Integrated Computer Aided Mine Planning Software (ICAMPS)

MineFire Manual with Example

For AutoCAD 2010 and above

Revision 09/04/2018

MFIRE 3.0 Calculation Engine

Ohio Automation, Inc.
62971 Siverly Creek Road
McArthur, OH 45651

740-596-1023
Ohioauto7@gmail.com

TABLE OF CONTENTS

Brief Introduction:

Integrated Computer Aided Mine Planning Software (ICAMPS) MineFire is an AutoCAD application that offers a powerful graphical user interface to the former U.S. Bureau of Mines MFIRE version 2.20 which has been expanded upon and improved by NIOSH to include real-time modifications, now called MFIRE 3.0. ICAMPS MineFire (from here on referred to only as MineFire) is full 3D and you can use any of the AutoCAD commands you like to manipulate the drawing. However, if there is a MineFire menu option to complete the same task it is better to use that. For those who are familiar with ICAMPS MineVent the interface will look familiar and you will see many of the same powerful features. Even if you are not a MineVent user the differences below will still offer valuable information about MineFire.

The main differences between MineVent and MineFire are:

1. MineFire uses the words Junctions and Nodes interchangeably as well as Airways and Branches. MineVent referred to Airways as Branches and Junctions as Nodes.
2. MineFire is 3D so there are 3D AutoCAD options under the Display menu.
3. MineFire uses elevations for the Nodes/Junctions and 3D polylines for the Airways/Branches.
4. MineFire requires more information for Junctions and Airways. Any Junction can be in the atmosphere and you can have duplicate nodes/junctions.
5. MineVent Fan files can be loaded into MineFire but since MineFire has the fan curve fitting option once you save the fan file it can no longer be used in MineVent. There will likely be a fix to this in the future. The fix will likely be made in MineVent.
6. MineFire is a dynamic simulation over time and has three stages: Steady State, Non-Steady State and Quasi-Equilibrium. Steady State is comparable to MineVent. Also MineFire has many more parameters to show than just pressures (in.w.g) and quantities (cfm). To accommodate this and show all the information graphically the Output menu is much different and uses an Advanced Color by parameter along with posting your choice of parameters for both the Airways and the Junctions all contained in a single tabbed dialog box.
7. MineFire only has Normal, Fan, FixedQ, and Fire branch types whereas MineVent has Normal, Fan, Leak, LimitQ, FixedQ, Regulator, and Dummy. As such the leakage branch calculate resistance formula called STOP-R is not available. Also it may be that the Fire Branch type is not necessary as you can put a fire in any branch either with the Time Table or while the program is running by adding a fire event.
8. In MineFire it is best to have drawings drawn to scale and the length of the branch in Define Branches or Modify Branches match that of what is on the drawing. Although this can be overridden just as it can be in MineVent and you can enter your own length. The length in MineFire affects the calculation of things such as fume front, recirculation etc... The length does not affect the calculation in MineVent other than when calculating the resistance with a formula that uses the length. For this reason there is a menu option under the Branch Menu called Get Polyline Length to aid in checking for this.
9. MineFire manipulates a lot more information. For this reason at this point in time we would recommend not running MineVent and MineFire at the same time. Exit out of AutoCAD after running either then re-run ICAMPS and choose one or the other from the Startup menu.

10. Finally, MineFire drawings are not compatible with MineVent drawings and vice versus. We will likely have a conversion routine in a future release.

Example File and Features:

After you have successfully installed MineFire for the appropriate versions of AutoCAD (versions 2010 through 2015 32 bit and 64 bit supported) you will be confronted with the Startup Menu and you can select Open File and browse to c:\applied.rxx\datadir and open **fire_example3.dwg**. Here the xx in applied.rxx refers to the version of AutoCAD….18 for 2010-2012, 19 for 2013-2014, and 20 for 2015. Then you can use the Startup menu again and select MineFire Menu.

You will see a toolbar on the left and the main menu across the top. You can, of course, customize commands and add your own toolbars, but be sure to save them as a future update may overwrite them, as such it is not recommended to do too much customization of the menus.

The first three menu options are mostly useful AutoCAD commands for dealing with MineFire drawings. The ones to be the most familiar with are Edit→ Delete Layer, Display→Modify Layer and Display→3D Views and Display→3D Orbit.

Important Note: If you were working on your own mine map and starting fresh the first thing to do would be to go to Output→Edit Schematic and set some defaults for node sizes, branch sizes, fan file location and cfg file location. After entering these numbers insert a few nodes and check to see that the circles and numbers are set to an acceptable size. Then insert a branch between two nodes and see that the arrows/cones and blocks are an acceptable size. These settings depend on the size and scale of your drawing. Some of these can be updated later as stated in Edit Schematic others are only for New or upcoming nodes/branches.

So considering the note above go ahead and go to Output→Edit Schematic and take a look at the settings for fire_example3.dwg. You will notice that if the Size of Node Circles is greater than the Size of New Nodes and the Size of Node Numbers is large enough it puts the number closer to the circle. The two you have to initially get down is the Size of New Nodes and the size of New Branch Blocks as they are only for New and upcoming Nodes and Branch Blocks. All the other factors can be changed at any time and updated on the drawing by clicking Update DWG Exit.

Just as an example set the Size of Node Circles to 4 and the Size Arrow Cone Base Diam to 30 and hit Update DWG Exit. You will notice that the node circles are now smaller and the node numbers are a little further away from the circle and that the arrow/cones have a larger base and look more like a larger triangle, at least in Top View.

In Edit Schematic the CFG files are compatible with MineVent CFG files. The Fan file and CFG file will be created if they do not already exist. The Fan File is created from Node Menu→Define/Edit Fans and the CFG file is created from Branch Menu→Configure Branches. Here again if you were starting a new drawing you should Configure Branches first just to set up the defaults for the colors and codes for branches. Configure Branches and the CFG file really has no effect on the calculations it is mainly used for the user to assign names via the 12 different codes with colors and resistance defaults and it places these branches on the layers MFC01 through MFC12 and assigns the color and line type to those layers.

You could use Display→Modify Layer later to change the colors of all the branches in the drawing. However, be sure to change it in Configure Branches as well. So each of the 12 codes in the list can have its own name (Intake, Return, Face, whatever you choose etc.) its own layer (MFC01-MFC12) its own color (ByLayer), line type (ByLayer), and set of default numbers for resistance calculations used in Calculate Resistance from Define Branches or Modify Branch Parameters.

Configure Branches for fire_example3 is very basic and it only uses the first 2 codes named RedBranch and YellowBranch.

Now that we have the drawing open and have discussed what you should likely do if this were a new drawing let's move on to the menu options and play around a bit with fire_example3.

Edit Menu:

The options under the Edit menu have been there for a long time in all of ICAMPS. However, some could be dangerous. Such as Erase, mainly Do Not use this command to erase nodes, use the Node Menu→Erase Node. Although branches could be erased with the AutoCAD erase command you should also use Branch Menu→Erase/Delete Branch because you must be sure to get the polyline, arrow, and attribute block. So be careful with anything under the Edit Menu. The most powerful option here is at the bottom called Delete Layer, which will delete all the entities on a typed in layer name. The same task may be performed by Display→Modify Layer in AutoCAD, but sometimes for "sticky" layers this Delete Layer under the Edit menu works better.

Display Menu:

The options at the top of the list Zoom, Pan, View are standard AutoCAD commands and some are more easily accomplished with the mouse, especially a mouse with both right and left buttons and a wheel. Here Modify Layer is the most useful because, as we will see when we get to the Output, MineFire can and does (unless you intervene) put each Colored Polyline Parameter on a different layer so that you can color several parameters then go back to AutoCAD and turn on layer MFAirFlow and see the Airflow colors and then turn on MFDeltaQ and see the DeltaQ colors. This can be done for any parameter that uses the color by. The layers always start with MF but you can color on a user defined layer as well.

Since MineFire is 3D the options at the bottom of the Display menu are useful for different 3D views. Of course you may not have a mine map that has different junction elevations. The 3D orbit is nice, but sometimes it takes a while to load depending on the version of AutoCAD, your graphics card, and the size of the drawing. If you ever get lost or are in a 3D view you do not like you can always go to Display→3D Views→Top.

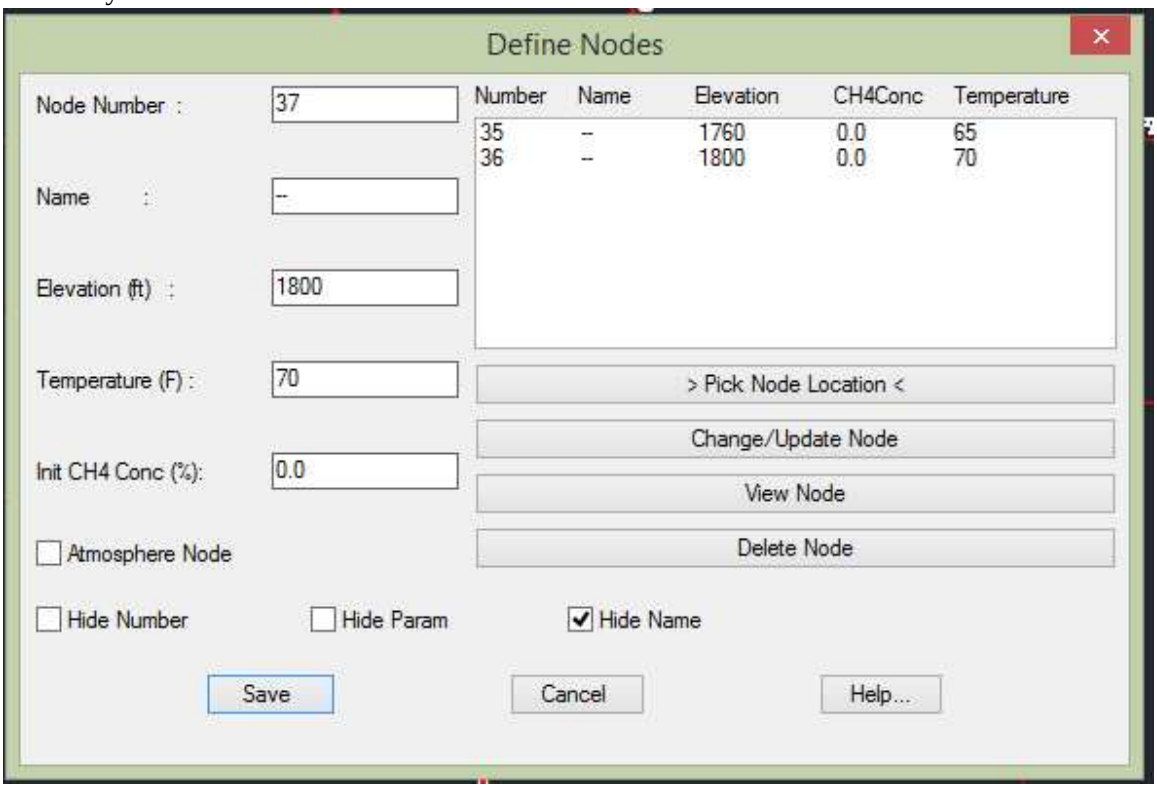
Node Menu:

→Define New Nodes:

The first option here is Define New Nodes as shown in the following dialog box. It is pretty self-explanatory, but there are a few things to be aware of. MineFire allows for Duplicate Nodes, however, you should insure that all duplicate nodes have the same information, mainly Elevation, Temperature, Initial CH4 concentration, and whether the node is in the Atmosphere.

Important Note: If you check the box next to Hide Param then when you use the Output Features it will not post the parameter on the drawing for that node. As we will see when we get to the Branch Menu

Define New Branches has the same hide feature. So if you are looking at the drawing after a posting and you do not see a number next to the node or branch it is likely because you chose to Hide it during Define or Modify.



It is advised to define all the nodes first on the drawing before connecting branches. You only need to insert nodes where branch/airway characteristics change and not for every place the airway bends.

→Modify Node Parameters

Nothing special here just use Pick Node at the top to pick the Node you want to modify or change the parameter for. **Important Note:** If you change the elevation the program will automatically update the branch polylines associated with the node so that any branch with this node as the start node or end node will be updated. It will also point to any bends in the branch and ask for the elevation and update the length in the branch attribute block as well.

→Move Nodes

Pretty much just the standard AutoCAD move command. However, do not move a node that has branches connected to it. Use Branch Menu→Move Multiple Branches or erase the nodes and branches and redo. Do not use AutoCAD commands to stretch the branch polyline as it will mess up the length for the branch and the location of the arrow and attribute block.

→Find Node

A dialog box pops up and you enter the node number you want to find. If it is in the drawing it will draw a green arrow pointing to that node. The green arrow will disappear if you zoom or pan away it is not actually drawn as any AutoCAD polyline. On the AutoCAD command prompt it will ask you if you want to Zoom Previous, if you enter N for No then it will stay zoomed at that node. If there are duplicate nodes it will point to the first one it finds and at the AutoCAD command prompt ask if want to zoom previous entering Yes or No will point to the next node it finds in the drawing.

→Erase/Delete Nodes

You should ALWAYS use this command to erase/delete nodes from the drawing and not the AutoCAD erase command as the nodes in the drawing have to be in sync with the nodes in the fnode.dat file. You can erase multiple nodes by windowing around them, AutoCAD 2015 expands on this and lets you try to draw some kind of polyline around, but it is usually best to just pick the lower left and upper right corner making a rectangle around the node(s) you want to erase.

→List Nodes

This option list all the nodes in the drawing with all the initial values and the X,Y,Z values. Nice thing here is that you can edit each node in the list or view any node in the list.

→List Unused Nodes

Here on the left it list any nodes in the drawing that are not connected to a branch. On the right it list node numbers that are available or were missed. So if you had a drawing and you inserted a node with number 2 then a node with number 5, it would list 3 and 4 as available. This is useful if you want to use a node number scheme to help you navigate the drawing.

→Check For Duplicates

Just pops up a dialog box showing the Nodes in the drawing that are duplicated.

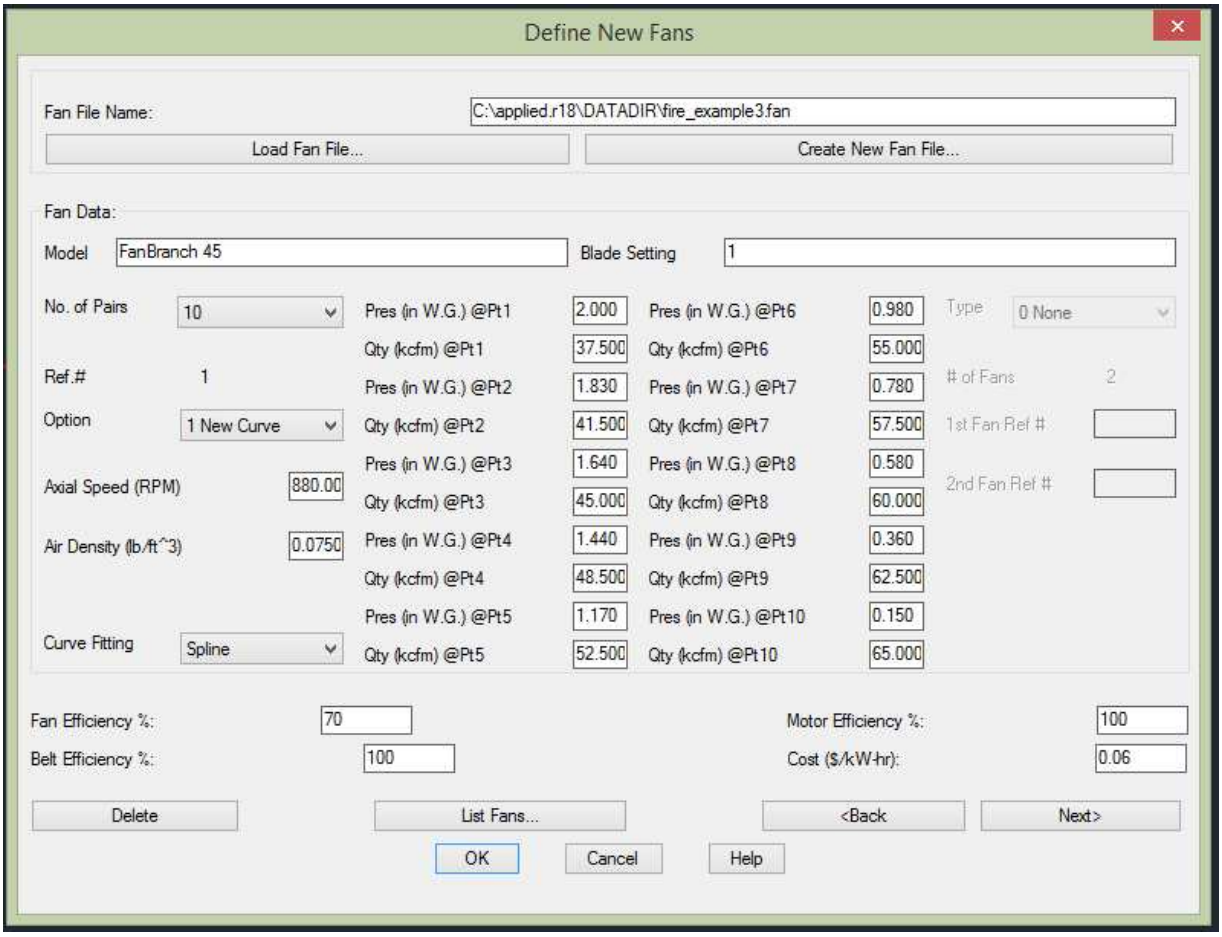
→Define/Edit Fans (Also under Branch Menu)

When you select this option the following dialog box appears for fire_example3. It loads the fan file that is set in Output→Edit Schematic. You can load a MineVent fan file into MineFire, however, if you do and you save it then it becomes a MineFire Fan file and is no longer compatible with MineVent because of the curve fitting option. This will likely be fixed in some way in the future, but will probably be a fix to MineVent and not MineFire. The program will warn you if you try to load a MineVent fan file. Also if you use a MineVent fan file for the Run MineFire part of the program you will get a message that says the curve fitting option will be set to 1 for Least Squares.

So starting from the top the Fan File Name edit box shows the current fan and the path to that fan file. Below that is Load Fan File and Create New Fan File. If you Load Fan File it will not save the changes to the existing fan file without you exiting out clicking save when it ask and coming back in. If you opt to Create New Fan File it will allow you to save the existing fan.

Model and Blade Setting are for the user's information purposes only. The Number of pairs tells how many points you want on the fan curve. The reference number (Ref#) is very important because this is what ties a fan branch to a fan curve. The Option drop down allows you to create a new curve or modify/combine two (or one) existing curves. If you select the Option Modify Curve then on the right Type becomes available and you can choose to create an equivalent fan curve of Unequal in Series, Unequal in Parallel, Equal in Series and Equal in Parallel, or based on a Speed/Density change.

The rest of the options are just entering the pressures and quantities for the fan curve. Note that the quantities are kcfm. At this point and time the Fan Efficiencies and cost are not used for MineFire.



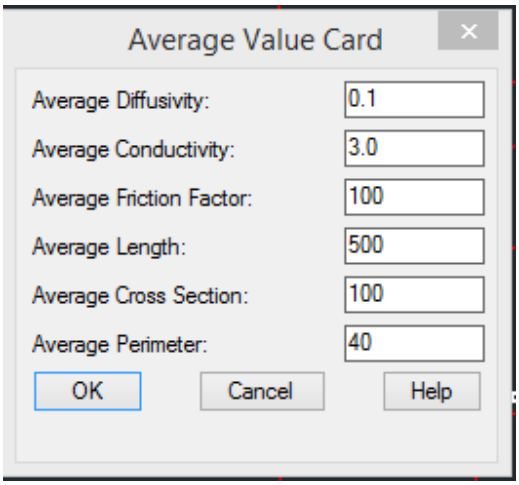
→Run MineFire (Also under Branch Menu)

Now to the heart of the program, when you select this option the dialog below appears for fire_example3. Note that depending on your version of AutoCAD and where fire_example3 is located it may ask for the location of the Fan and CFG files, clicking the OK button for the defaults should work. If not you can change them in Edit Schematic, otherwise you will be presented immediately with the Cerate MFIRE 3.0 Input Files dialog which simply the first phase of Run MineFire.

1. **Fan File Name to Use**: is the fan file that will be used for the calculations this is set based on whatever is set in Edit Schematic and not saved as all the other data is. Below that you can see the button to Browse Existing Fan Files which will let you use a different fan file for this session. If you browse for another fan file it will NOT retain this for the next time you Run MineFire. So if you want to use one fan file more than others please set it in Edit Schematic.
2. **Fan Curve Boundry**: 1. Gradient Both Ends: the curve is extended following it's gradient at both ends. 2. Left Boundry Gradient extended following gradient, right boundary is sent to zero. 3. Both Boundaries Zero sent to zero.
3. **Legacy Mode**: Check this box to use the legacy mode of MineFire which more closely resembles the original MFIRE 2.20 from the U.S. Bureau of Mines. This option will run not allow you to Add Events (change resistance, modify fan, add fires etc..) in real-time. Also does not allow you to change the speed of time is set to the maximum.

