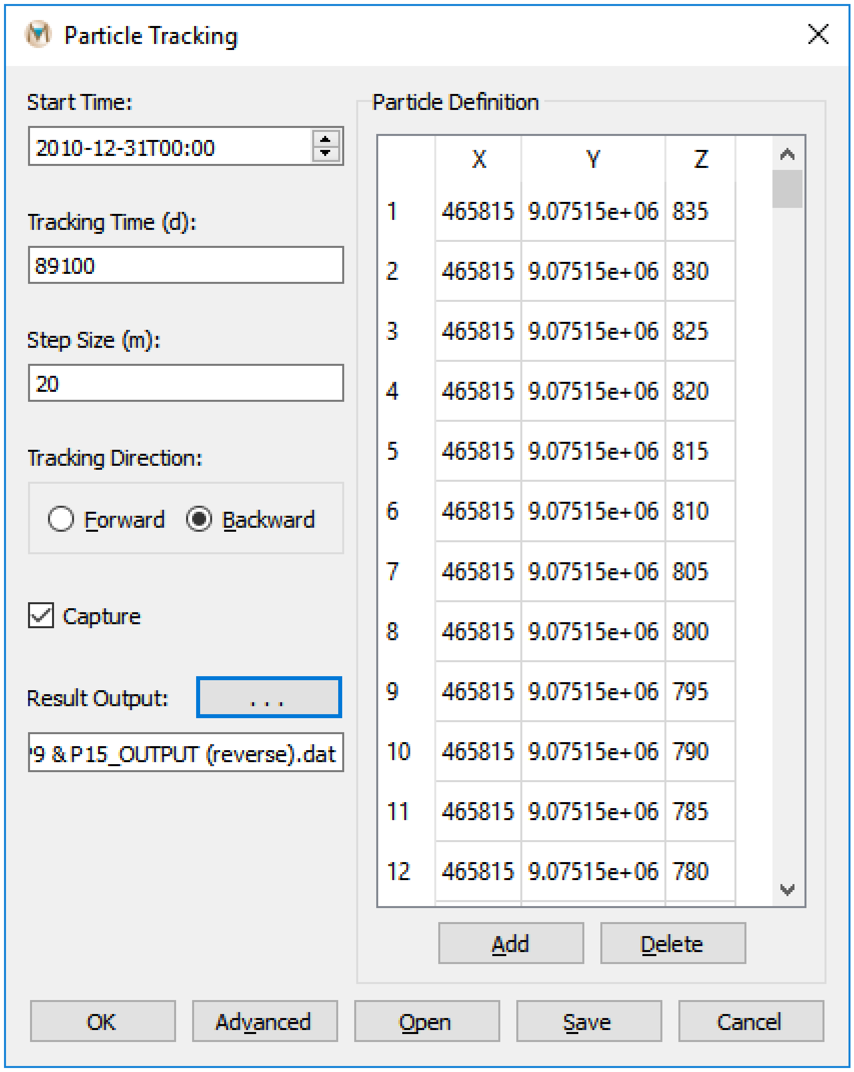


Figure 2.42. Parameters for reverse particle tracking from a pumping well

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 2.43 below should display. This same process can be followed to evaluate the capture zone of any one of the wells used in a simulation.