4. **Continuous Mode**: Check this box to make the simulation run continuously until you tell it to stop or End. This option cannot be used with Legacy Mode. It is not advisable to set continuous mode then leave for lunch or several days because MineFire manipulates a lot of data and would create huge files and consume computer resources.
5. **Omit Junctions**: This is disabled and will likely stay disabled because it omits the node/junction data and would result in just the network calculations being performed and could cause some of the Output features to read the file incorrectly or at least it would not have much of the data you would be interested in.
6. **Network Only**: This is disabled and will likely stay disabled for the same reasons mentioned above in Omit Junctions. There may be a need to only want the Network results, but why not just do it all anyway?
7. **Reference Temperature**: is the reference or air average temperature in the mine in degrees F. If the input value of Reference Temperature is less than 10 F or higher than 110 F., the default value of 75 F. will be used.
8. **Max Temperature Iterations:** is the maximum number of iterations in the temperature section. If its input is less than 10 or larger than 80, the default value of 15 will be used.
9. **Max Dynamic Iterations:** is the maximum number of iterations in the dynamic simulation section. If its input is less than 10 or larger than 80, the default value of 15 will be used.
10. **Temperature Only:** When Yes is selected only network balancing with consideration of temperature effects are performed. Select No for the simulation to proceed to transient state and fire modeling. Recommended setting is No.
11. **Time Increment:** is the time increment for the dynamic simulation section in seconds If the input value is 0 or less, default value of 15 sec. will be used. Once the program starts running this is when it will report for the Non-Steady State. Note: If there are not significant changes for a Time Increment then data in the report may be omitted.
12. **Time Span:** is the time span for the dynamic simulation of the proposed event in minutes. The default value is 5 minutes. This is how long the program will run unless you chose Continuous Mode.
13. **Output Detail:** is used to define the level of detail of the output file. The Options are Brief; Normal; Detailed; and Most Detailed. Currently disabled and set to Most Detailed to ensure that all the Output features have a consistent output file to read from.
14. **Time Interval For Report:** is the time interval for output in minutes. If events are changing significantly, it is not followed. Needs to be the exact multiples of Time Increment, otherwise it will use the value of Time Increment. Just use Time Increment.
15. **Transient State:** used during development it is disabled and set to No. Certain information would be omitted in the output file if set to Yes.
16. **Reference Density:** is the reference density in pounds per cubic foot of air corresponding to the reference temperature.
17. **Average Value Data Present:** If set to Yes an average value card is present in the data file to supply required values which may have been omitted in the data file. Use the button at the bottom Average Value Data to set these numbers.

18. **Start Junction:** is the ID number of starting junction in the ventilation system. At the start junction (usually the atmosphere), conditions never change.
19. **Start Junction Temperature:** is the air temperature at the starting junction in deg. F.
20. **Time Equilibrium:** Time span to assume quasi-equilibrium in hrs. Default 5.
21. **Fume Criteria:** is the accuracy criterion for fume calculation in %. The default 0.005.
22. **CH4 Criteria:** is the accuracy criterion for methane calculation in %. The default 0.01.
23. **Temperature Criteria:** is the accuracy criterion for temperature calculation in deg. F. The default value is 0.1.
24. **Pressure Drop Warning Limit:** is the pressure drop criterion in in. w.g. for issuing a warning message. The default value is 0.01.
25. **Fume Warning Limit:** is the fume concentration criterion in % for issuing a warning message. The default value is 0.05.
26. **CH4 Warning Limit:** is the methane concentration criterion in % for issuing a warning message. The default value is 1.0.
27. **Temperature Limit:** is the high temperature criterion in deg. F for issuing a warning message. The default value is 100.0 degrees F.
    The data items 21 through 27 serve as calculation accuracy criteria or warning message issuing criteria. In most cases, the available default values will be suitable. To activate the respective default values, place a zero at that location in the dialog.
28. **Average Value Data:** Use this if you chose Yes from the Average Value Data Present to set the average value data.

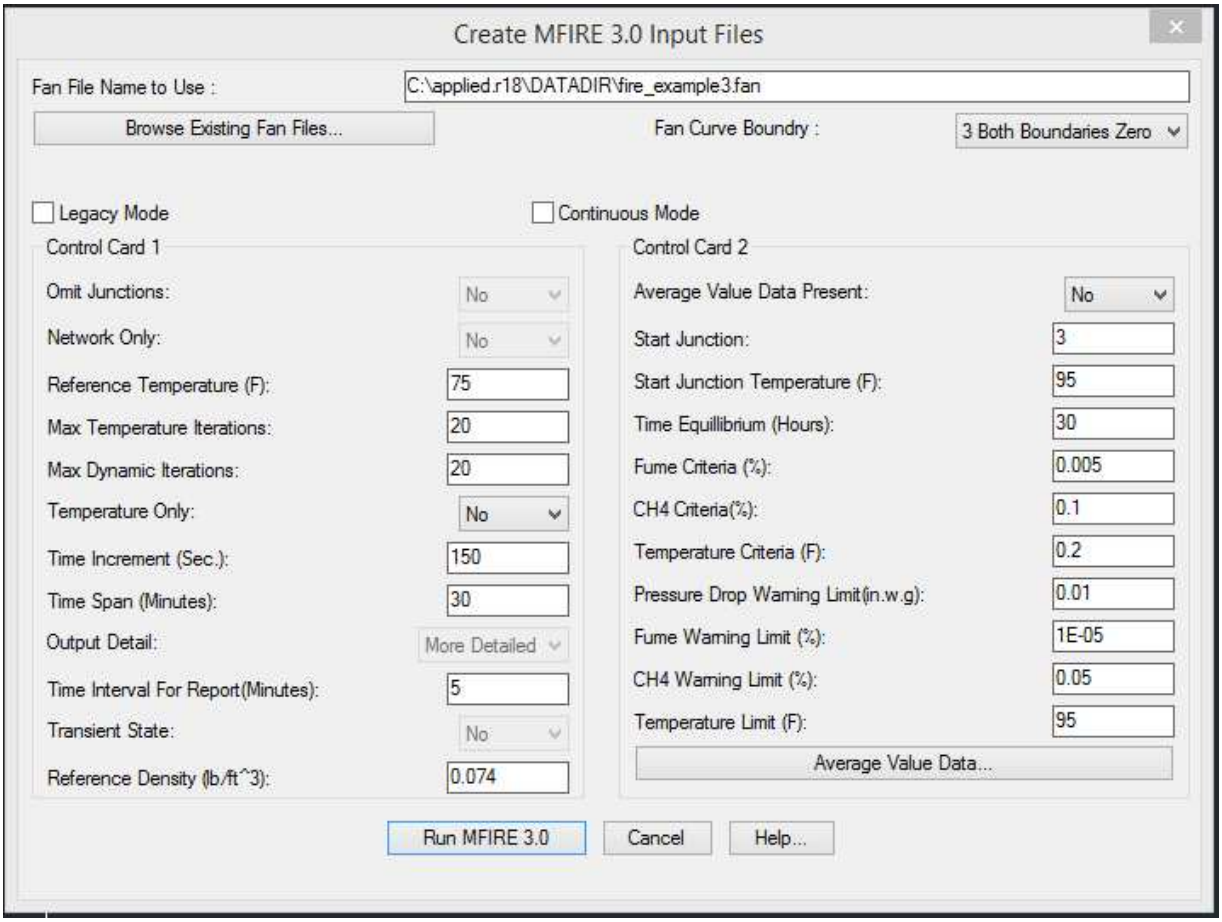


   a. **Average Diffusivity:** is the average thermal diffusivity of rock in ft_2/hr. The default value is 0.1.
   b. **Average Conductivity:** is the average thermal conductivity of rock in Btu/hr*ft*F. The default value is 3.0.
   c. **Average Friction Factor:** is the average friction factor in 1.0*E-10 lbm*min2/ft4. The default value is 100.

d. **Average Length:** is the average length of an airway in ft. The default value is 500. This would likely never be used as MineFire always puts in a length unless you set the length to zero.
e. **Average Cross Section:** is the average cross sectional area of an airway in ft_2. The default value is 100.0.
f. **Average Perimeter:** is the average perimeter of an airway in ft. The default value 40.0

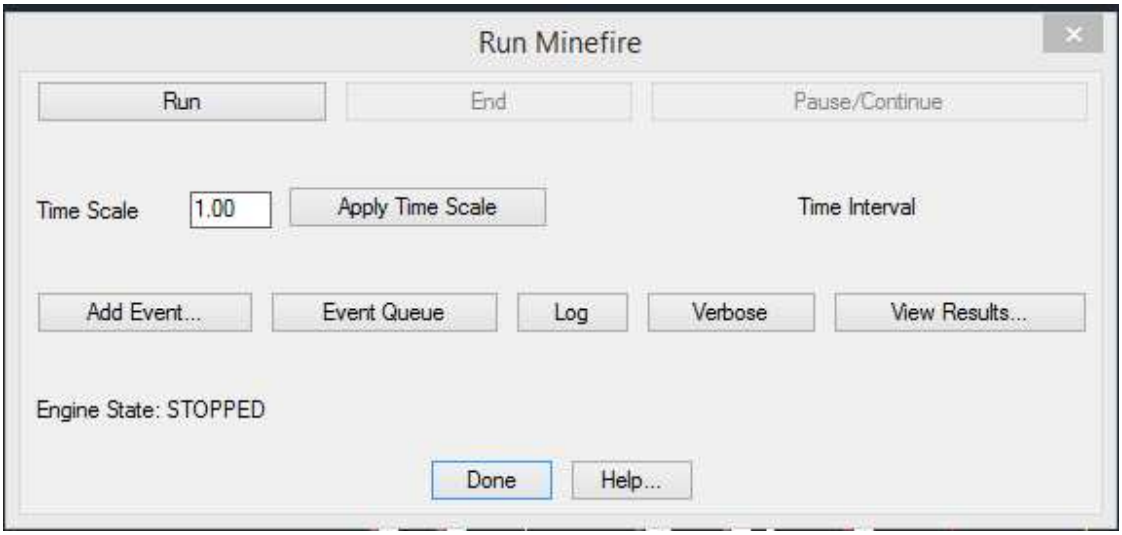
To select a default value, use zero for the respective data item. If the entire set of default values are to be used, set Average Value Present at the top to No.

Finally, below is the dialog for fire_example3 with some reasonable defaults set. Unless Continuous Mode is checked the program is set to run for 30 minutes and report every 150 seconds in the Non-Steady State output as set by Time Increment.

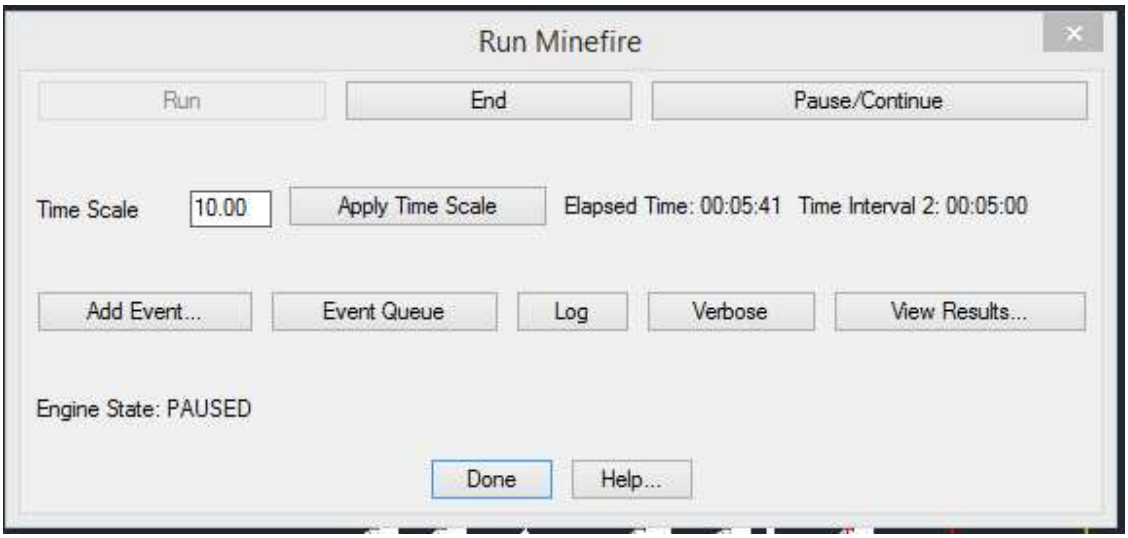


Now for the fun part. Click on **Run MFIRE 3.0** and you should get the following dialog box, provided you are using fire_example3. Now you could click the Run button right away to start the program, but let's wait a minute. The Time Scale is initially set to 1, which in essence means that you would have to wait the full 30 minutes in real-time to for the program to finish. So go ahead and set the Time Scale to 10 and Click Apply Time Scale. You can change the speed while the program is

running, by entering a Time Scale and clicking Apply Time Scale. Then go ahead and Click Run. You will see the Elapsed time and the current Time Interval/Increment. As well as the options at the top to End or Pause/Continue. Pause at about 5 minutes and 41 seconds. You can slow the Time Scale down if you want to be exact. Below the time Scale you will see Add Event (unless you are in legacy mode), Event Queue, Log, Verbose and View Results. The most powerful of these and maybe most useful are Add Event which allows you to add or change events in real-time and the View Results, which lets you graphically and colorfully view the results on the drawing. All of these options below Time Scale will Pause the program temporarily while you check things out and then continue automatically once the option returns to the main screen, unless you click Pause yourself before you click one of the buttons below Time Scale.

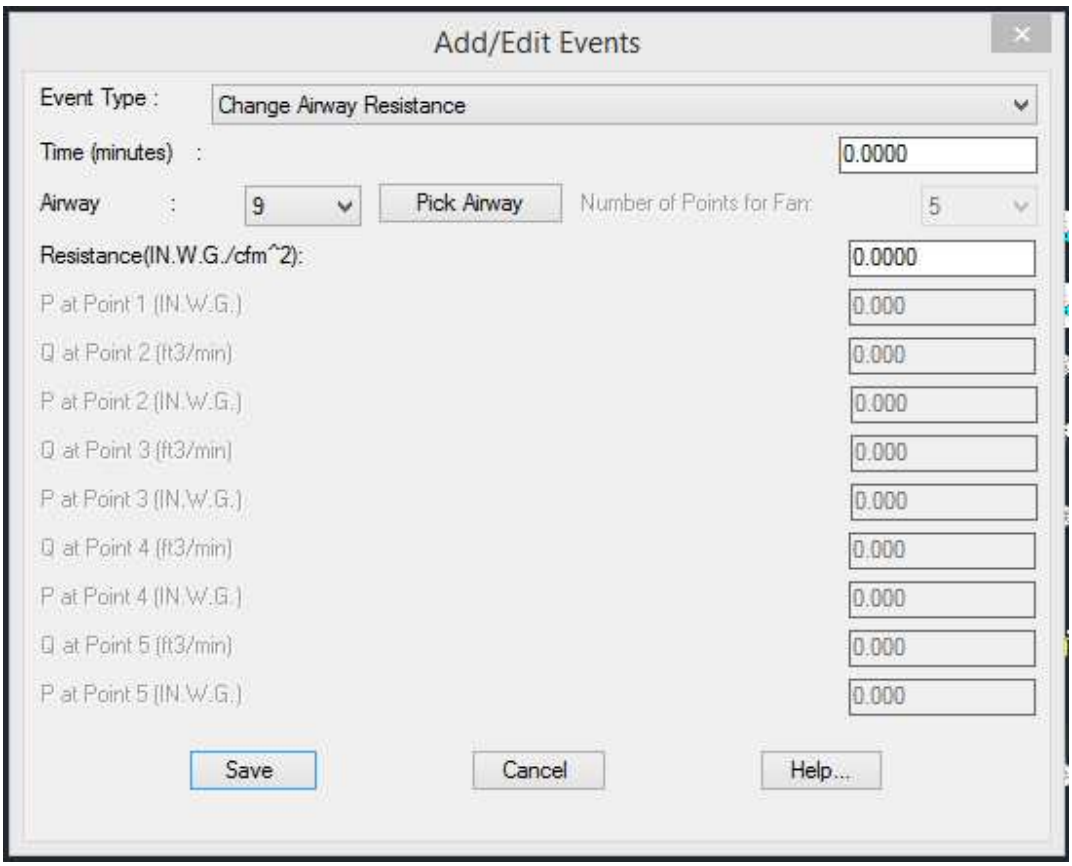


Here is what it looks like at 5 minutes and 41 seconds into the simulation:



I paused it here, just for example and to Add Event and show the Event Queue. You can see at the bottom that it always tells you the current engine state even if an error occurs so keep an eye on that. Now for the main buttons and what they do:

Add Event: This allows you to add/change events in real-time. You can add events at predetermined times by using the Time Table under the Branch Menu. Below is the dialog box that comes up to Add an Event it is the same one that is used for Time Table.



The drop down at the top is to select the type of event you want to add or change. The options are Change Airway Resistance, Edit Fan, Add Fire, Remove Fire, Change Time Increment, and Change Output Time Interval. The edit boxes below will enable or disable depending on the selection here.

Important Note: For Editing a Fan the quantities are NOT in kcfm as they are in Define/Edit Fan but rather cfm.

Important Note: Time edit box below the Event Type dropdown must be later than the current simulation time or it would have no effect, so in this example, it would have to be something greater than 5 minutes and 41 seconds. Although you must enter the time as a decimal number, so anything greater than 5.67 minutes in this box.

Below the Time is the Airway you want to make this change on you can either pick it from the drop down list of all the airways or select the branch attribute block from zooming around the drawing.

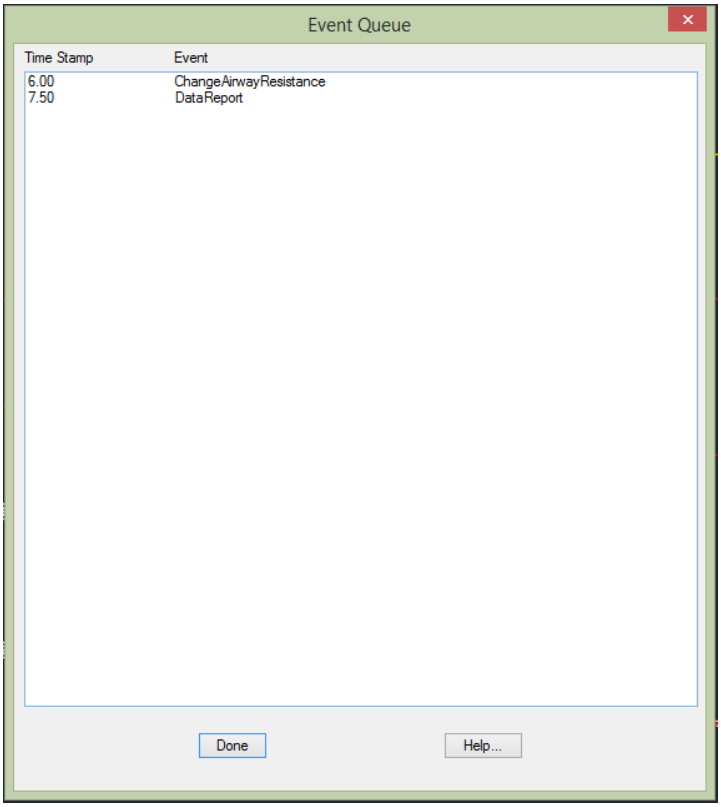
Obviously changing the Time Increment or Time Interval for report does not require an airway to change.

The Event Type's that require the most input are Edit Fan and Add Fire. We will talk more later about the Fire parameters.

Important Note: There are three ways to add a fire. The first is when defining branches to select a branch of type Fire and input the data. The second is to use the Time Table and add a fire event at a certain time before you run the simulation. The third is to add a fire event in real-time while the simulation is running. The second and third are the most flexible because you can decide when the fire started. In fire_example3 it is done in the Time Table under the Branch Menu and starts immediately at time 0.00 minutes.

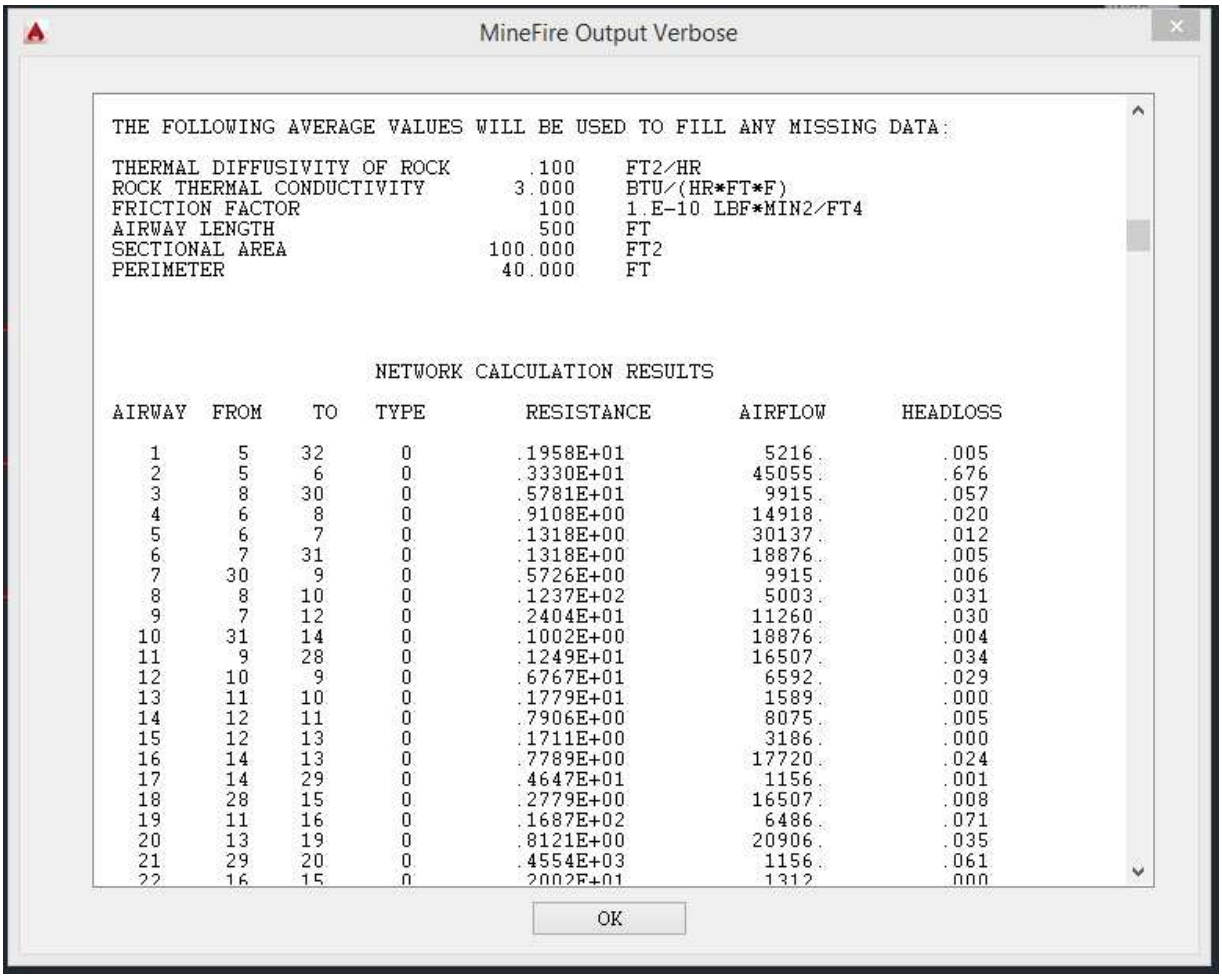
Assuming you Paused the program at 5 minutes and something less than 6 minutes go ahead and select the Change Airway Resistance Event type, then 6.0 minutes for the Time and select airway 25 with a resistance of 0.234 and click Save. If you would have clicked Cancel it will warn you that No Event was added and return to running unless you paused.

2. Event Queue: With the program still Paused at some time before 6 minutes on the Elapsed Time click the Event Queue button and you should get the following dialog box. It only shows upcoming events, at the top you will see where we added the Change Resistance Event at 6:00 minutes.



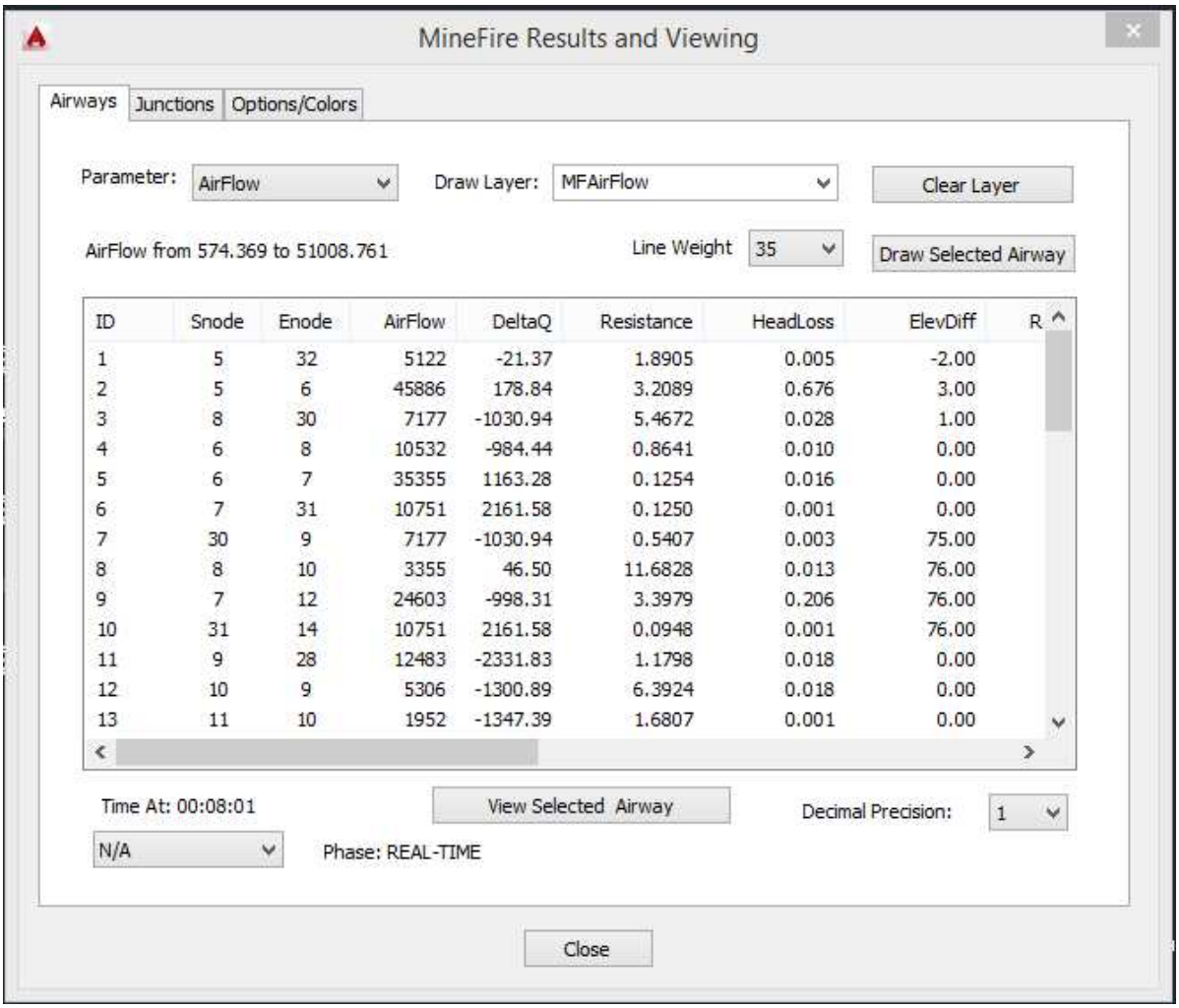
Log: This brings up the same dialog box as Event Queue, but has all the program Log information, and is filled mostly with debugging information and everything the program has been doing. If the program were to have some sort of error, this would be the place to look to see what happened.

Verbose: This button brings up the main Most Detailed MFIRE 3.0 report. This report has a wealth of information. Since we clicked the button during running in real-time it only shows the information up to the point when we clicked the button. However, it should at least contain the Steady State (Basic, Network and Temperature) and the Non-Steady State, and will likely not show the Quasi-Equilibrium state.



Important Note: There is information that is only available in this file. All the features under the Output Menu read from this file and most if not all the information that can be displayed graphically is from the Output Menu. However, there are things that are not available to see from the Output menu graphically such as units being used, default values assumed, Fan information, corrections per mesh, iterations, and most importantly warning messages about fires and airway changes. The only thing left in this file that could be displayed graphically is the DATA RECORD IN AIRWAYS in the Non-Steady State, we do allow to show the DATA FOR THE FUME FRONT IN AIRWAYS. Currently, we do not show the fire information labelled DATA FOR HEAT SOURCES.

View Results: When you click this button you get the power house of the graphical display in real-time or from the Verbose file above if from the Output menu. The following dialog appears. This is the interface to become the most familiar with as it is used extensively under the Output menu after the program finishes and under the Output Menu you have the options of seeing data from each stage/phase (Steady, Non-Steady, Quasi Equilibrium) as well as Time Intervals for the Non-Steady.



Important Note: As mentioned above this same dialog interface is used to display most of the MineFire information, however, the information changes depending on where you invoke this dialog from. The one we just clicked from the Real-Time View Results button has by far the most information. Notice the Airway Parameter drop down and the horizontal scroll bar across the bottom. Same thing for the Junctions tab, that is, in real-time the most information is available. From the Output menu information is limited to what is stored in the Verbose Output File.

→***Airway Tab***: The above dialog box shows the Airway Results Tab, so let's break it down:
The airway tab has more selections than the Junctions Tab because Airways are the only ones that can be colored up using the Options/Colors.

1. **Parameter:** This drop down list shows all the Airway parameters that can be posted and/or colored on the drawing depending on the min and max range of the parameter. This will make more sense when we get to the Options/Colors tab. When you select an item from the Parameter drop down two things happen immediately. The first is that it changes the Draw Layer to more closely match the parameter name and the second is that it shows below the Parameter drop down the Parameter Name and the minimum to maximum value of the selected Parameter.
2. **Draw Layer:** This is the layer it will use to superimpose the airway 3D polyline on. There are only two places where this is used. The first is if you select/highlight an airway in the list and click the Draw Selected Airway (it will just draw Airway with the chosen Line Weight with a color of Magenta). The second place this is used is when you select the Options/Colors tab it will draw the Airways matching the color range and Line Weight you chose here on the selected Airways in the drawing. You can type in your own layer name and it will draw on that layer, however, it will not keep it added to the list after you either re-select the Draw Layer drop down or exit and come back in. Once you do a Draw Selected Airway or a color by airway parameter then the layer chosen here is added to the AutoCAD layers and you can manipulate them from there or Display→Modify Layer.
3. **Clear Layer:** This clears the layer chosen in the Draw Layer drop down. Potentially, you could draw ALL the parameters with your choice of colors and ranges on different layers and use the Display→Modify Layer to turn on and off the ones you want to see. The most important thing to remember here is what layer (from the Draw Layer drop down) you drew the 3D polylines on so that you can clear them later. It essentially deletes all the entities on a selected layer. But if, for example, you draw the AirFlow parameter on layer MFAirFlow then switch to DeltaQ and hit clear layer it will not delete/clear the MFAirFlow layer. It ONLY clears the layer selected under Draw Layer.
4. **Line Weight:** This is simply the 3D polyline thickness or line weight that the program is going to use to superimpose the 3D polyline on the airways if you use Draw Selected airways or the Options/Colors tab.
5. **Draw Selected Airway:** To use this option first select a Line Weight then an airway from the main Airway List (the one with ID, Snode, Enode etc…) then click this button and it will draw/highlight the Airway with the selected line thickness and the color of Magenta.
6. **Main Airway List:** This is the big list in the middle of the screen and like the Parameter drop down the parameters listed here can change depending on which phase (Steady, Non-Steady, Quasi-EQ etc..) you are in. The one we looked at above is from the real-time and has the most information as you can see by scrolling to the right with the horizontal scroll bar across the bottom. Clicking in this list will select an Airway for Draw Selected or View Airway.
7. **Time:** On the bottom left you will see Time At: (00:08:01 in this example) this only applies to Real-Time and Non-Steady State options from the Output menu and is there for your information to let you know what time or time interval you are looking at. The Time drop down below shows N/A in this example, however, if you were looking at it from any of the Non-Steady State from the Output menu it would have list the time intervals from the

report and selecting different time intervals would change all the Airway data in the main lists.

8. **View Selected Airway:** To use this option first select an airway in the main list and then click this button. It will show (not draw) a green arrow pointing to the Airway attribute block then ask you on the AutoCAD command prompt if you want to Zoom Previous (Yes/No) the default is Yes, but if you want to stay Zoomed in to the airway it is pointing to then select N for No or type No. Since this option zooms in to the area around the airway it can be used in conjunction with Draw Selected Airway. For example you could Draw Selected Airway and if it is not visible on the screen you could use View Selected Airway.
9. **Decimal Precision:** This drop down lets you choose how many places to the right of the decimal point you want to use to see the selected Airway Parameter on the drawing. Looking at the main list you can see that Airflow really is not that precise so selecting 0 for the decimal precision would be good enough. Resistance is shown in the list to 4 decimal places so maybe 4 would be a better choice etc..
10. **Phase:** This text is at the bottom and always shows what phase you are looking at in this dialog box interface. In this example it shows REAL-TIME. Others that you might see are STEADY STATE BASIC, STEADY STATE TEMPERATURE, NON-STEADY, QUASI-EQ etc.. It is important to keep an eye on this if you ever forget or select something on the Output menu intending to select something else. This will always tell you what results you are looking at.

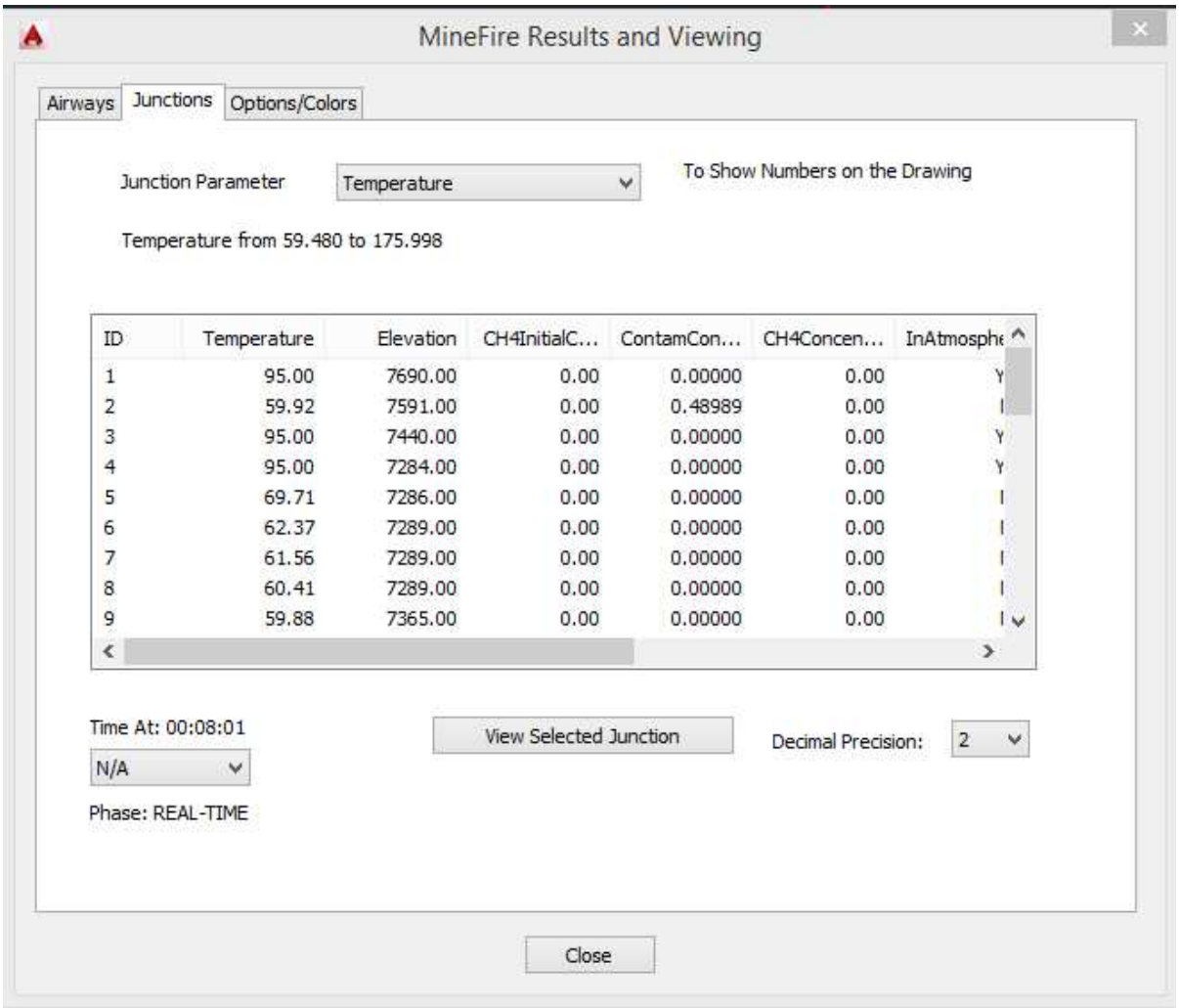
→Junctions Tab

When you select this tab you will get the dialog below, for this example in real-time, otherwise same dialog just different parameters. Note here again that Real-Time shows the most parameters. To see all the parameters in Real-Time you can scroll to the right with the horizontal scroll bar on the bottom of the main list.

1. **Junction Parameter:** Use this drop down to select the Junction Parameter that you want to see posted on the drawing next to the junction number itself. Below this you will see the min and max range of the selected Junction Parameter.
2. **Time:** On the bottom left you will see Time At: (00:08:01 in this example) this only applies to Real-Time and Non-Steady State options from the Output menu and is there for your information to let you know what time or time interval you are looking at. The Time drop down below shows N/A in this example, however, if you were looking at it from any of the Non-Steady State from the Output menu it would have list the time intervals from the report and selecting different time intervals would change all the Junction data in the main lists. *Note that if it were in the Non-Steady State you could select a different Time Interval for the Junction than what is selected for the Airways which in most cases may be undesirable. So be sure this is what you want to do or make sure that the Junction Time Interval matches the Airway Time Interval.*
3. **View Selected Junction:** To use this option first select a Junction in the main list and then click this button. It will show (not draw) a green arrow pointing to the Junction number then ask you on the AutoCAD command prompt if you want to Zoom Previous (Yes/No) the

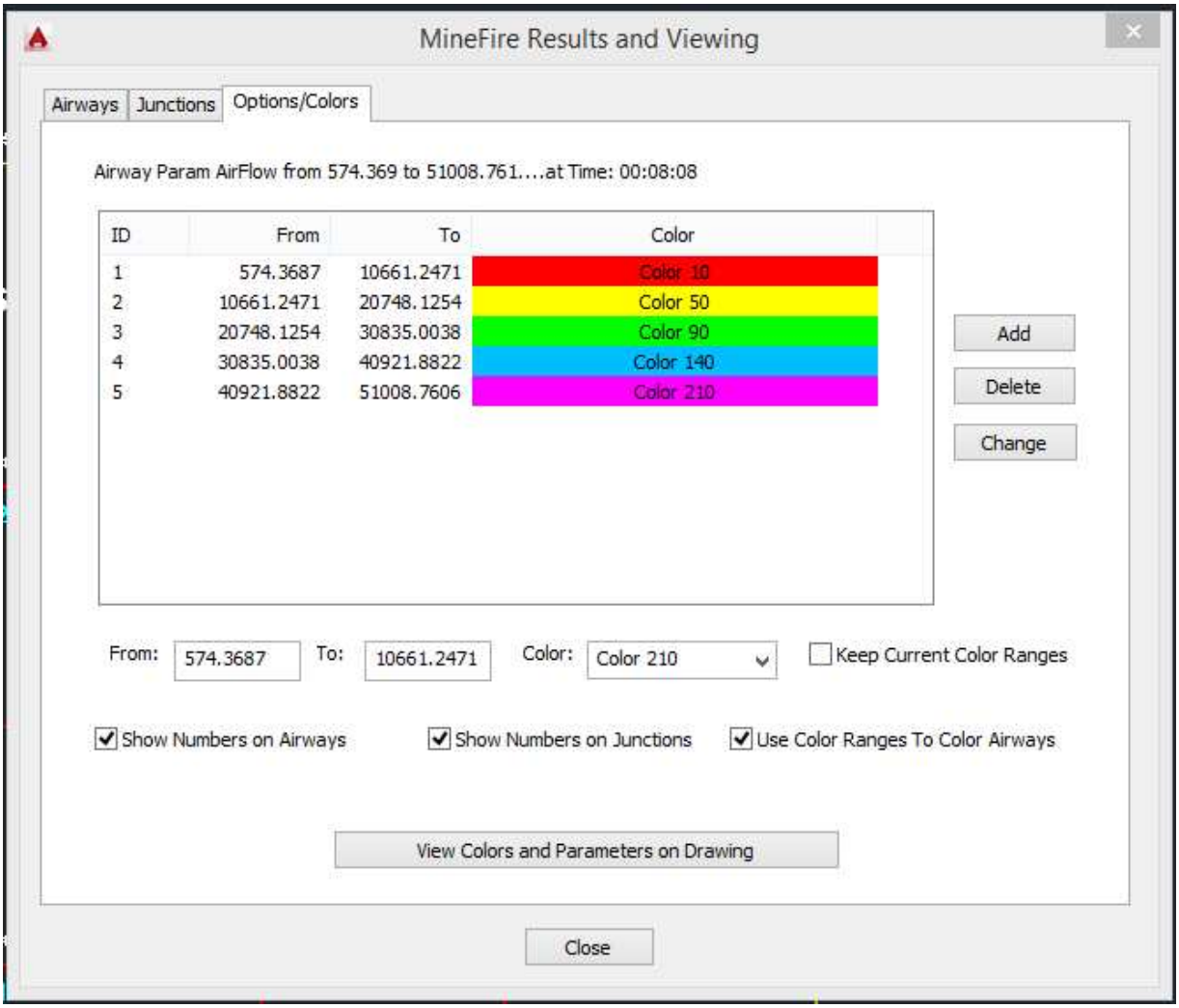
default is Yes, but if you want to stay Zoomed in to the junction it is pointing to then select N for No or type No.

4. **Decimal Precision:** This drop down lets you choose how many places to the right of the decimal point you want to use to see the selected Junction Parameter on the drawing. Looking at the main list you can see that Temperature is shown at 2 decimal places so 2 or 1 for the decimal precision would be good enough. ContamConcentration is shown in the list to 5 decimal places so maybe 5 would be a better choice etc..
5. **Phase:** This text is at the bottom and always shows what phase you are looking at in this dialog box interface. In this example it shows REAL-TIME. Others that you might see are STEADY STATE BASIC, STEADY STATE TEMPERATURE, NON-STEADY, QUASI-EQ etc.. It is important to keep an eye on this if you ever forget or select something on the Output menu intending to select something else. This will always tell you what results you are looking at.



→Options/Colors Tab

This tab is where the action begins as far as posting the selected Airway Parameter and Junction Parameter numbers on the drawing as well as coloring the Airways with 3D polylines that fall within the ranges you select with the Line Weight selected on the Airway tab. At the top you will see the Airway Parameter that it is going to color as well as the minimum and maximum range of values for the parameter. The program automatically calculates a default color range based on 5 ranges divided up equally based on the parameter min and max and adds 0.001 to the last To. If you switch back to the Airway tab and choose a different parameter it will re-calculate the automatic color ranges. However, you may want to change these and you can Add, Delete and Change any value in the list.



1. **Add button:** This button is used to add another color range to the list. Fill in the From and To edit boxes at the bottom and select a color and click Add. It will add the new color range at the end of the list.
2. **Delete button:** This button is used to delete a color range from the list. Just select the color range you want to delete and click this button.