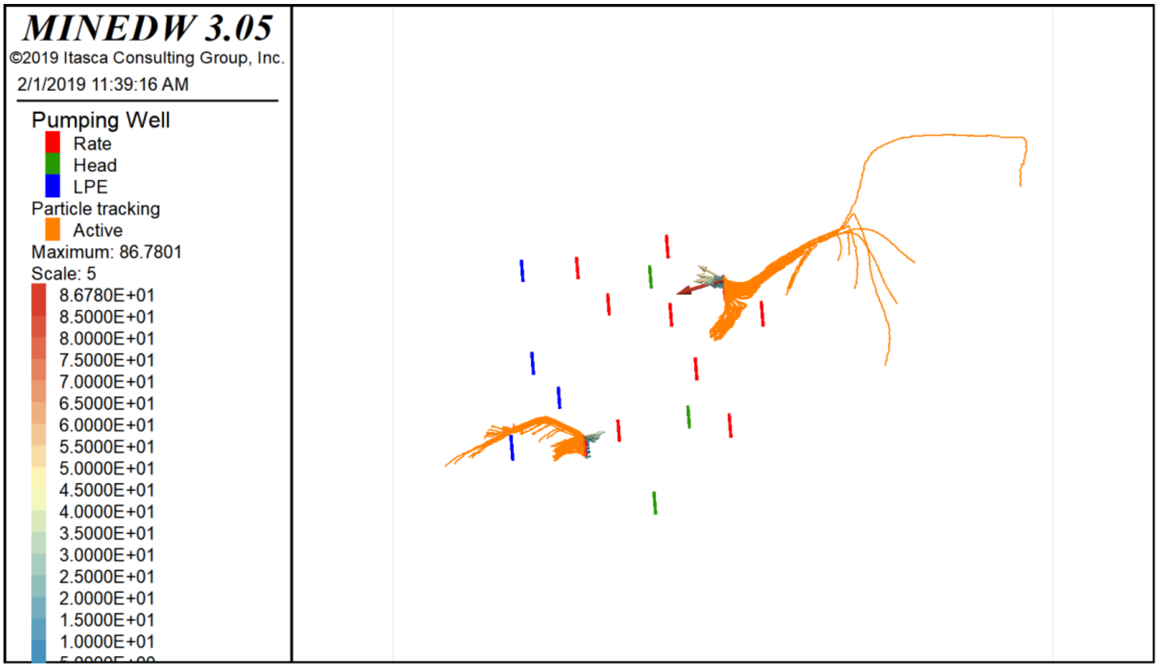


Figure 2.43. Reverse particle tracking results for two pumping wells

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 2.44 shows the results of particle tracking for the points in the *"Region_INPUT (forward).dat"* file.

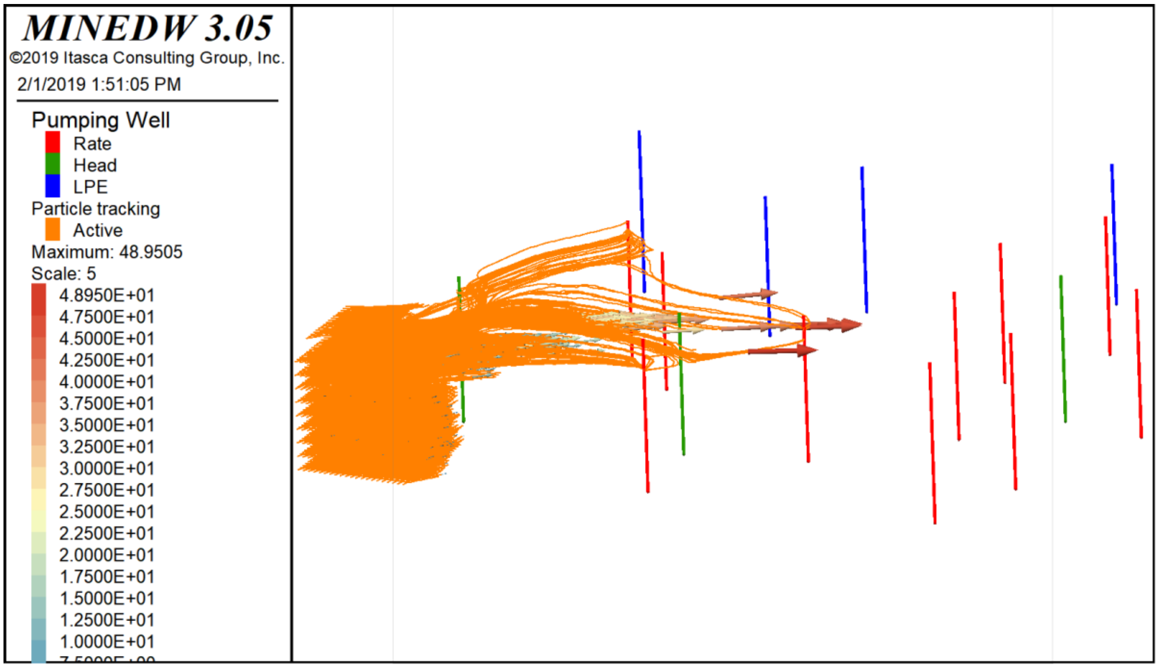


Figure 2.44. Forward particle tracks for an area upgradient of the simulated well field

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.