3. **Change button:** This button is used to change the From, To, and color of an item in the main list. First select an item in the main list then make the necessary changes to the From edit box, To edit box and select a new color. Note that it automatically sets the From and To edit boxes to the values that were in the main list, however, it does not set the Color drop down. There is a reason for this the color drop down can only hold up to 15 colors and you can select custom colors so the color in the main list could be missing from the color drop down. Not to worry though whether the color is in the color drop down or not does not change the fact that whatever is in the main list is the color that will be used to color the Airways with the selected airway parameter for that range of values.
4. **From:** The number in this edit box determines the starting value for the airway parameter that falls equal to or after this number. Note that even if you do not change any values in the From and To edit boxes to switch away to another tab you have to put some value in there, just use 0 nothing will be affected, this is just a check to make sure real numbers are there and not letters or something before switching to another tab.
5. **To:** The number in this edit box determines the ending value for the airway parameter that falls less than or before this number. **Important Note:** For the last value in the list the To number is *less than or equal to*, this ensures that if you have an airway parameter that is 51008.7606 in this example it will get colored. You can always edit the last item in the list and make the To value great enough (51009 for example) that it will not miss coloring the last item.
6. **Color:** This is an enhanced AutoCAD color combo drop down and it holds up to 15 colors custom or otherwise. You will notice at the bottom there is a Select Color which lets you choose from the AutoCAD Palette and a Windows color which lets you select from a Windows palette. This gives you great flexibility for choosing a wealth of colors. If you choose the Windows option you can click the Define Custom Colors to have more power.
7. **Keep Current Color Ranges:** Use this check box if you have modified the main color range list by using Add, Delete, and/or Change and you want to switch back to other tabs, but still keep the custom color ranges you have chosen. By default every time you switch back to the Airways tab and select a different parameter or time interval it will automatically re-calculate the ranges based on 5 ranges. This is done to ensure that the color ranges in the main color range list match the parameter. **Important Note:** If this box is checked and you un-check it then it re-calculates for you based on the current parameter and you lose any custom color ranges.
8. **Show Numbers on Airways:** This box is checked by default and if it is checked then it will post the airway parameter numbers in selected on the Airways tab above the airway attribute block. If it is unchecked then it will leave whatever is currently shown above the airway attribute block and not change anything.
9. **Show Numbers on Junctions:** This box is checked by default and if it is checked then it will post the junction parameter number selected on the Junctions tab next to the junctions on the drawing. If it is unchecked then it will leave whatever is currently shown next to the junctions in the drawing and not change anything.

10. **Use Color Ranges To Color Airways:** This box is checked by default and if it is checked then it will use the color ranges in the main color range list to color up the airways in the drawing based on the values that fall within the ranges in the list with the Line Weight (or thickness) specified to draw 3D polylines and on the layer specified in the Airways tab. If this box is unchecked it will not color up the drawing.
11. **View Colors and Parameters on Drawing:** This is where the action begins. Clicking this button post the airway parameters, and junction parameters specified/selected on the Airway and Junction tabs and colors up the drawing depending on what you checked or unchecked for 8-10 above. It pauses after each one and ask you to press Enter/Return to continue. The color of the airway depends on the range the selected airway parameter falls within. The Line Weight (or thickness) and the layer for the 3D polylines depend on what was selected on the Airway tab.

Example Use of View Results:

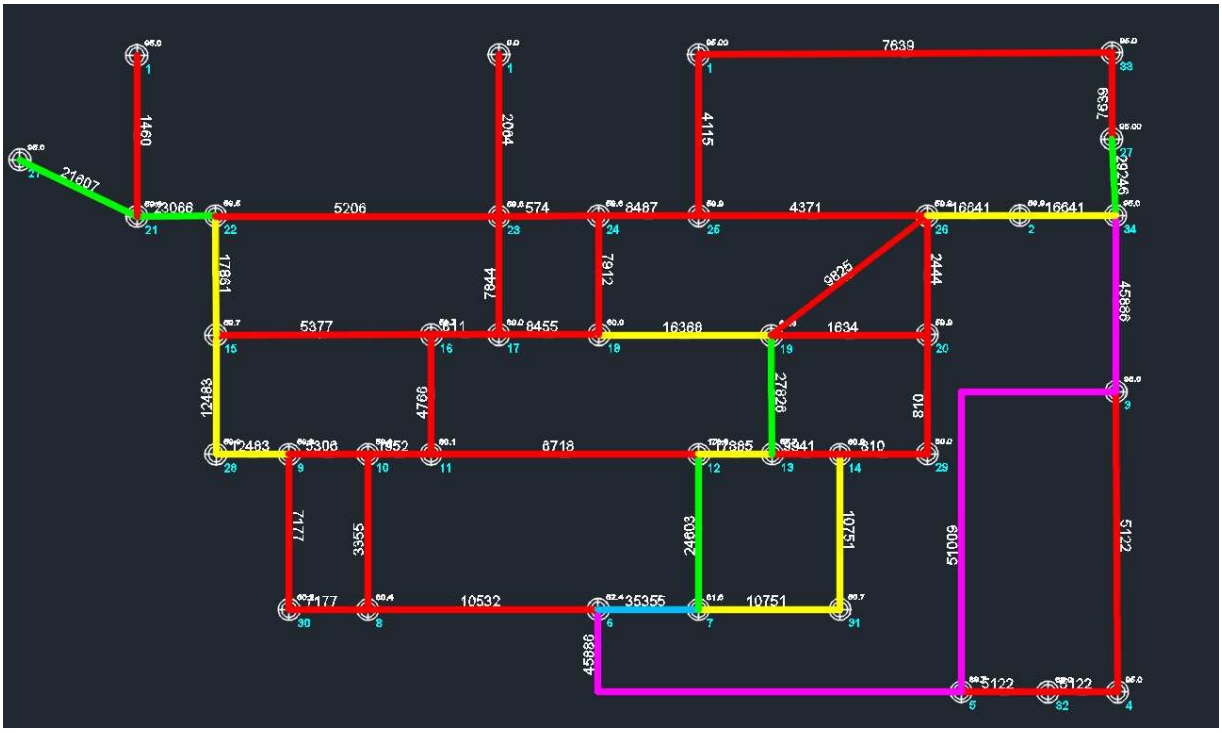
So now let us run through an example and see what this View Results (Called MineFire Results and Viewing on the caption of the dialog box) does and how to interact with the tabs. Assuming you are still looking at fire_example3 at 00:08:08 in real-time let's go back to the Airway Tab select Airflow for the parameter, notice that the Draw Layer says MFAirFlow (leave that alone) then for the Line Weight select 60 from the drop down and then select 0 for the Decimal Precision. You could select other parameters here to see what happens to the Draw Layer and the min and max ranges, but for now let's just use the AirFlow parameter because it is likely one you would use often.

Next switch to the Junctions tab then select here the Junction Parameter drop down. Go ahead and select each parameter and see how that under the junction parameter it gives you the min and max ranges (this happens on the Airway tab as well) and is useful as a first glance whether there might be some problems. Notice that the Temperature parameter ranges from 59.480 to 175.998 degrees F. So we know right off that 176 degrees F is pretty high. So select the Temperature parameter here and set the decimal precision to 1.

Finally select the Options/Colors tab and for now we will just keep all the defaults here and click the View Colors and Parameters on Drawing button at the bottom. First it will post the AirFlow numbers above the branch attribute block with 0 decimal places, then ask you to Press <RETURN> to continue (actually you can press any key) then it will post the Temperature next to the junctions with 1 decimal place, ask you to Press <RETURN> to continue, and finally it will color up the airways and ask to Press <RETURN> to continue. The reason we put these pauses in here is so that you can zoom/pan around and look at numbers etc. on the drawing after each posting and any colors you are particularly interested in.

You should get something that looks like the below on the fire_example3 drawing. After you press Return the final time the MineFire Results and Viewing dialog box re-appears and you can go back to the Airway tab and clear the layer (MFAirFlow in this case) select other parameters and have a another go at it. **Important Note:** If you ever click Clear Layer and it appears to do nothing or not clear the colored up drawing you have selected the wrong layer in Draw Layer. If you have forgotten what layer you drew on last you can use the AutoCAD LIST command and select a 3D polyline to tell

you the layer. Also you could use Display→Modify Layer to turn on/off layers and see what is what combined with the Edit→Delete Layer and enter the layer name. Or easier to just find the layer and re-invoke the dialog an select the Draw Layer then Clear Layer.



You can go ahead and select different options and play with the different tabs to see what happens, just remember to Clear Layers. One thing to be aware of is that Clear Layer ONLY clears the colored polylines it does not clear the numbers posted on the Airways and Junctions. However, anytime you re-post or use the View Colors and Parameters on Drawing Button it will overwrite/reset to the current parameter if you have the boxes checked to do so.

One more example and this time we will change the color ranges in Options/Colors. Click the Airway Tab then Clear the MFAirFlow layer then choose the AirFlow Parameter again (Note: The Draw Layer automatically changes to MFAirway), set the Line Weight to 100, set the Decimal Precision to 2. Then click the Junctions tab and choose ContamConcentration (this is Fumes) set the Decimal Precision to 5. Now select the Options/Colors tab and select the first line in the main list then in the From edit box below type in 331 and in the To edit box type in 20713, and for the color choose Red then click the Change Button. Now select the second item in the list and click Delete. Finally, click the View Colors and Parameters on Drawing and take a look around. You should see something such as in the following on the drawing. You can see that it is mostly Red now because we expanded the color range for red. In practice, you may want to just take a look at the min and max and put in your own colors for the range that you are interested in. For example maybe you just want to see airways that fall below 200 cfm as red. Or you can just use one color range in the list that covers what you want to see, anything not within that range will not be colored on the drawing.

At this point we realize that we may need a way to put a legend somewhere, well this is a work in progress. However, for now the crude way to do this (although kind of nice) is to use the Microsoft Windows Snipping tool and capture the Options/Colors tab portion that shows the color ranges. You can then copy it and use the PASTECLIP command in AutoCAD or you can save it as a jpg file and attach the jpg file to the AutoCAD drawing if you like.

Here is how this is done. Find the Snipping Tool, its location varies depending on your version of Microsoft Windows, then open it and when you are at the Options/Colors Tab you want to capture bring up the Snipping Tool and click New then window around the Colors so that you have something like this:

Airway Param AirFlow from 591.000 to 50851.000....at Time: 150. Sec.

ID	From	To	Color
1	591.0000	10643.0000	Color 10
2	10643.0000	20695.0000	Color 50
3	20695.0000	30747.0000	Color 90
4	30747.0000	40799.0000	Color 140
5	40799.0000	50851.0000	Color 210

Then while in the Snipping Tool that shows the above go to Edit→Copy, then switch back to AutoCAD and type PASTECLIP and select an insertion point. From there you can resize it or select it and use the AutoCAD Move command. However, for the best resolution I would suggest keeping the size about the same. Below is what the fire_example3 drawing looks like after performing this action. The legend gained from the snipping tool is now part of the drawing if you save it.

That concludes the discussion of the options under the Node Menu and the bulk of the program has been covered. Namely, Run MineFire input, then actually running the program in real-time, and mostly the MineFire Results and Viewing which will be used extensively under the Output menu with a variety of data in the same or a very similar dialog box.

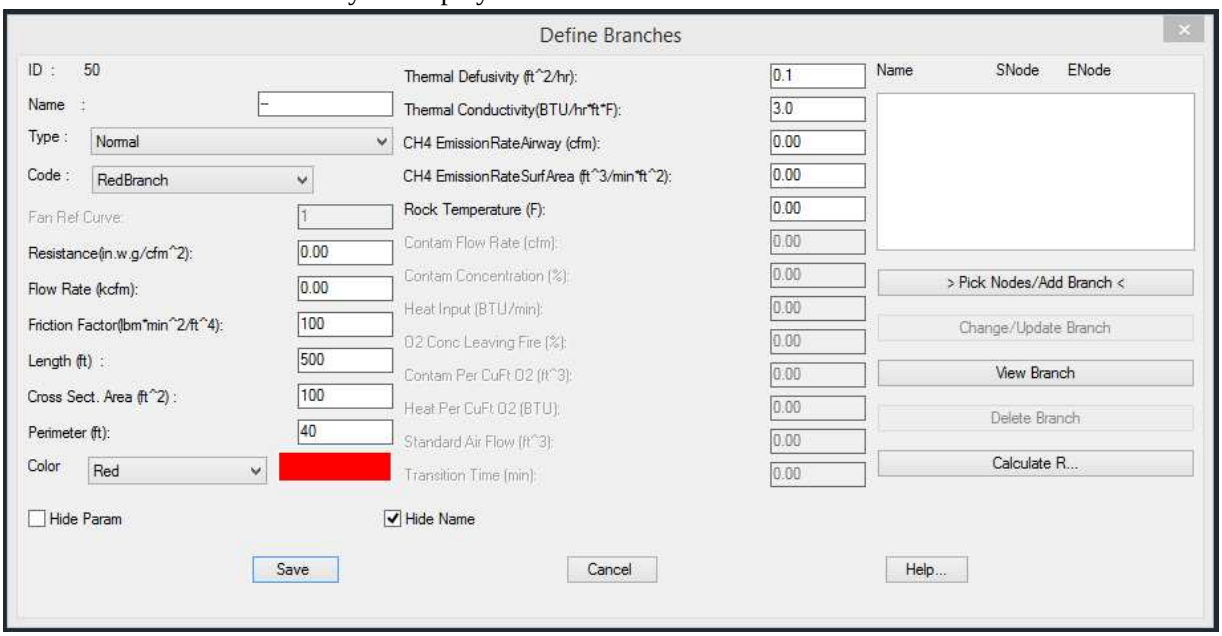
Branch Menu:

This is the main menu for defining, editing, listing and manipulating branch or airway information. It also has the Time Table menu option that allows you to set up events such as Change Airway Resistance, Edit Fan, Add Fire, Remove Fire, Change Time Increment, and Change Output Time Interval at specified times before the program starts running. Which is the same interface you get with Add Event in real-time. Defining and Editing branches will likely be your most common task, however, it is not necessary in testing fire_example3 because all the branches have been defined. You may choose to Modify a branch and/or write down the information for the Modified branch and then use Define Branch to see how it works.

→ Define New Branches

This is where you create or connect the branches/airways to the nodes/junctions. So you would need to have the nodes in place before you select this menu option. Remember that you do not have to have a node just because the branch bends (you can left click out in space) but it will ask for the elevation (Z value) at the bend. Once you reach the end node you can press Enter to complete the branch. **Important Note:** Just because you might click on an intermediate node does not connect a branch to that node. The start node is the first node you click and the end node is the last node you pick when you press Enter any actual nodes you click in between it just goes right through and the actual branch is not connected to that node. Any values below that have a default value putting in zero will make it use the default value.

When you select this menu option the following dialog appears. It is fairly large and takes up most of the screen so if you get any error opening it that indicates the size of the dialog is too large you may need to select a smaller resolution for your display.



ID: This is the branch identification number and is automatically generated by the program using the next available number. The branch ID number starts at 1 when you insert the first branch. The branch ID number must be unique so that is why the program automatically assigns it. There is a way to change it using the DDATTE AutoCAD command and selecting the branch attribute block after you have defined a new branch. However, you must use caution if you do this and make sure that you do not have any other branches in the drawing with the same ID number. For this reason it is not recommended.

Name: Any name you want to assign to the branch to more easily identify it. For example, FAN, Slope, etc.. It is recommended to keep it under 8 characters.

Type: This is the type of the branch you want to define. In most cases it will be Normal. The different types are Normal, Fan, FixedQ and Fire. Depending on what you select the dialog box will enable or disable the relevant edit boxes to input information. Selecting the Fan type enables the Fan Ref Curve where you put in the Fan reference number you want from Define/Edit Fans to tie that curve to this branch. Selecting the Fire type enables all the Fire parameter edit boxes on the right. Selecting the

FixedQ enables the same as Normal, however, the Flow Rate (kcfm) you put in is fixed for the program.

Code: This is the code you want to use for the branch and is only for branch cosmetics (colors, layers, code name) and resistance calculation defaults. All this is setup under Branch Menu→Configure Branches. In fire_example3 we decided to keep it simple and just setup RedBranch and YellowBranch on layers MFC01 and MCFC02 which can be changed in Configure Branches.

Fan Ref Curve: This is the Fan Curve Reference number that you want to use. It comes from Define/Edit Fans and Ref. #. Under the No. of Pairs.

Resistance: This is the resistance of a Normal airway in 1.0*E-10 inches water gauge/cfm^2. Could also be expressed as in.min^2/ft^6 using the Friction Factor 1.0*E-10 lbm*min^2/ft^4. When the input value of R is equal to or less than 0, R will be calculated from Friction Factor, Length, Cross Sectional Area and Perimeter on this main dialog box NOT what is in Calculate R. If these input data items are not complete, an error message will be issued. When the input value of R is larger than zero, it will override the value calculated from Friction Factor, Length, Cross Sectional Area and Perimeter. *For a Fan branch Resistance is the fan pressure in in. w.g. A fan without input for its characteristic curve will be regarded as a fixed-pressure fan and the value of Resistance will be used to define the fan pressure.*

> **Important Note:** You can use the Calculate R button to calculate the resistance for a branch instead of setting Resistance to zero and letting MFIRE calculate it, however, there is one difference to be aware of and that is that the MFIRE 3.0 code multiplies the Calculated Resistance by (Reference Density / 0.075) So if you want to be consistent here any formula in Calculate R that uses length and K factor etc.. you should multiply by the (Reference Density/0.075). The Reference Density is defined in the Run MineFire dialog box. We have left it this way for now just to give the user a choice as to whether or not to multiply by this value.
>
> So if R is zero MFIRE 3.0 uses:
> R = FrictionFactor*Length*Perimeter / (5.2*CrossSectionalArea^3) * ReferenceDensity/0.075
>
> And MineFire Calculate R button on either Define Branches or Modify Branches uses:
> R = FrictionFactor * Length * Perimeter / (5.2 * CrossSectionalArea^3)
>
> So that you have the flexibility to do what you deem correct here.

Flow Rate: is the volume airflow rate in the branch in KCFM so that if it is 1000 cfm put in 1. The program should also calculate this. Unless a FixedQ branch this may change throughout the calculation.

Friction Factor: This is the friction factor in 1.0*E-10 lbm*min2/ft4. The default value is 100. (Typical Friction Factor values are shown in Table 5 in Appendix 1 of the U.S. Bureau of Mines MFIRE Users Manual Version 2.20.)

Length: This is the length of the airway in ft. The default value is 500. Anytime you define a new branch and pick the start node and end node it will automatically update this Length edit box to the actual 3D length of the branch/airway as defined by the mine map in AutoCAD. You have the option to change this to something other than the actual real-world mine map length, but you must do so

after you pick the start node and end node, as it automatically sets this value for you when you do. It is recommended to let the program calculate the length and use that instead of entering your own lengths. There are a couple of reasons for this. One is that when you are using features to draw the Fume Front the length of the branch is crucial to see the proper results. Two is that the length is used in the calculation of resistances as well as used in the program for recirculation paths etc. Yes you can override the actual real-world length of the mine map but I cannot see a reason for doing so other than for testing purposes. The Define Branches dialog is the only place the program calculates the length automatically, Modify Branch does not. If you have put in one of your own lengths for testing purposes there is an option under the Branch Menu called Get Polyline Length and you can put that number back in with Modify Branch.

Cross. Sect. Area: This is the cross-sectional area of airway in ft2. The default value is 100.0.

Perimeter: This is the perimeter of airway in ft. The default value is 40.0

Color: This is the color of the branch 3D Polyline and arrow/cone. It is recommended to leave it alone and let it color by layer based on the code as set in Configure Branches. The reason for this is that you can change the color of all the branches with a certain code globally just by going to Display→Modify Layer and changing the color for that layer. Note: You should also change it in Configure Branches before or after you change it in Modify Layer.

Thermal Diffusivity: This is the thermal diffusivity of rock in ft2/hr. The default value is 0.1.

Thermal Conductivity: This is the thermal conductivity of rock in BTU/hr*ft*F The default value is 3.0. (Typical values for thermal diffusivity and thermal conductivity for various rock types are shown in Table 6 in Appendix 3 of the U.S. Bureau of Mines MFIRE Users Manual Version 2.20.)

CH4 EmissionRateAirway: This is the methane emission rate for the whole airway in cfm under the reference conditions (1 atmosphere and Reference Temperature under Run MineFire Control Card 1). If CH4 EmissionRateAirway is zero or less, the methane emission rate in the airway will be calculated from CH4 EmissionRateSurfArea.

CH4 EmissionRateSurfArea: This is the methane emission rate per unit surface area of the airway in ft3/min*ft2 under the reference conditions. If CH4 EmissionRateSurfArea is input as zero or less, the methane emission rate will be calculated from the methane concentration in junctions. If both CH4 EmissionRateAirway and CH4 EmissionRateSurfArea are 0, methane emissions are not considered.

Rock Temperature: This is the rock temperature in degrees F. of the airway (quasi-equilibrium wall surface temperature before the fire started). It is normally calculated in the program run and does not require input from the user. If the user has better data available, it may be input here to override the calculated value. The acceptable range of user specified rock temperature is from 0 F to 150 F.

Important Note: The rest of the edit boxes here are only available for fire branches, but it is a judgment call as to whether to have a fire branch or just use the Time Table and add a fire event or use Add Event in real-time to add a fire event. In any case, it is advisable to do one or the other and not both. Either define a fire branch or use Add Event under Time Table or Add Event real-time.

Contam Flow Rate: This is the contaminated gas inflow from the contamination source in cfm.

Contam Concentration: This is the concentration of fumes in the contaminated gas, Contam Flow Rate in %.

Heat Input: This is the heat influx at the fire source in Btu/min.

O2 Conc Leaving Fire: This is the oxygen concentration leaving the fire source in %.

Contam Per CuFt O2: This is the fume production in ft3 for each ft3 of oxygen reaching the fire source.

Heat Per CuFt O2: This is the heat generation in BTU for each ft3 of oxygen reaching the fire source.

Important Note: Contam Flow Rate, Contam Concentration and Heat Input define a fixed heat input fire source. O2 Conc Leaving Fire defines an oxygen rich fire. Contam Per CuFt O2 and Heat Per CuFt O2 define a fuel rich fire. When they all have non zero input values, the oxygen rich model overrides fixed heat input fire model, while the fuel rich fire model overrides the oxygen rich model. The oxygen rich and fuel rich fires are primarily research tools. While the oxygen rich and fuel rich fire models were retained in MFIRE 2.0, the fixed heat output model will be suitable for most mining uses.

Standard Air Flow: This is the reference airflow in ft3 at which the above parameters are defined. When the airflow is less than or greater than Standard Air Flow, the heat generation from the fire source will decrease or increase accordingly but in a damped fashion. If Standard Air Flow is less than 10 cfm, the damping is not performed.