Tutorial 3. Underground Mining

3.1 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.

3.2 Transient Simulation

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

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 3.1.

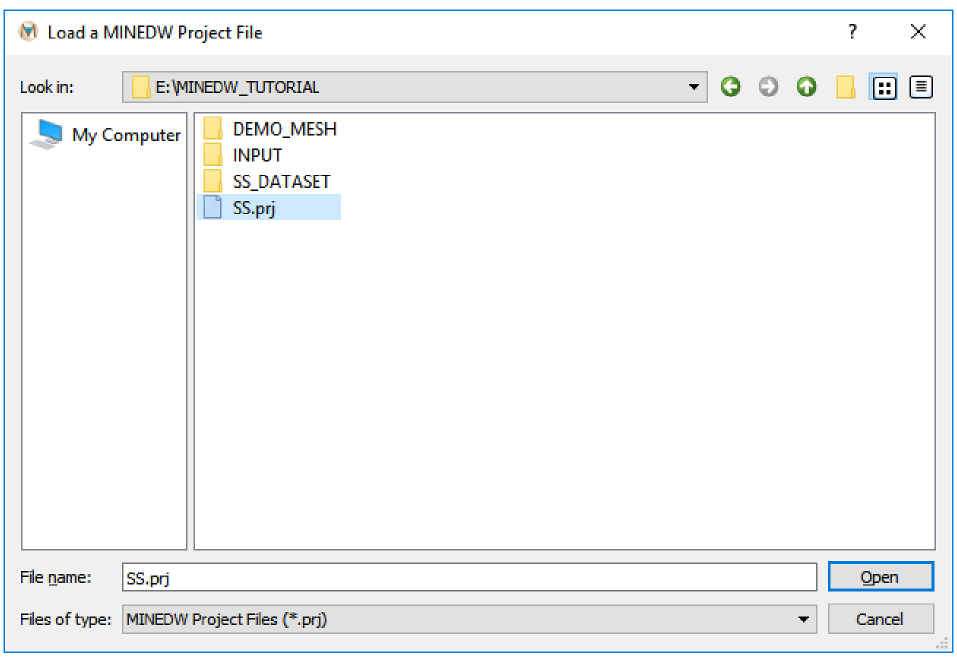


Figure 3.1. The "*Load a MINEDW Project File*" dialog box

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).

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 3.2 appears. Make sure all the fields contain the same values as shown in Figure 3.2. Select the "*Transient*" option for this simulation.

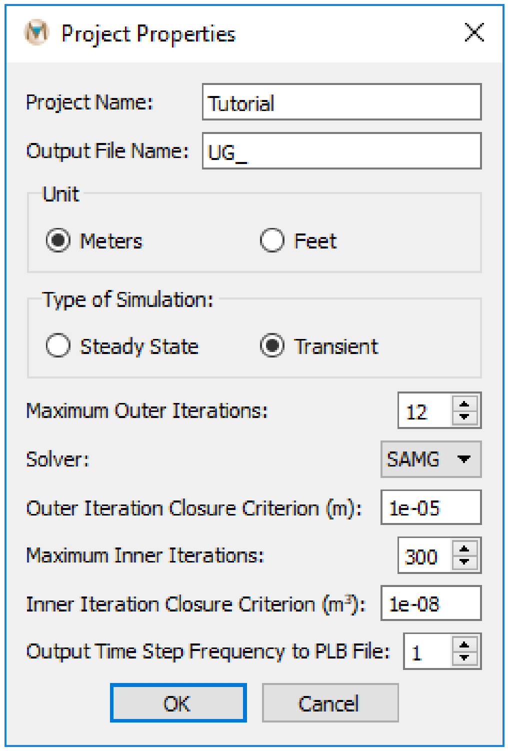


Figure 3.2. The "*Project Properties*" dialog box

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 3.3, then click "*OK.*"

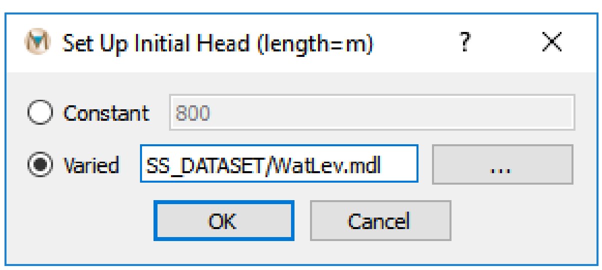


Figure 3.3. The "*Set Up Initial Head*" dialog box

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 3.4).

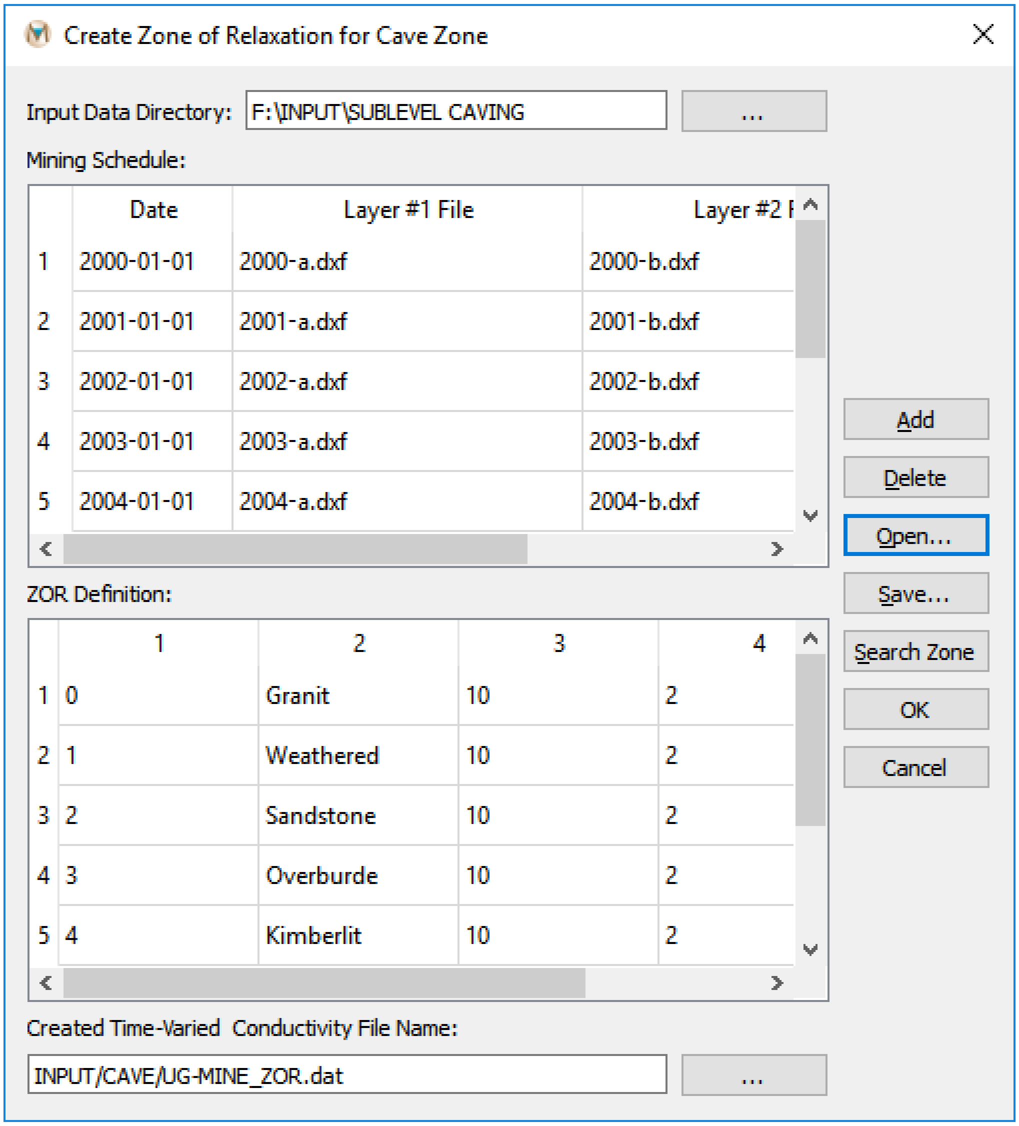


Figure 3.4. The "*Create Zone of Relaxation for Cave Zone*" dialog box

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.*"

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 3.5). 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.