Transition Time: This is the leading time period in minutes for an oxygen rich fire to reach full strength. Heat generation is calculated from available oxygen. Increasing the airflow to a fire will increase heat output, however the heat output will lag the increased airflow. The quantity of lag will vary with the fuel type and is user defined with Transition Time. The effect is linear over the Transition Time.

Hide Param: If this box is checked the program will hide (not show) the parameter number posted above the branch during the View Results or MineFire Results and Viewing during real-time or by using most Output Features.

Hide Name: If this box is checked it will hide (not show) the name of the branch. In most cases this is okay. It depends also on how large you set the Size of Branch Names under Output→Edit Schematic. If it is only 1 or so you may not want to hide the name as it will be small enough already.

Main List of Branches (Name SNode ENode): This list all the branches defined in this session or run of Define New Branches. If you click okay these branches will be saved with the drawing. If you click Cancel it will ask you to confirm that you want to Cancel and delete these branches. So use caution here even if you have defined 50 or 100 branches in this session and click Cancel then Yes it will delete all the branches in this list.

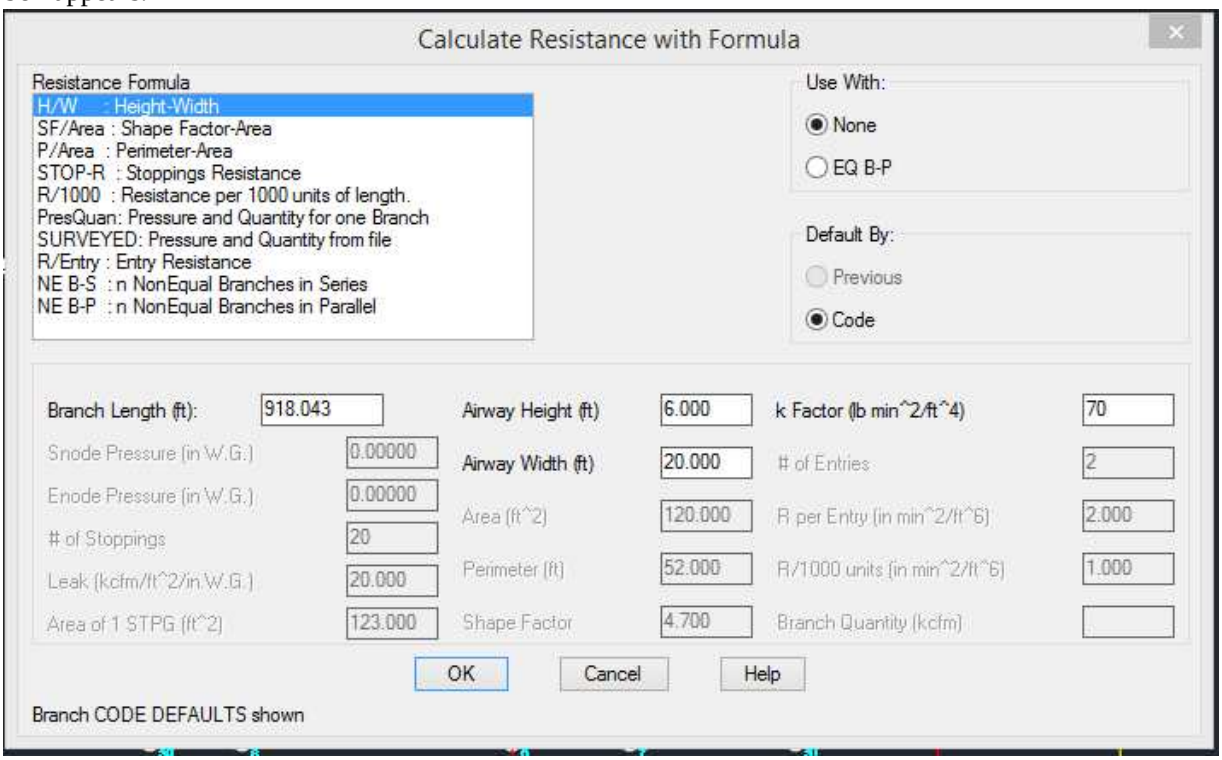
Pick Nodes/Add Branch: After filling in all the relevant information mentioned above, you finally click this button to select the start node, any intermediate nodes (by left clicking out in space and entering the elevation at the bend) and then left clicking the end node. Right after you left click on the desired end node press Enter to complete the branch.

Change/Update Branch: If you want to change one of the defined branches in the list first select the branch from the Main List of Branches then change the desired information in the drop downs, edit boxes, or check boxes then click this button to change that branch.

View Branch: If you want to view a branch in the Main List of Branches select the branch in the list and then click this button. It will show a green arrow pointing to the branch and ask if you want to Zoom Previous (Yes/No) with a default of Yes if you hit Enter. If you want to stay zoomed into the branch enter N or No and press Enter.

Delete Branch: If you want to delete a branch in the Main List of Branches select the branch in the list and click this button and it will remove the branch from the drawing and the list.

Calculate R: You may want to click this button right after Pick Nodes/Add Branch to use some Ohio Automation ICAMPS custom resistance formulas. Or you can select any branch in the Main List of Branches and click this button to calculate a resistance. If you click this button the following dialog box appears.



Important Note: Using the Calculate Resistance above and using the P/Area (or Premieter Area formula) has one difference to be aware of and that is that the MFIRE 3.0 code multiplies its calculated resistance by (Reference Density / 0.075) When R is zero. See Note above under Resistance for the difference between formulas.

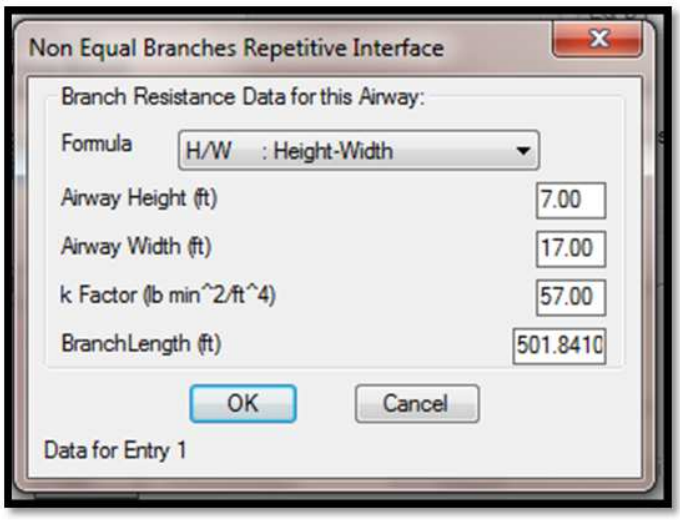
Select a formula that is appropriate for the branch being defined. After you select a formula, only the applicable parameters in the box will be highlighted. For a new branch the parameters contain the code defaults as indicated by the Code button under Defaults By option and the message "Branch CODE DEFAULTS shown" appears at the bottom of the dialog box. The parameter values correspond to the code default values you entered in the Configure Branches option. If you are computing the branch length from survey data, the branch length is not used in the calculation.

If you edit a previously defined branch, the Default By**:** option changes to Previous. You can change any of the highlighted parameter values. The branch length is automatically calculated from the branch polyline, but you have the option to change the length if necessary. The message at the bottom of the dialog box changes to "Previous resistance defaults shown: Branch Length = Previous Length".

The PresQuan and SURVEYED formulas apply if you know the pressure and quantity in a branch. ***Important Note:*** *Branch Quantities are in kcfm*. After you have done a ventilation survey, you can enter the survey data one branch at a time using PresQuan or you can use the SURVEYED formula with existing files or create files of the surveyed pressures and quantities. The pressure data is stored in a file you designate as <DWGNAME.P> which has the format (node number, pressure) and the quantities are stored in a file <DWGNAME.Q> which has the format (start node number, end node number, quantity in kcfm).The program searches the .Q file for the node combination which matches the branch you are entering. If the branch is in the .Q file, the program then looks for the surveyed node pressures in the .P file and computes the branch resistance.

The formula NE B-S: n None Equal Branches in Series has been disable because such branches are difficult to modify. If you have a situation where non-equal entries are in series, create a separate branch for each segment in the series.

When you select the NE B-P : n Non Equal Branches in Parallel formula, you must indicate the number of branches (entries) and then click the OK button. The following dialog box will appear, once for each of the entries.



After you select the resistance formula for each branch, the required parameters and their default values will appear. The branch length is the length of the polyline. If the non-equal branches are not the same length, you can change the branch length for every entry.

For single entry airways, click the None button under Use With and select one of the other formula options. For multiple **equal** entries in parallel you need to select one formula and set of parameter values.

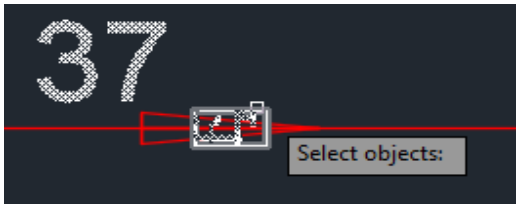
Click the EQ B-P radio button under Use With, select any of the single entry formulas, enter the parameter values and type the # of Entries.

Important Note: This is the same Calculate Resistance dialog box used in MineVent and as such it still has the STOP-R formula for leakage branches in MineVent. This formula is not available as of yet in MineFire. The main reason is that instead of turbulent flow leakage branches in MineVent used laminar flow and instead of the formula P = RQ^2 for leakage branches it was P=RQ^CValue. Where the CValue was some time tested value for laminar flow, such as 1.3 or whatever was suitable for the mine.

OK: Click the OK button and the branch resistance will be calculated and appear in the Resistance edit box in the Define Branch dialog box.

→Modify Branch Parameters

This menu option allows you to change all the information entered in Define New Branches above in existing branches. The only additional feature here is Reverse Branch. The first thing you will do is click the Pick Branch Attribute Block at the top then you will be presented with the pickbox as shown below and you can click the number above the branch (the number will depend on what you posted last) or you can click the attribute block which is the rectangle that is at the center of the last two polyline vertices near the arrow/cone. Once you select it press Enter and the Edit Branch Dialog will re-appear with the selected branch information. You can change the size of the pickbox by typing PICKBOX at the AutoCAD command prompt. A smaller pickbox usually works better with zooming in. The below pickbox is set to 3.



→Erase/Delete Branch

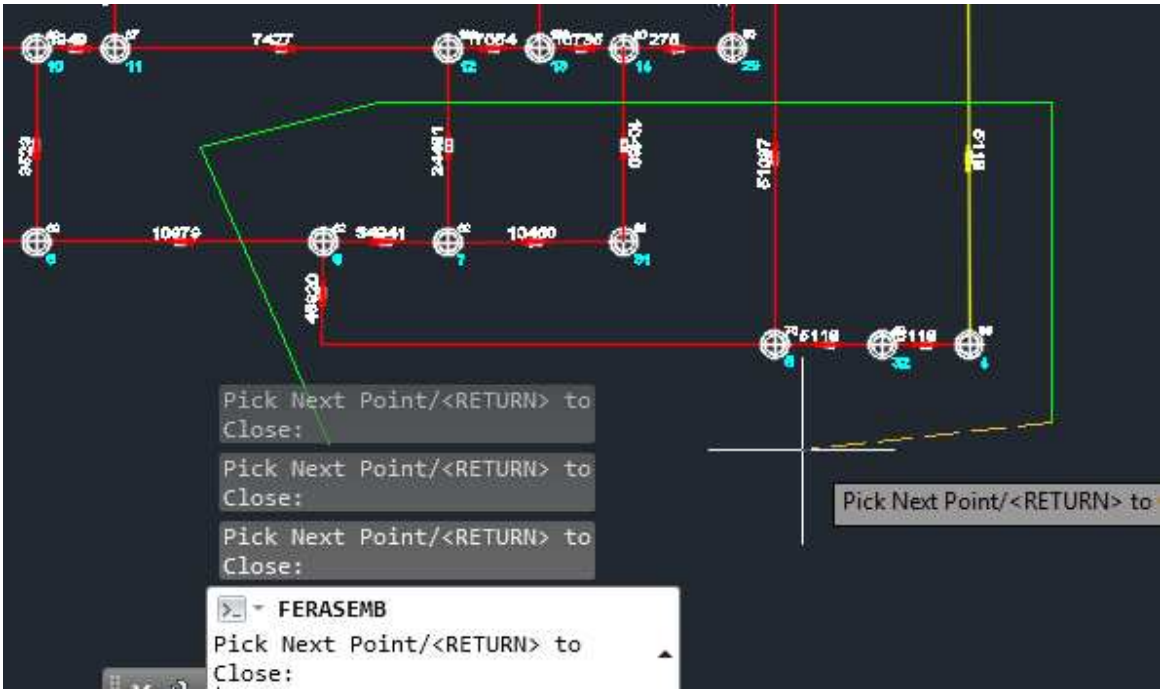
When you select this menu option you will get the same pickbox as described above in Modify Branch. Just select the branch attribute block and press enter and the branch will be removed from the drawing.

→Erase Branch Snode Enode

This option is useful if for some reason you cannot see the branch attribute block. When you select this option from the menu it ask you on the AutoCAD command prompt to enter the start node and end node and then it removes that branch from the drawing.

→Erase Multiple Branches

This is a very powerful option to erase or delete a group of branches in the drawing or even all of them if you like. When you select Erase Multiple Branches the AutoCAD command prompt will ask you to "Create closed polygon: Pick Start Point:" from here you can draw a closed polygon around the group of branches you want to delete. Note that you only need to include the branch attribute blocks in the polygon as shown below:



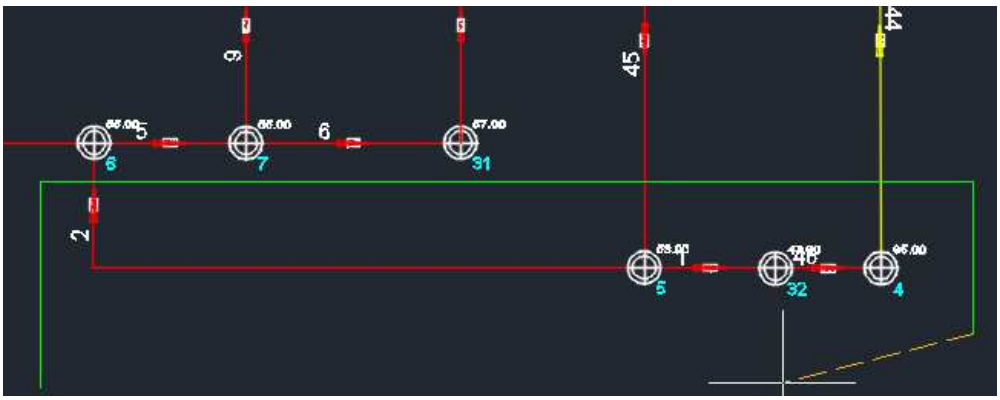
Notice that you do not want to try to manually close the polygon, just press Enter/Return and it will close the polygon for you and delete the branches as shown above once you press Enter all the branches with their branch attribute blocks within the polygon will be deleted from the drawing.

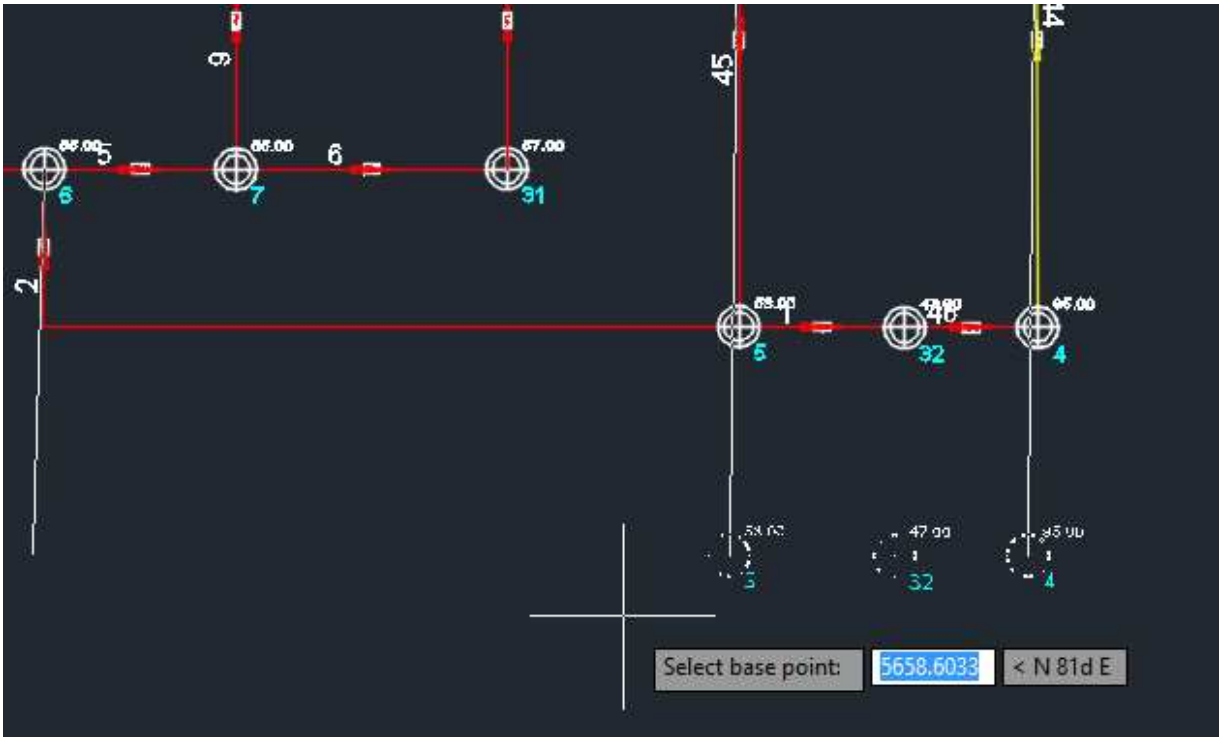
→Get Polyline Length

When you select this menu option you will get the same pickbox as described above in Modify Branch. Just select the polyline (NOT the branch attribute block) and press enter and a dialog will appear with the 3D polyline length.

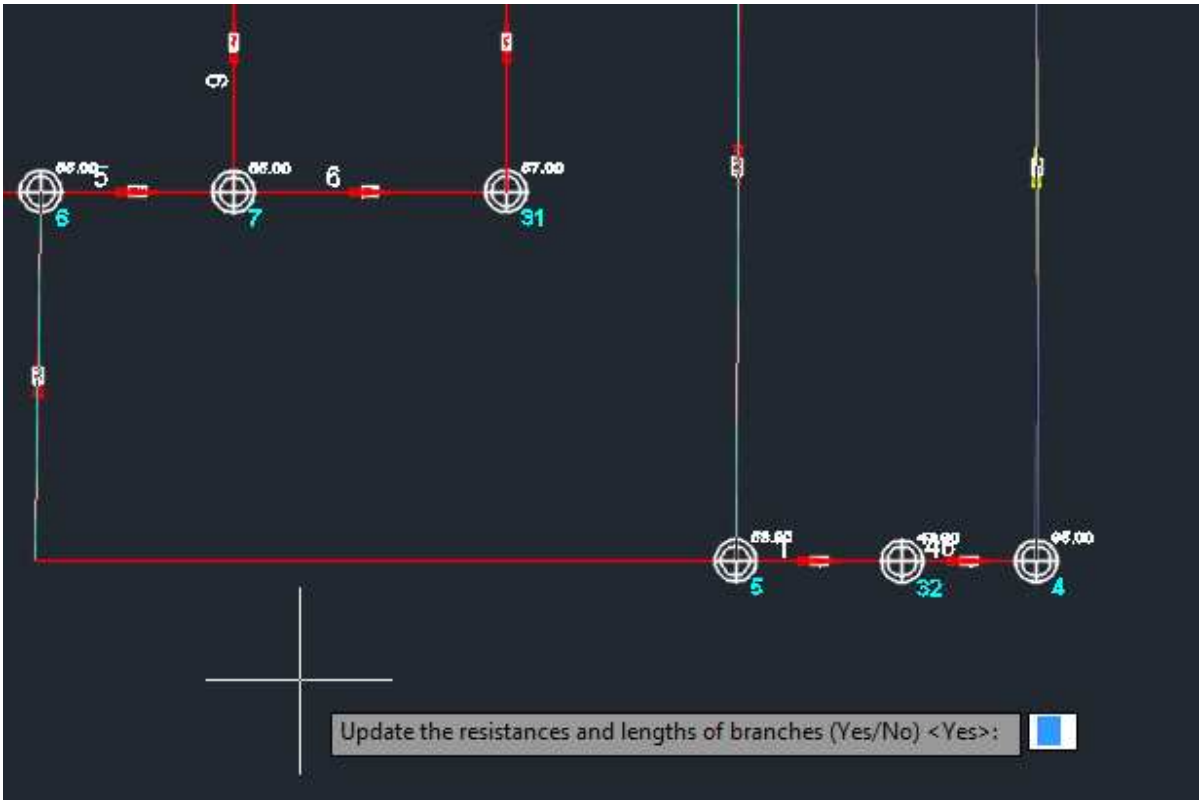
→Move Multiple Branches

This option allows you to modify a network where the change involves moving a group of branches such as lengthening or shortening mains or a longwall panel. When you select this option, the command line will ask you to create a closed polygon (same as Erase Multiple Branches) around the branches to be moved and then to drag the selected branches (making the main or panel longer or shorter).





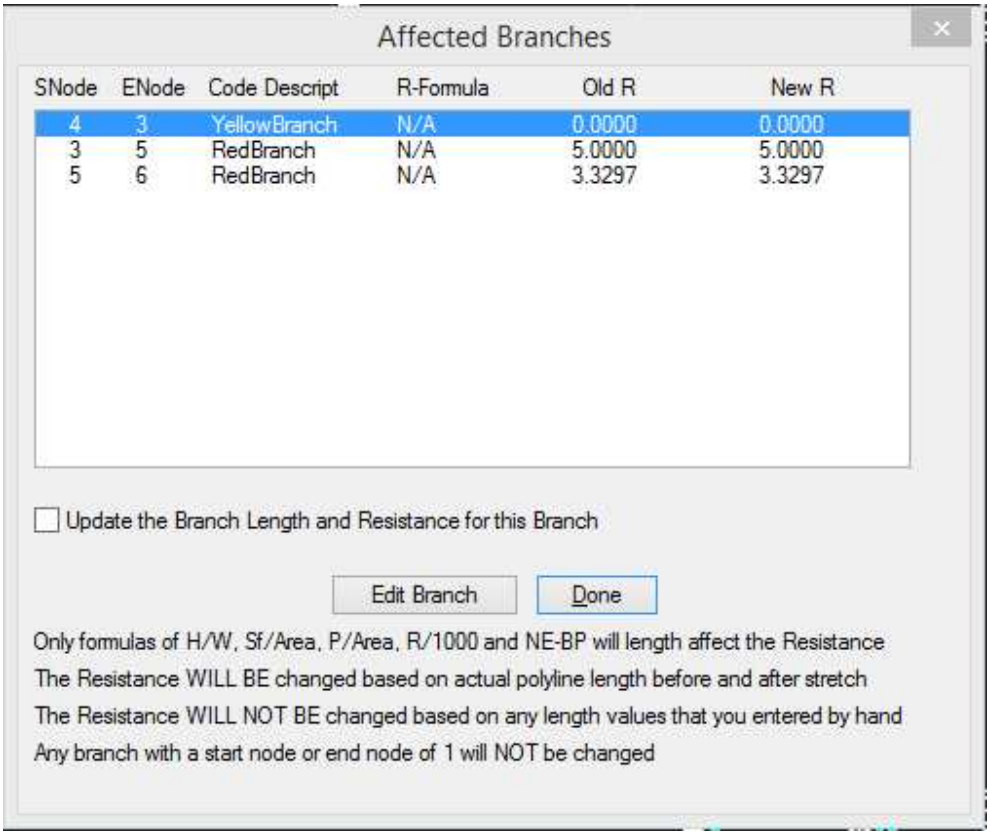
It will ask to select the Base point so left click somewhere desirable (maybe a node) then drag to the desired location and left click again. Also you could have enter @x,y,z for the second prompt.



The coordinates of the moved nodes will be updated and the following command line prompt will appear.

Update the resistance for the new branches (Yes/No) <Yes>

If you hit <Enter> or type Yes to the above prompt then the following dialog box will appear allowing you some interaction for the proposed branch length/resistance changes.



The main areas to pay attention to in the above dialog box is the Resistance Formula, the Old-R (branch resistance before the stretch) and the New-R (branch resistance that will be assigned when you hit Done). By default the branches that have a resistance formula that uses the length will have a check in the "Update the Branch Length and Resistance for this Branch" checkbox and the resistance formulas that do not use a length will not be checked. So that for each branch in the list you can check or uncheck the checkbox and watch the New-R change accordingly. If you have a branch that does not use the length in the calculations, such as Pressure and Quantity Resistance formula, then you can edit them individually from the list by selecting the branch in the list and hitting the Edit button, which will bring up the standard Modify Branch Parameters dialog box.

CAUTION: *The objects inside the window move without changing their shape or position relative to other objects within the window but objects outside the window on branches cut by the window are repositioned in proportion to the distance the window is moved. Use the following precautions:*

1. If you are shortening branches, be careful not to window the arrow and block on any branch that will be shortened or else they may overlay other branches of the network.

2. Node entities include the circle, sphere, node number and parameter. Likewise branch includes the attribute block, arrow/cone and parameter. If you are moving nodes or branches, be sure that the window encloses both the entity and all of its associated objects.

→Copy Branches

This option allows you to edit a ventilation schematic by copying a section of an existing ventilation schematic diagram. You select the nodes and branches to copy by drawing a multi-sided closed polygon window around the nodes and branches to be copied. All the nodes inside the window and the branches which connect them are copied. You must specify the node number as it is copied. The default is the next number that is higher than the largest number on the drawing, but you can specify any unused number. You can also enter an existing node number but it will warn you that it is a duplicate and you will not be given the option to change it. The system also asks for the new name of the nodes and branches as they are copied. All the other parameters are assumed to be the same as the original values including resistance formulas and resistance data.

When you select this option, the following command line prompts appear.

Create Closed Polygon: Pick Start Point

After you select a point, the following command line prompt reappears after each selection. The polygon can have any size or shape.

Pick Next Point / <Return> to Close

After you close the polygon, the prompt becomes

Specify base point

After you select the base point, the prompt asks for the point where the base point will be moved.

Specify second point

You can pick a point with the cursor or use the command line to specify a distance and angle or the coordinates of the point. After you specify the point, the nodes are moved, one at a time. The command line will show which node is being moved. You will be asked for the new node number. The prompts are:

Copying Node # X

Node Number for New Node <Y>:

where **X** is the existing node number and **Y** is the default node number. The default is the next number higher than the largest node number on the drawing. You can specify any unused number. If you enter a lower number, the next default will be the first available node number larger than the number you specified.

If you entered a duplicate node number you will get a warning to insure this was your intent. You are not given the option to change it at this time, however once this Copy Branches ends you can use the

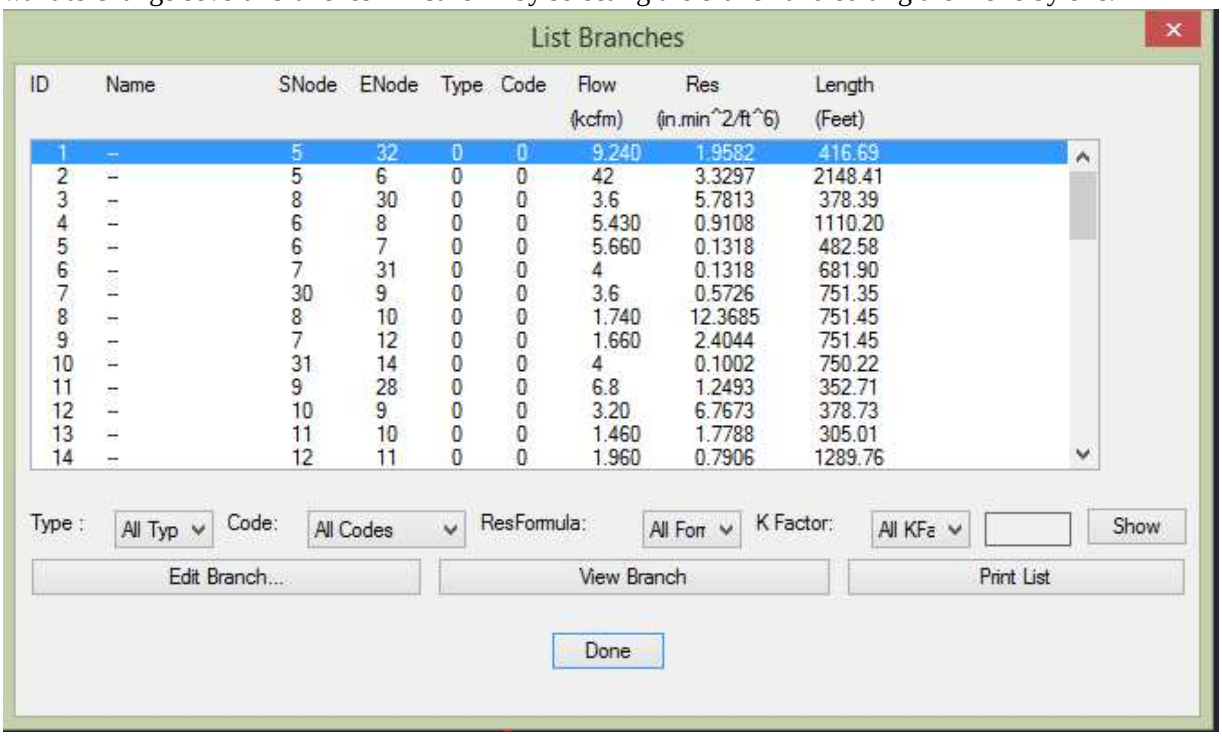
AutoCAD command DDATTE to select the node and then change the number. However, the AutoCAD DDATTE command does not check for any duplicate nodes and is not a recommended way to change *any* node values. It is recommended to use Node Menu→Modify Node Parameters for all else but DDATTE is the only way to change a node number or a branch ID number, again not recommended.

Enter New Node Namee <-->:
Uses the default name of just – but you can change it to suit your needs. Duplicate Node Names are not allowed.

The copying prompts repeat until all the nodes within the window have been moved. Then the branches connected to the nodes are copied. You are asked to specify names for the copied branches but all other information is automatically transferred. Branch names have to be unique, because unique names are helpful when evaluating ventilation networks.

→Check For Duplicates
This option just checks for duplicate branches or more specifically branches that have the same start node and end node and list them in a dialog box.

→List Branches
This option brings up a list of all the branches in the drawing. It is good for trouble shooting or if you want to change several branches in list form by selecting the branch and editing them one by one.



Showing Specific Branches: To view only branches of a certain type, code, resistance formula, and/or k factor you can select your search criteria from the pop down menus and then hit the *Show* button to see them in the list. The different search filters are listed and discussed below.

Type: You can list all the branches or only those for a specific branch type. The list defaults to all types. To select a specific type, click the drop down list to the right of the word Type to display the available branch types as shown below and then pick the desired type and then hit the *Show* button.

Code: You can list only those branches for a specific branch code. The list defaults to all codes. To see the branches that contain a specific code, click the drop down list next to the word Code as shown below and then pick the desired code and then hit the *Show* button. This drop down list the descriptions for the code defined in Configure Branches.

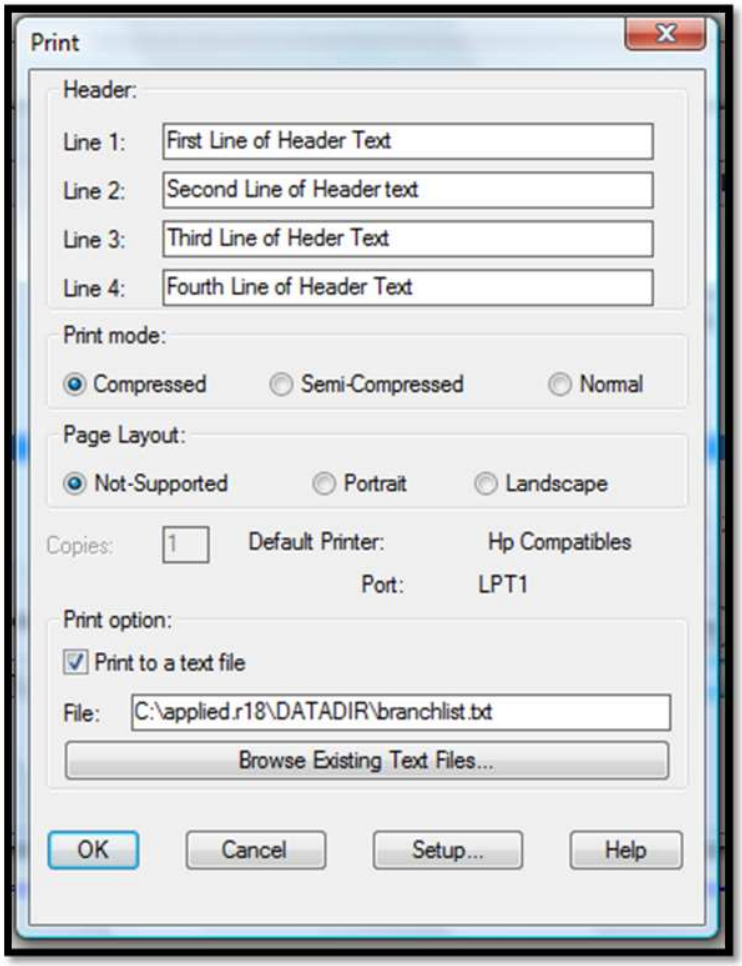
ResFormula: You can list only those branches for a specific resistance formula. The list defaults to all resistance formulas. To see the branches that contain a specific resistance formula, click the drop down list next to the word ResFormula as shown below and then pick the desired resistance formula and then hit the *Show* button.

K Factor: You can list only those branches for a specific K factor. The list defaults to all K factors. To see the branches that contain a specific K factor, click the drop down list next to the word K Factor and then pick the with or without option, fill in the edit box to the right with the K factor you want to filter by and then hit the *Show* button. This option lets you show branches that contain the specific K factor if you use with or the branches that do not have the specific K factor if you use without.

Edit Branch: This option allows you to exercise the Modify Branch Parameters option from the listing. To edit a branch in the list highlight the branch by picking it from the list and then click the Edit Branch button. You can now perform the enabled edit functions of Modify Branch Parameters without picking the branch attribute block. Note that when editing a branch from the list you cannot use the Pick Branch Attribute Block, Reverse Branch or Update Branch buttons, they are disabled for a reason. The reason has to do with hiding the List of Branches and conflicts with editing the drawing.

View Branch: Highlight the branch of interest and click the View Branch button. The system will zoom in on the branch and a green arrow will point at its attribute block. This function can help find lost branch attribute blocks. If a branch appears on the list but you cannot find it on the drawing you can write down the start node and end node and use the Branch Menu→Erase Branch Snode Enode and enter the start node and end node and it will erase the branch. If a branch is on the list and you cannot see it or view it then you should definitely use the Erase Branch Snode Enode and erase it.

Print List: This option allows you to print a list of all the branches in the list box. When you click this option, the following dialog box of printing options appears.



Make sure Copies is set to the correct number and does not have erroneous data in it!

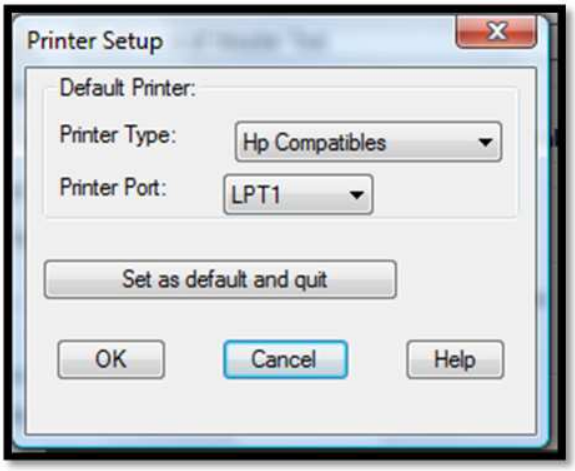
You can attach an identifying header to print; up to four lines and 80 characters per line are allowed. You must also select the print mode. By default the output goes to the LPT1 printer port. Since most printers are USB the best option is to use a network printer and re-direct LPT1 to the network printer.

One way to do this is:

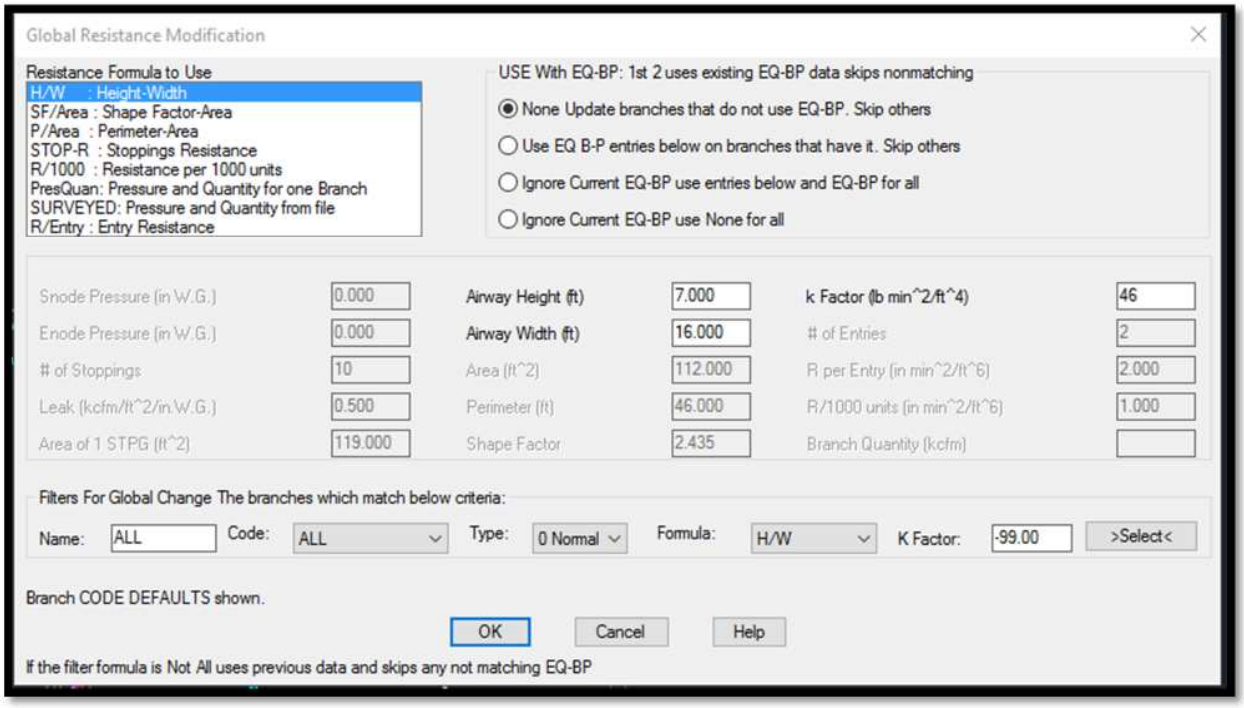
1. Go to Windows Accessories and select Command Prompt.
2. At the command prompt type in: *net use lpt1 \\server\printername*
3. Ask your Network Administrator for the server and printer name.

CAUTION: *If you try to print to LPT1 and there is no printer attached to the port or LPT1 is not re-directed to a valid network printer then the computer may freeze. In this case it is best to print to a file and open it in Notepad, Microsoft Word or any other word processor or text editor to print it from there. This also has the advantage of letting you change the fonts and other text formatting.*

The Set-up button brings up another dialog box, as shown below, for selecting the printer type and port. Since most printers are Hp Compatibles this is the only option right now. We may print directly to all Windows printers in the future but at this point it is about the same as printing to a file and opening it in Notepad or any other word processor. You should normally hit Set as default and quit to exit out of this dialog box.



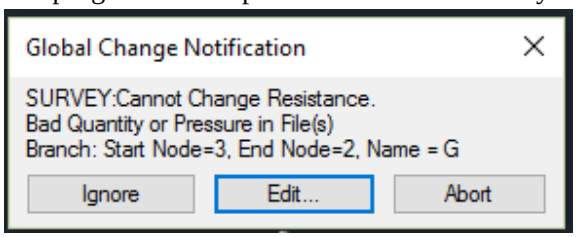
→Global Resistance Change: Use this option to change the resistance of multiple branches. The resistance can be changed for all branches with a specified name, code, type, formula and/or selection. It can also change branches regardless of name, code, type, formula or selection set by choosing ALL in the edit box or pop down lists.
When you select this option, the following dialog box appears.



Most of the information here is also in the help file if you click the help button on the dialog. However,

the below is easier to read and has important information.

The program has the ability to change branches based on surveyed data from a file using the SURVEYED: Formula Pressure and Quantity From File. But you must have the dwgname.P and dwgname.Q files for this to work. If either of those files do not exist it terminates. If there is a missing branch/node in one of those files or bad data then you can either Ignore, Edit for that branch or Abort. If you choose to Abort it will stop the program but the previous branches before you hit abort will still be



updated with the new resistance data.

Enhancements:
→Uses Length from Calculate R when possible. Usually the same as the polyline length but could differ
→More flexibility and ease for branches that use or do not use Equal Branches in Parallel (EQ-BP)

Resistance Formula to Use At Top Left Main Resistance Formula:
Use this list box to select the formula that will replace the existing formulas in the desired branches. This is the formula that will be used to calculate (and replace) all the branch resistances to be changed based on the Filters for Global Change.
USE With EQ-BP (Equal Branches in Parallel)***: 1st 2 uses existing EQ-BP data Skips nonmatching:***

→If you select None Update Branches that do not use EQ-BP Skip Others. It will only change existing branches that do not already use EQ-BP.

→If you select the Use EQ-BP circle it will only change existing branches that use EQ-BP.

→If you select Ignore Current EQ-BP use entries below then it will use the EQ-BP formula on the branches that meet the criteria and make them all EQ-BP based on the # of entries.

→If you select Ignore Current EQ-BP use None for all then it will convert/update all branches that meet the filters to NOT using EQ-BP.

(see below Filters and Formula)

Filters For Global Change The Branches which match the below criteria:
This section contains the parameters that need to match in order for the branch to use the new formula.

→**Name**: Use ALL to change all branches regardless of the name. Or enter a valid branch name. If you name one or more branches you would like to change.

→**Code:** Use this option to change all branches with the desired code.

→**Type:** Use this option to change all branches with the desired type.

→**Formula:** Use this option to change all branches with the desired formula.
***** Increases the power and flexibility dramatically. But has implications *****

1. If the main resistance formula matches the filter formula:
Then it uses existing data where appropriate if the formulas match:

a) STOP-R (Leak Branches) will use the # of stoppings from the previous branch data.

b) For the H/W formula you can enter X+(a number) to height. But only height and H/W formula.

c) Any existing lengths entered by hand not calculated will be used this is valid not only for matching formulas but for anyone that uses the length.

d) Use With EQ-BP. (Equal Branches in Parallel) It will compensate by making sure that previous branches using EQ-BP match and skip any that do not.Unless you use one of the Ignore options.

If you select None Update Branches that do not use EQ-BP Skip Others. It will only change existing branches that do not already use EQ-BP.

If you select the Use EQ-BP circle it will only change existing branches that use EQ-BP.

If you select Ignore Current EQ-BP use entries below then it will use the EQ-BP formula on the branches that meet the criteria and make them all EQ-BP based on the # of entries.

If you select Ignore Current EQ-BP use None for all then it will convert/update all branches that meet the filters to not using EQ-BP.