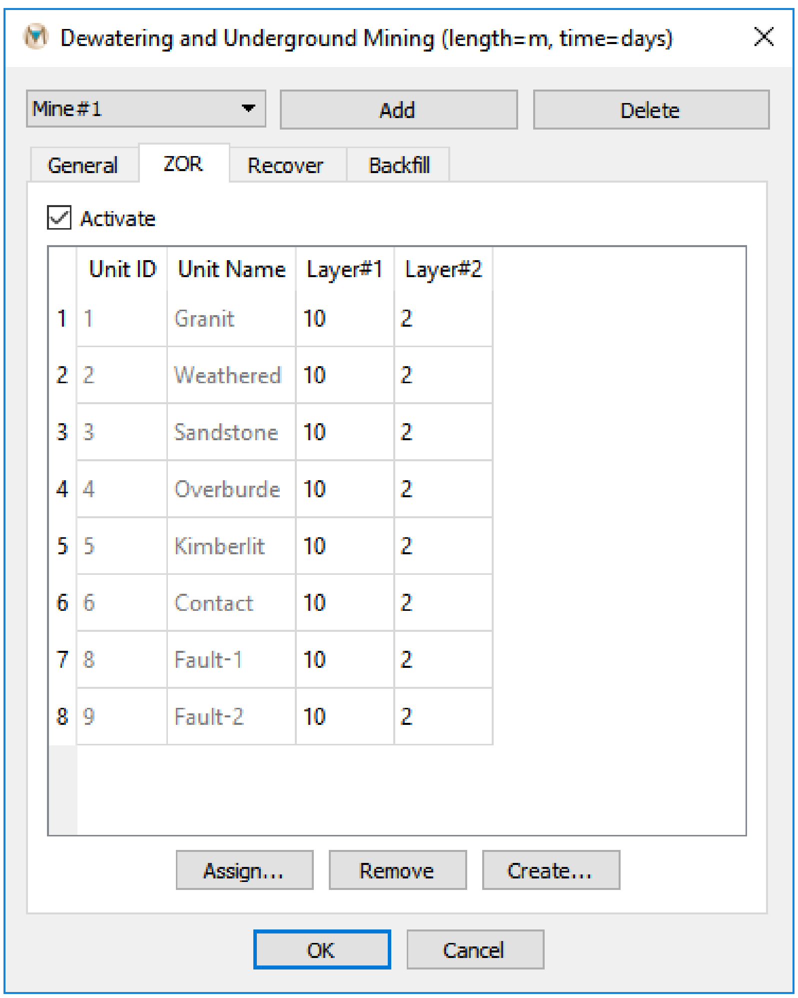


Figure 3.5. The "*Time-Varied Conductivity*" dialog box

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 3.6.