2. If the main resistance formula DOES NOT match the filter formula:
Then it uses the data in the dialog edit boxes to *replace* the branches with the filter formula with the main resistance formula at the top:

a) Selecting ALL/None replaces all branches with the main resistance formula and calculates the length based on the branch polyline. It **does** consider the name, type, code, and selection filters for the change. But NOT the K-Factor because that may not be defined if a branch has no existing resistance data. THIS OPTION COULD BE POWERFUL FOR CHANGING ALL OR A GROUP OF Branches TO A DESIRED RESISTANCE ALL AT ONCE. SUCH AS THE SURVERYED FORMULA: Pressure and Quantity from file. Or any formula for that matter. Surveyed likely would make the most sense.

b) If the filter formula is Not ALL then it will use existing branch resistance data for the length when applicable. Note: For formulas that use the length Calculate Resistance

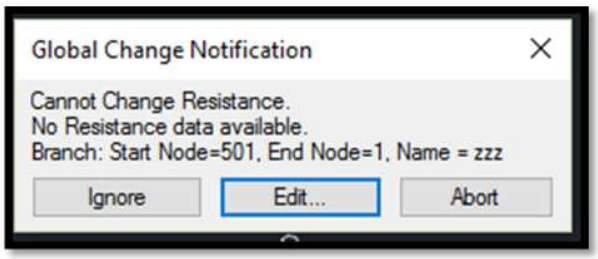
allows the user to enter a length that is NOT the actual polyline length which it initially calculates.

c) USE EQ-BP: If filter formula is ALL it replaces all branches matching the other criteria to EQ-BP or None regardless of what their state was before. The ignore options are available. For branches that have a resistance formula based on the number of entries. Basically everything except the last one R/Entry. If the Filter Formula is not the main it will still convert based on the number of entries in the "other/old" formula.

→**K-Factor:** Use this option to change all the branches with the desired K-Factor. If this is -99.0 then it is ignored and all branches will be changed regardless of the K-Factor.

→>**Select Button<:** If you use the Select button the command line prompts allow you to draw a polygon that encloses the region containing the attribute blocks of the branches you want to modify. Only the branches inside the polygon that meet the selection criterion are modified.

Enter a Name or leave ALL then pick the branch code, type and formula from the drop down lists and enter the K Factor then click the select button if you only want to change a selection set of branches. To change all the branches selected use the following settings: Name: ALL, Code: ALL, Type: ALL, Formula: ALL (see notes above as for the consequences), K Factor: -99.00. Setting these values to anything else will filter accordingly. If you do not use the Select button the program will change all the branches in the drawing based on the Resistance formula and filters chosen. Branch length from Calculate R is used (not necessarily the actual poyline length but would be unless you manually entered a length in

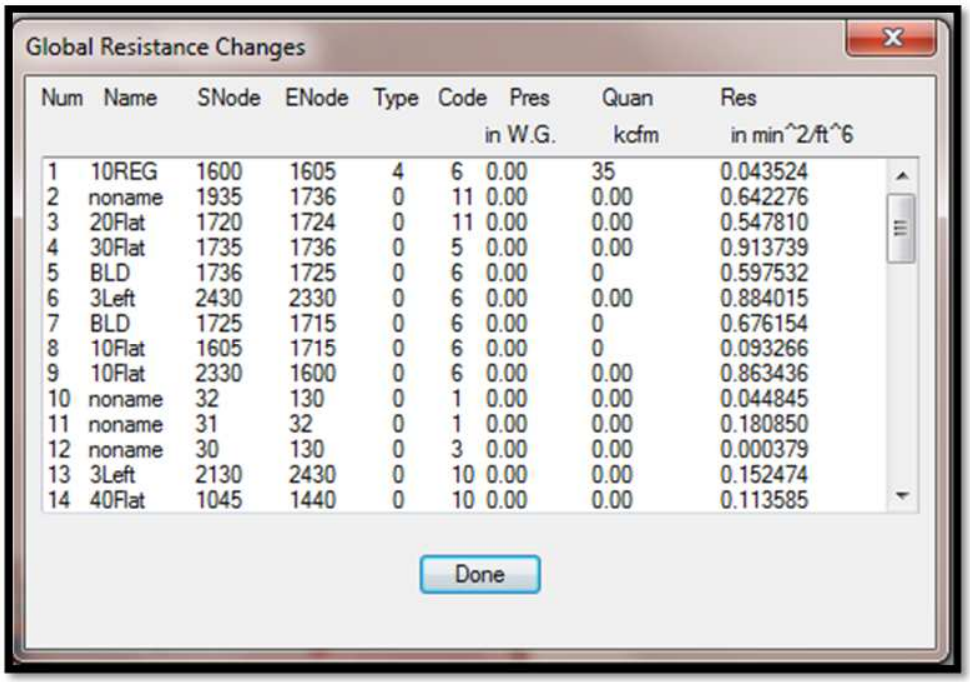


CALCR for that branch). If any of the branches in the selection do not have a resistance formula and data available then the following dialog box appears and gives you the option to either ignore the branch resistance or edit it and change the resistance as desired. If you choose to Abort it will stop the program but the previous branches before you hit abort will still be updated with the new resistance data.
The Edit button brings up the Modify Branch dialog box as in section 4.2.

The parameter values correspond to the defaults in the branch configuration file based on the last code you selected from the dropdown list. You can change any of them; usually only the K factor is increased to reflect the deterioration of the airway walls due to age. However, if all you want to change is the K-Factor you may want to consider using Global K-Factor Change but this option is more flexible based on formulas and filters. There is a hidden feature for the Airway Height only: If, for example, you put X+3 in the Airway Height box it will add 3 to the existing Airway Heights. If this feature is useful to you in other areas you may request it. Right now only for the Height and Width Formula.

Note: This option does not change the default values in the cfg file. To change the defaults use Configure Branches.

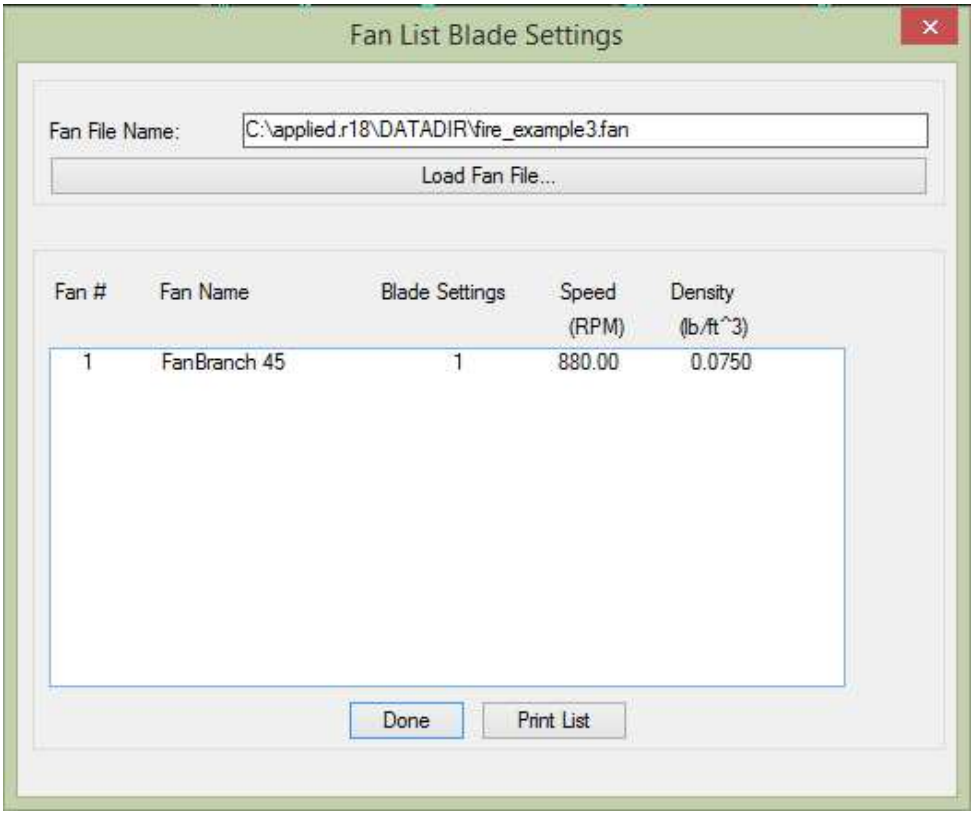
After the changes have been made the following dialog box comes up showing the new resistance values and the branches that were changed:



If a branch is detected that does not have any resistance data associated with it, i.e. Calculate R was not used. Then the program catches this and ask you if you want to Ignore or Edit the branch in question. It does this for all the branches found in the selection set that have no resistance data.

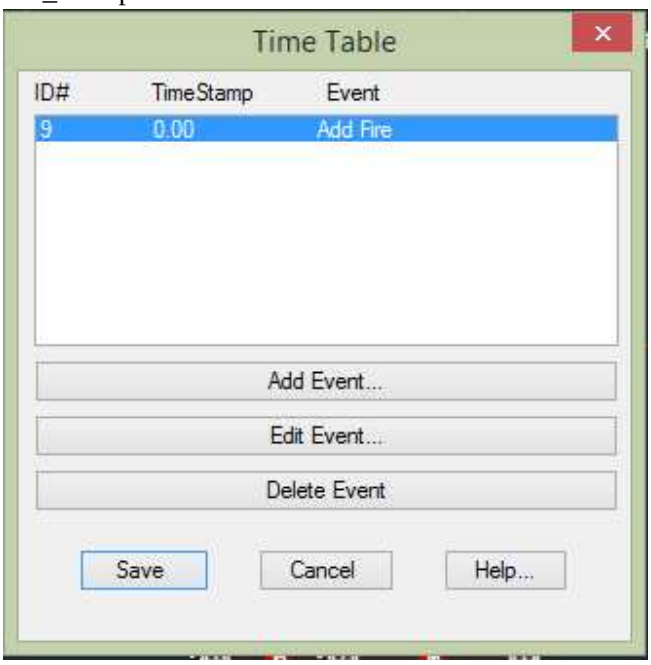
→List Fans

The list of fans appear in a dialog box as shown in the following example. This option allows you to load in any fan files that were created in the option Define/Edit Fans. It displays the fan reference number, name, blade setting, axial speed, and air density that was entered. It also allows you to print the list.

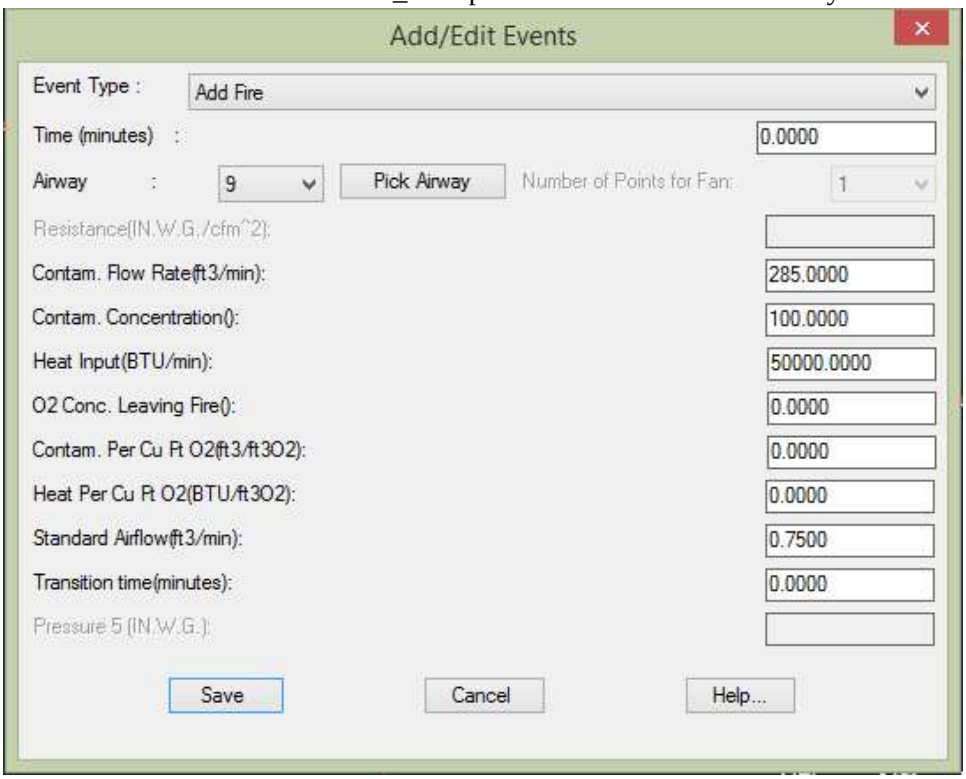


→Time Table

This is where you can add/edit and delete events that occur during the simulation. It is the same as the discussion above under Add Event in real-time during Run MineFire. Below is the fire event for fire_example3.



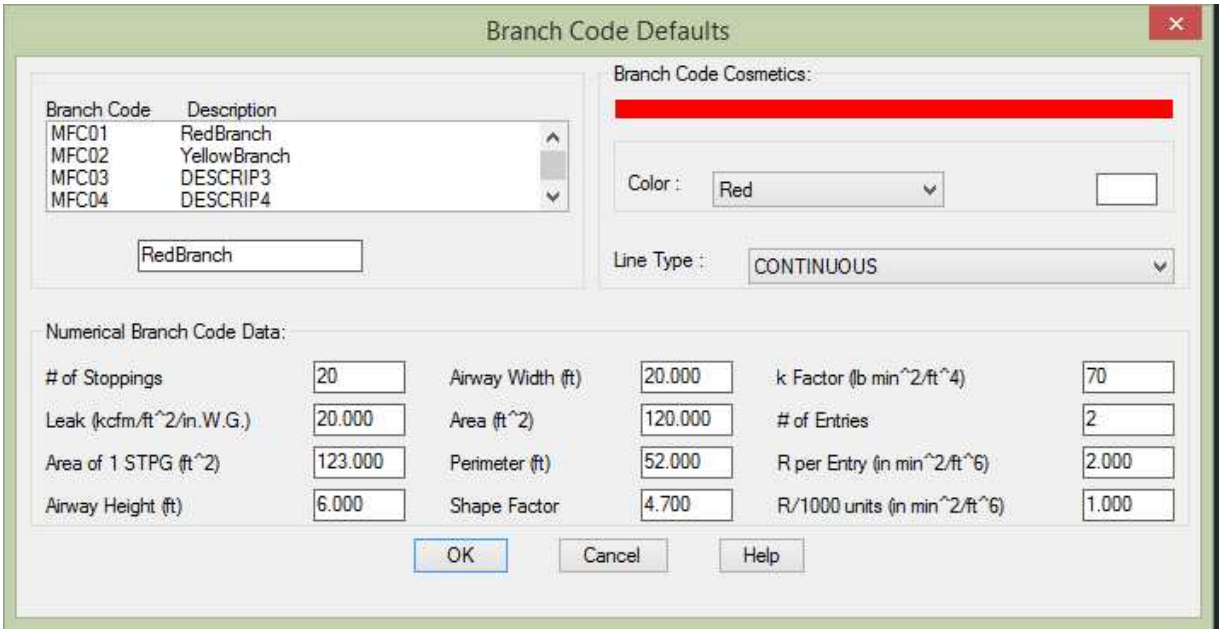
Once you add/edit or delete an event here then start the simulation the event is pretty well set in stone, so to speak, and will happen during the simulation at the TimeStamp indicated here. Below is the fire event when we select Edit Event for fire_example3. The fire starts immediately at 0.000 minutes.



Important Note: The Time Table events are stored in a file called <dwgname>.tbl (in this case fire_example3.tbl) and are NOT saved or stored with the drawing itself, so if you add/edit or delete events it is permanent unless you run Time Table again even if you do not save the drawing.

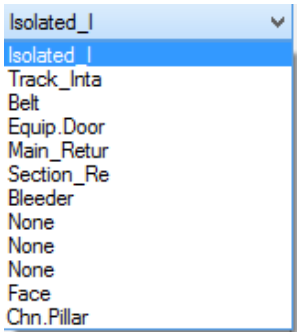
→Configure Branches

The branch configuration file contains the color, line type and default parameter values for calculating the resistance for each branch code. The parameter values you specify will appear as the code defaults in the resistance calculation edit box when you are creating a new branch or modifying an existing branch. When you select this option, the following dialog box appears.



Each ventilation drawing can have a unique branch configuration file. A default configuration file called VENTCODE.CFG is provided with the software and is automatically loaded into the configuration file (drawing name).cfg. You can edit the file as required. Some parameters are not relevant to all branch codes and can be ignored, for example, # of stoppings applies to leakage branches that are not currently supported in MineFire.

Branch Code-Description: Branch codes are used to specify a group of branches that will have the same appearance on the drawing, i.e., the color, line type and branches for which a common set of parameters will be used to compute the branch resistance. The list of codes and their default names are given in the code list, but you can customize the names by adding a Description which will appear in all references to the branch code in other options. The Description display will accommodate a maximum of 15 lower case or 12 upper case characters. Below is an example of some more meaningful Descriptions than the RedBranch and YellowBranch in fire_example3.



Branch Code Cosmetics: You can specify a default Color and Line Type**:** for each code.

Color: You can select the color by scrolling through the drop down list or type in the color. Except for the standard AutoCAD colors, the color are designated by a number. When you select or type in a color number, the actual color shows in the dialog box, so you will know how the branch will appear in the

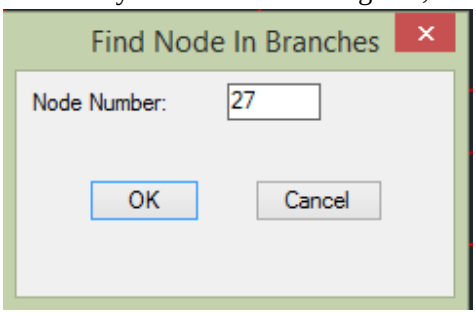
drawing. To assign a color by typing, key in the number and press the Enter key. Adjacent color numbers tend to be similar in appearance. This feature makes it easy to assign different shades of one standard color to a common group of branches, such as all intakes.

Line Type: Select CONTINUOUS for MineFire. Work in progress.

Numerical Branch Code Data:: You can assign default dimensions to the various parameters which are used to compute the branch resistance. All parameters are given a default value, but not all apply to a specific branch code. In addition, you may want to compute the resistance for rectangular intakes using the height-width formula and other shaped intakes with the perimeter, area and shape factor information. Therefore the area default does not need to equal the product of the airway height multiplied by airway width.

→Find Node in Branches

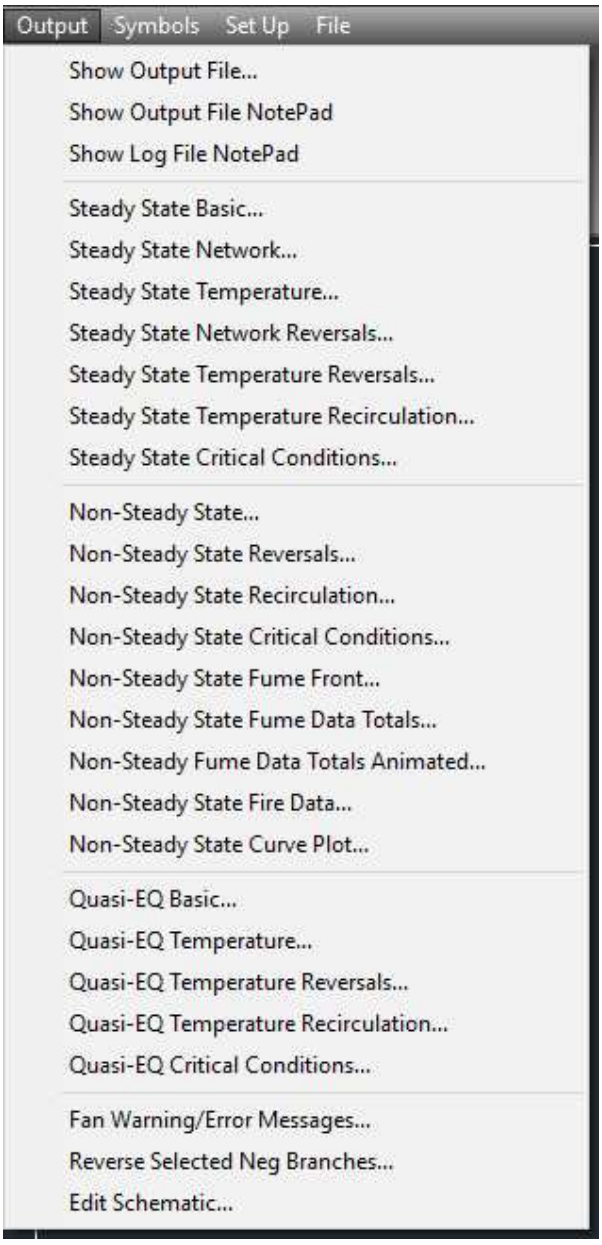
This option is mainly a problem solving utility and it searches all the branch attribute blocks for a certain node that you enter in the dialog box, whether it is the start node or the end node of the branch.



If you click OK here on fire_example3 you will notice that it shows a green arrow pointing to the first branch block that has the end node of 27 then ask if you want to Zoom Previous (Yes/No) just enter N for No, then it shows a green arrow pointing to the next branch block that has a start node of 27, again you may enter No on the Zoom Previous then it points to another branch block because node 27 is a duplicate node in the system. Sometimes it is a judgment call to use this function or the Find Node under Node Menu.

Output Menu

Almost all of the options under this menu directly read from the output files created by Run MineFire, mainly the file called <dwgnameou.txt> or for this example fire_example3ou.txt. The only exceptions to this are the options of Show Log File, Fan Warning/Error Messages, Reverse Selected Negative Branches, and Edit Schematic.