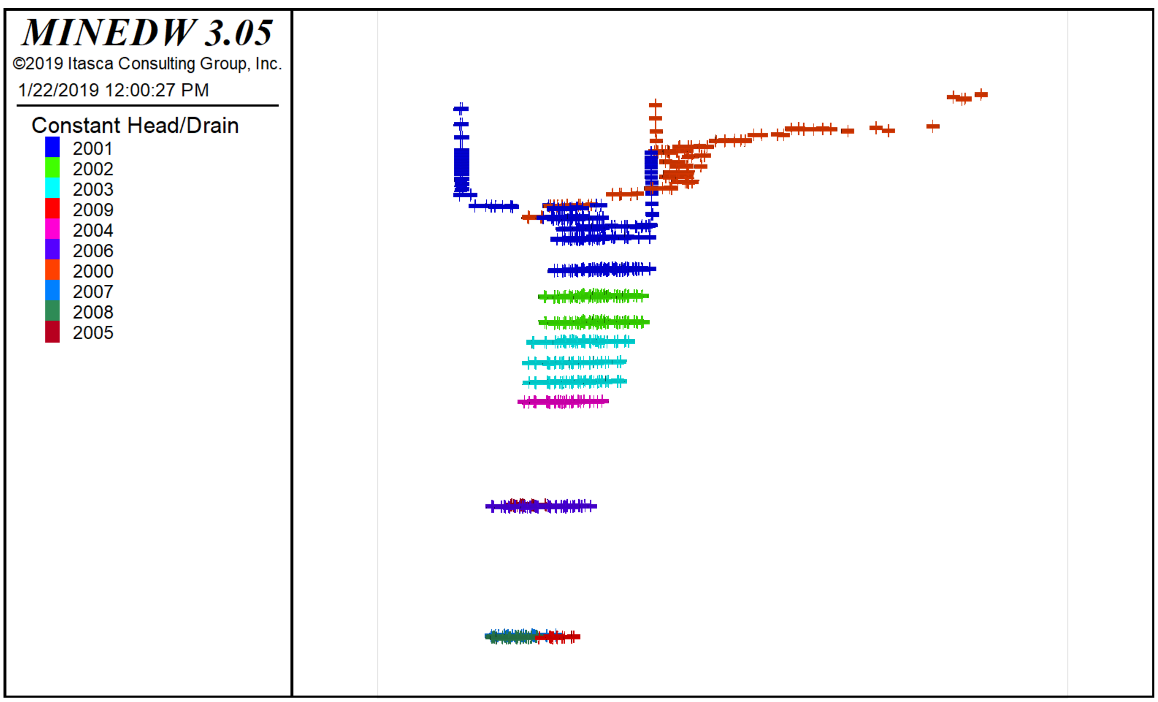


Figure 3.6. The drain nodes used to simulate the underground workings

Once the drain nodes have been assigned to the model domain, the activation time will need to 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 3.7. Now the model is ready to run. Refer to the following section for running the underground mining simulation.

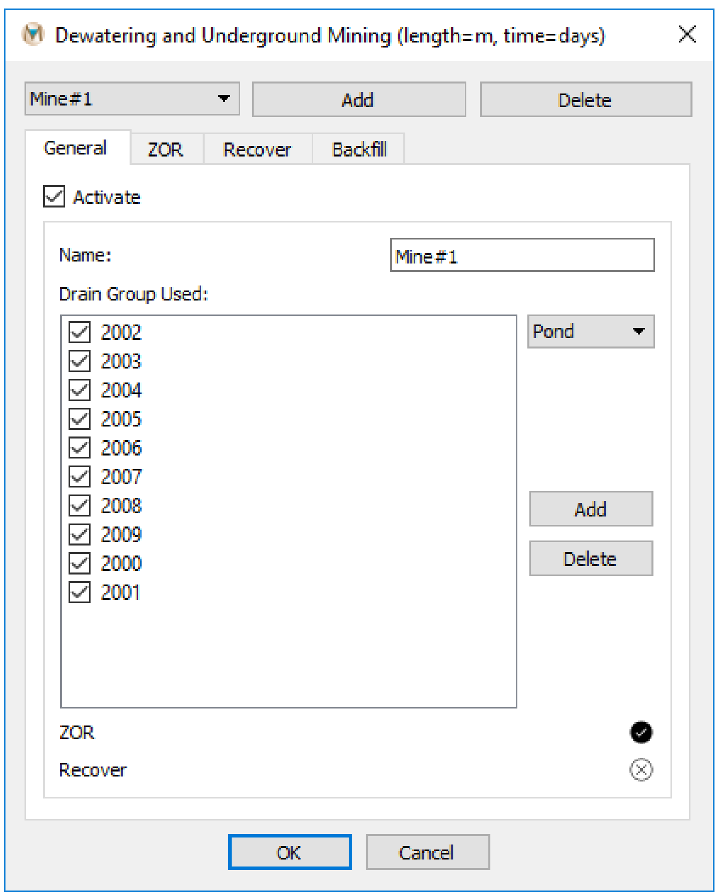


Figure 3.7. Drain groups used to simulated underground mining

3.5 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.

3.6 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 3.8 displays a cross section of head at the end of mining just before dewatering operations end.

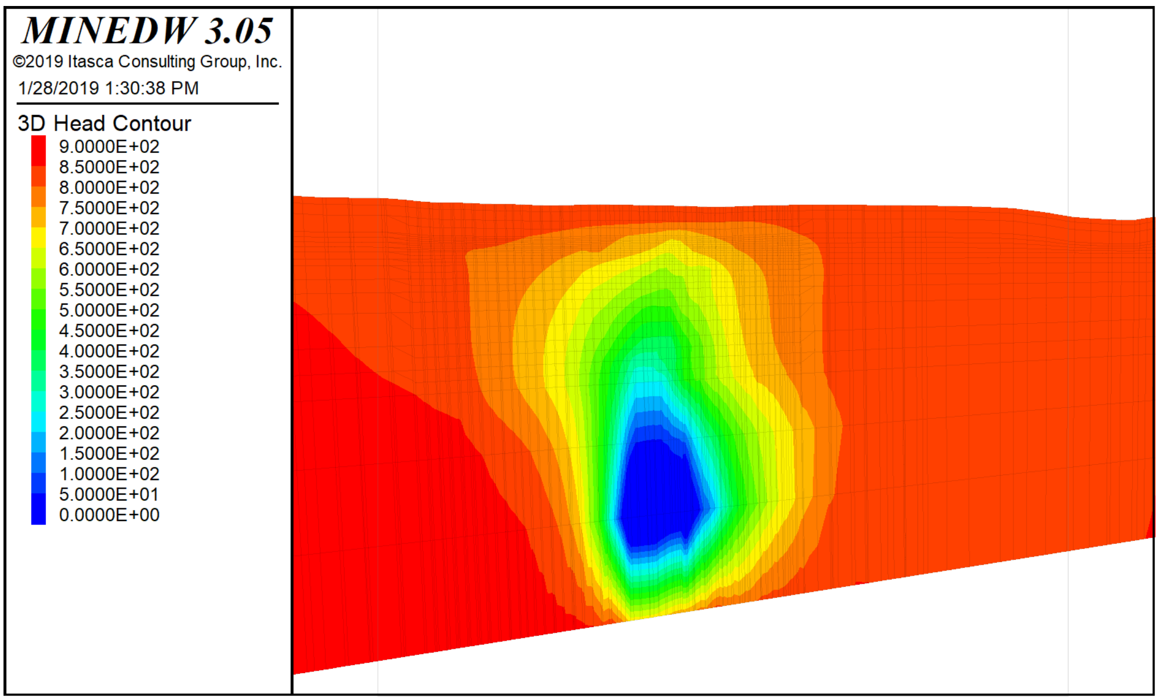


Figure 3.8. Cross section through the mining area displaying hydraulic head

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 3. 9 shows total inflow to the underground workings.

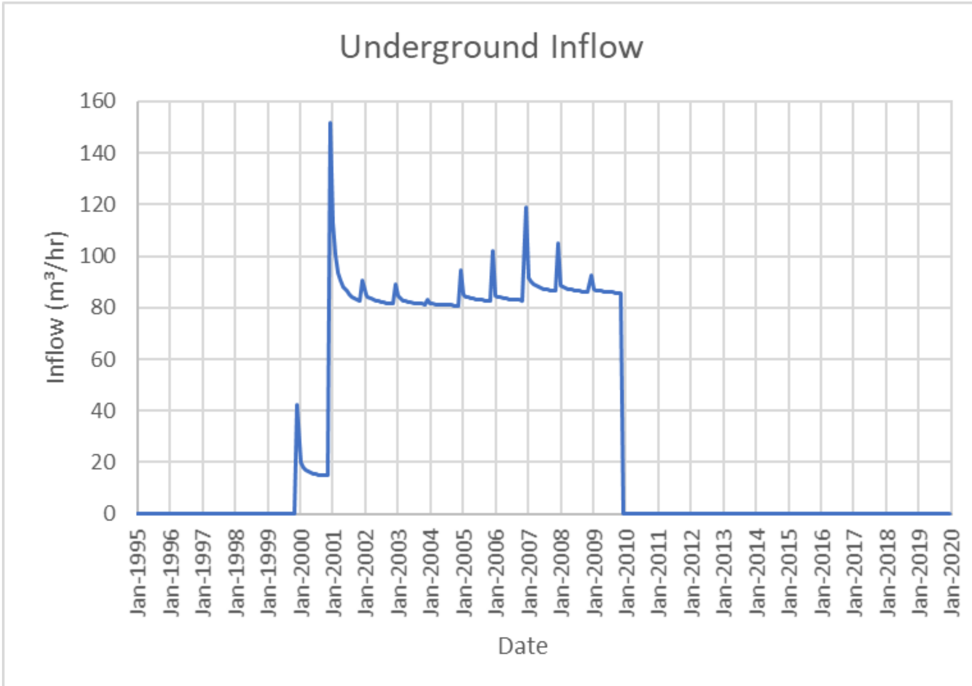


Figure 3.9. Total inflow to the underground workings through time

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."*