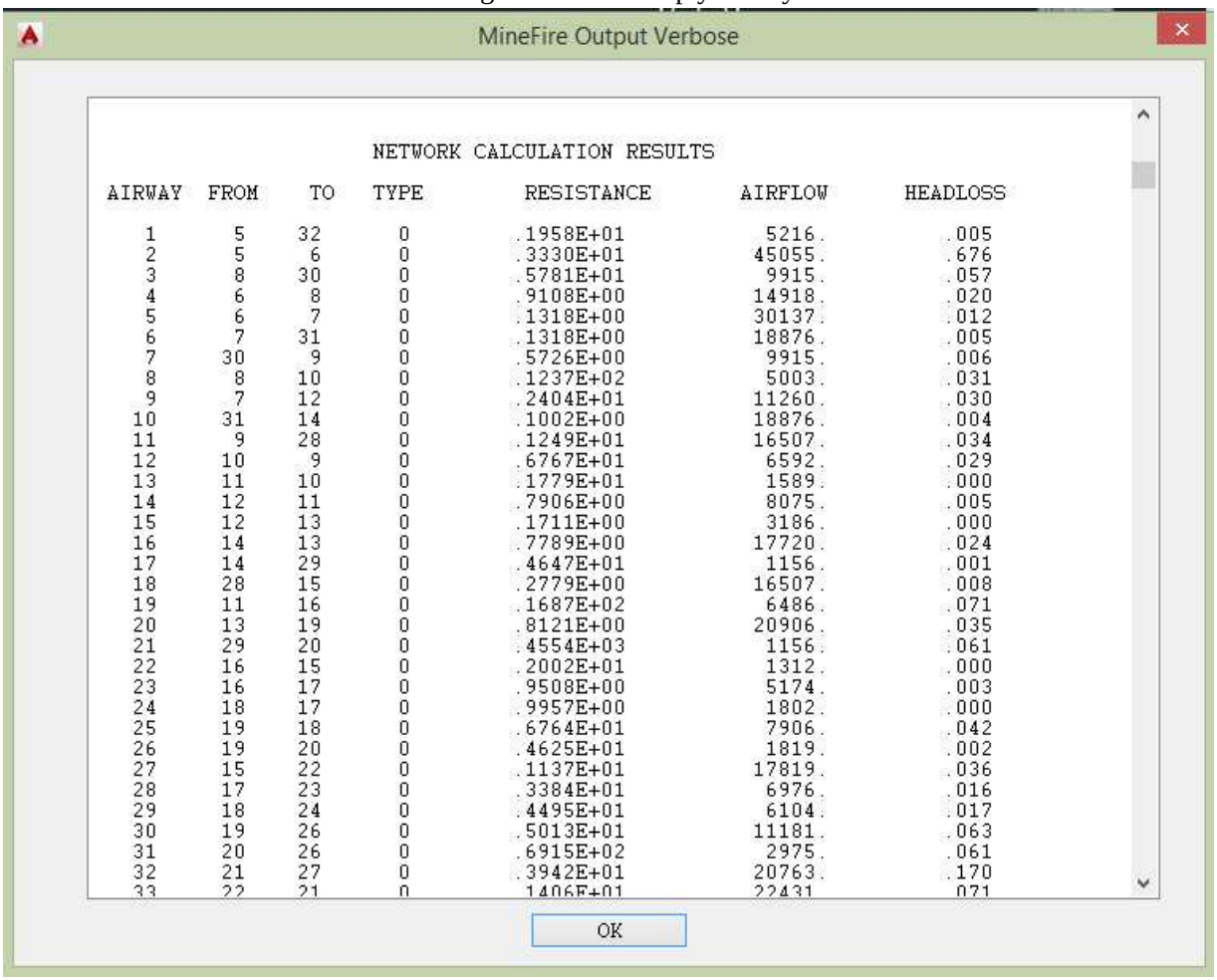


Important Note: The Output file has more stages or phases than what can be seen in real-time, such as the Steady State, the Air Reversals, Recirculation, Critical Conditions, Fume Front etc.. However, the real-time portion has more parameters for Airways and Junctions and is accurate to more decimal places. The Output menu here simply breaks down the output file picking apart each phase or stage you may be interested in. Do not let the number of menu options intimidate you it is all pretty much the same interface as was in the real-time View Results, except for cases where a simpler but still similar dialog interface

makes more sense, such as Air Reversals or Recirculation. Anytime you select something from the menu if you forget where you are keep an eye on the Phase and Time near the bottom of the dialog. Also it is advised to run the simulation to completion or some of the options may not have data, such as Quasi-EQ.

→Show Output File

This option reads directly from the output file and brings up the same dialog as Verbose in real-time. The dialog should look okay and everything lined up as long as you have the Courier New font installed on your system. This is the complete history of the simulation with all of the data in text format. Below is what it should look like for fire_example3 scrolled down to the NETWORK CALCULATION RESULTS section. If it does not look like this or things are not lined up you may not have the Courier font it needs.



→Show Output File NotePad

This option shows the output file in Microsoft Windows NotePad by “shelling out” of AutoCAD and bringing up the file in Notepad. This may be a more desirable way to view/search the output file than the above Show Output File as Notepad has more of these features and you do not have to worry about the font. There are two things to be aware of here. The first is that you should not change anything in the output file in Notepad as the other options on the Output menu read from this same file, you could get unexpected results or even crash AutoCAD. The second is that you can go back to AutoCAD, with the file in Notepad still open and press Enter/Return to terminate the shell command and look at the output

file in notepad beside or at the same time as other Output menu features and then close Notepad when desired. It is not recommended to keep the output file open in Notepad while running the simulation.

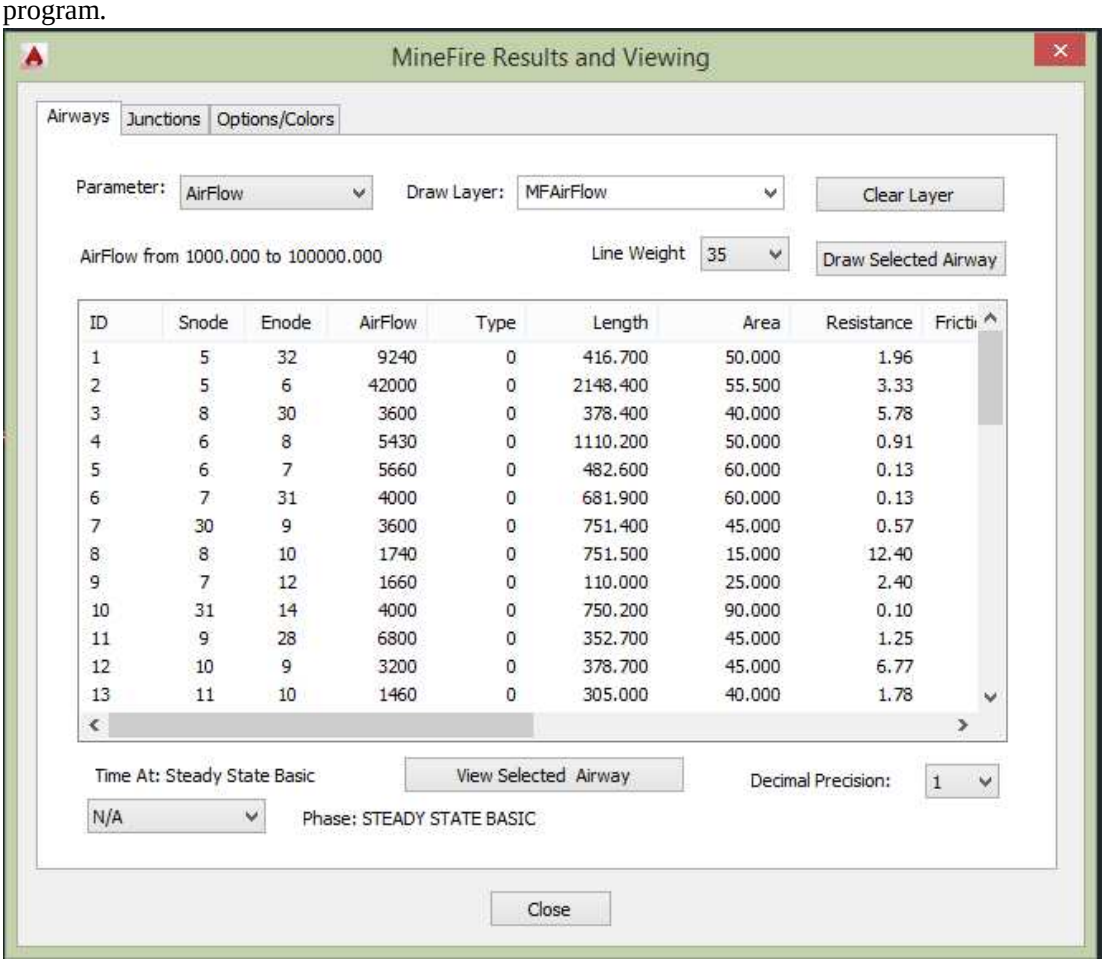
Important Note: There are some things in the output file that are not on the menu (as of yet). See the important note under the Run MineFire and Verbose section.

→Show Log File NotePad

This option brings up the simulation log file in Notepad. This file is essentially every event that is logged by the program from the start of running the simulation to the end. This file would likely be used if something went wrong and you wanted to see what all the simulation did up to that point. However, it could also be used just to see what all happened during the simulation and iterations or correction factors etc..

→Steady State Basic

This menu option brings up the standard MineFire Results and Viewing dialog as discussed in the real-time section. The difference is the information in the dialog box. Steady State Basic is just the input data to the program under BASIC DATA FOR AIRWAYS IN THE NETWORK and DATA FOR CONCENTRATION AND TEMPERATURE CALCULATION found in the output file. Shown below for your convenience. It has the power to post and color up the drawing with any of the parameters listed and could be a good alternative to List Branches as it has more information about the input to the program.



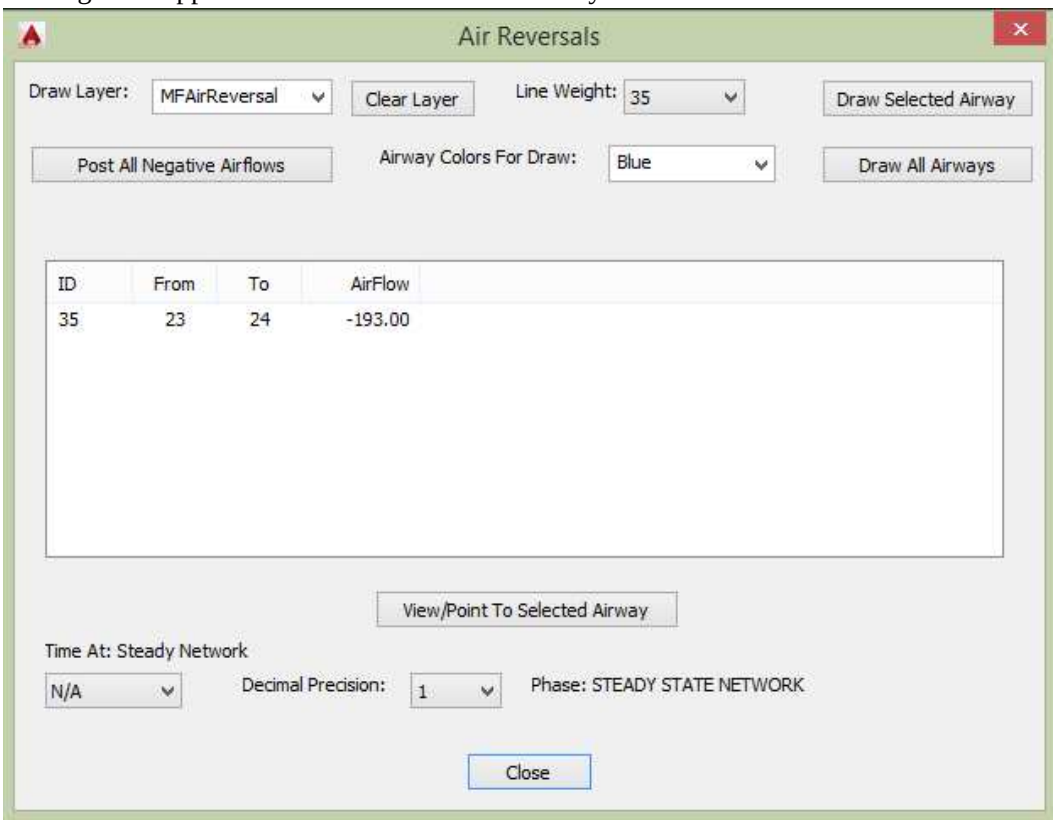
Important Note: All of the options that start with Steady, Non-Steady, or Quasi-EQ can be found in those sections of the main output file report. In case you need to compare data or see where it came from.

→Steady State Network
Reads from the NETWORK CALCULATION RESUTLS section of the output file and presents the same dialog interface. The junction data is the same as that of Steady State Basic. This is the only option that lets you post the Airway ID on the branches. Fan branches are posted at the bottom with the airflow and pressure.

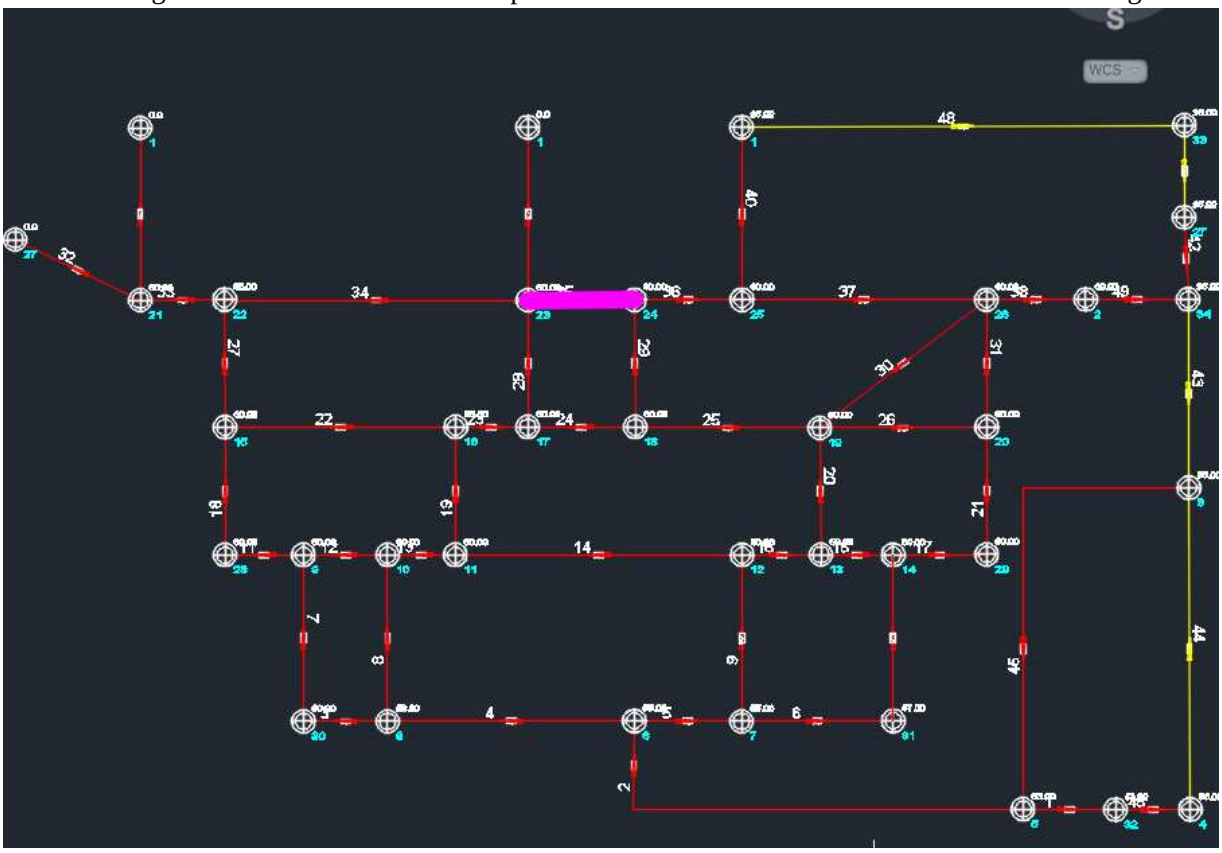
→Steady State Temperature
Again same dialog just different data this time from the Steady State Temperature portion of the output file. This option reads from the OUTPUT OF THE TEMPERATURE PART OF THE PROGRAM and the headings of TEMP. AND CONCENTRA. AT AIRWAY ENDS, HEADLOSS IN AIRWAYS for the Airways and PARAMETERS OF AIR IN JUNCTIONS. On the Airways tab it list the DeltaQ, but at this point in the program that is not applicable, so you can disregard. It was just a programming convenience as the Non-Steady uses the DeltaQ and the dialog is populated in the same way.

Important Note: The terms CH4 and Methane are interchangeable as are the terms Contaminate and Fumes.

→Steady State Network Reversals
This is when air reversals occur under the Steady State portion below the NETWORK CALCULATION RESUTLS to mean that even in the steady state without the effect of temperature there is already air flowing in an opposite direction than was entered by the start node and end node of the branch.



The above it what it looks like for fire_example3. It shows the negative airflow in the list with the branch ID number. So just go ahead and set the Line Weight to 140 and color to Magenta and click Draw All Airways (even though there is only one in the list). Depending on where you are zoomed in you should see something like the below. Remember to press Enter/Return to return to the Air Reversals dialog.



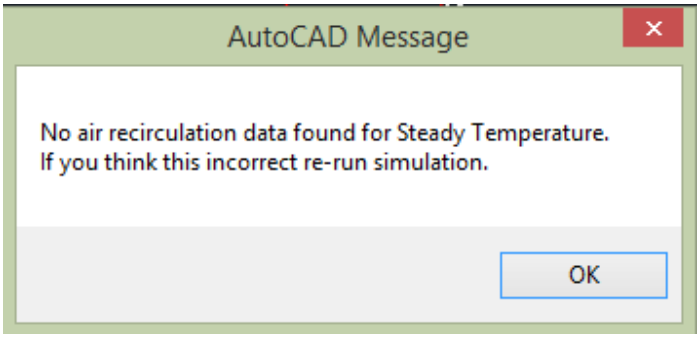
Once your are back at the Steady State Network Reversals dialog box you can Clear the Layer and/or click Post All Negative Airflows which will put the Negative number above the branch(s) in this case it should put -193.0 above branch 35 if you have the decimal precision set to 1. It does not post any numbers on any of the other branches only the ones in the list. Sometimes it is helpful to select an airway in the list and use a combination of View/Point to Selected Airway and Draw Selected Airway, especially if you do not have an excessive amount of air reversals. Draw Selected Airway uses the Line Weight and Color as does Draw All Airways just does it one at a time as you select them from the list rather than all at once.

→Steady State Temperature Reversals

Exactly the same interface as above the only difference is that you may have more or less or the same (as in this case) amount of air reversals in the Steady State Temperature part of the program.

→Steady State Temperature Recirculation

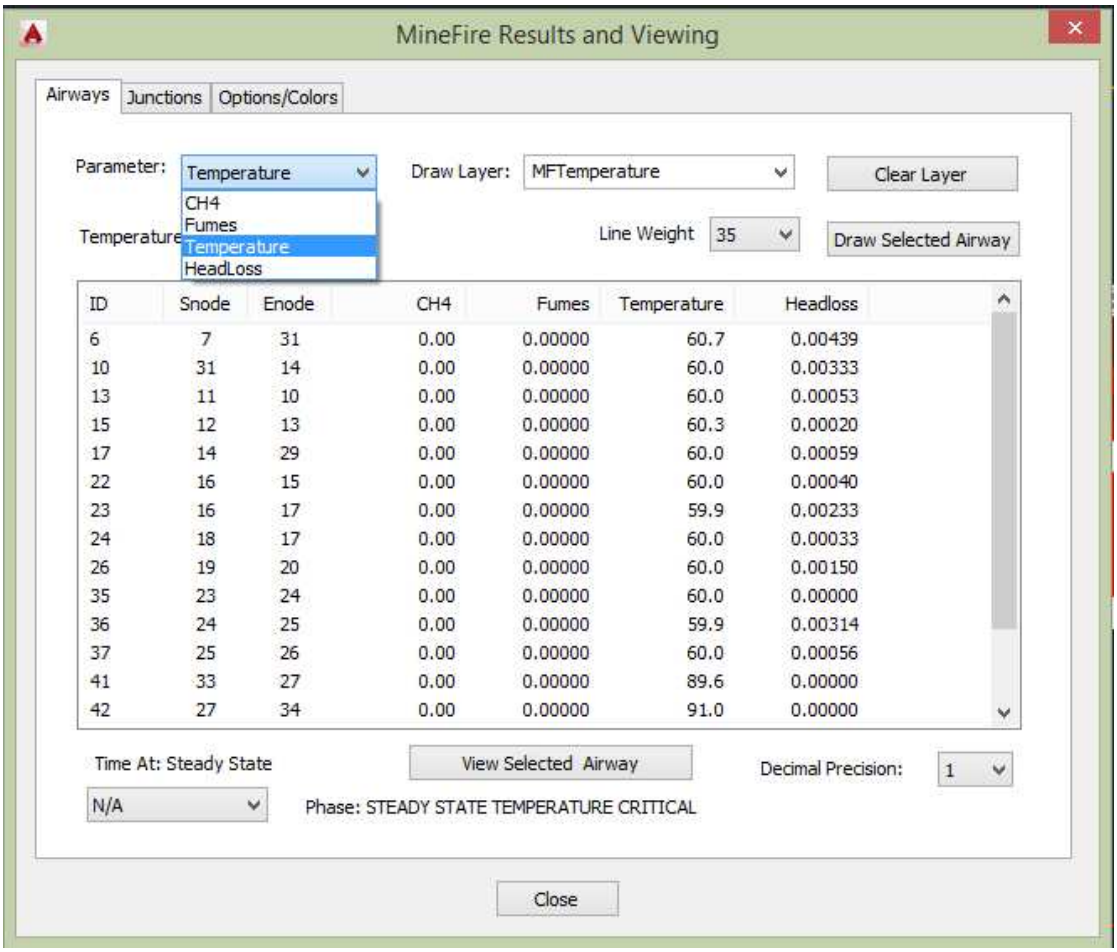
Shows any air recirculation paths in the Steady State Temperature part of the program. In this case for fire_example3 there are none and you get the following warning.



This will be a common type of warning for anything under the Output menu that does not have any data in the output file or if the output file were somehow corrupted. If you believe otherwise you should re-run the simulation (Run MineFire) to completion. The only place in fire_example3 that has recirculation is at the end in the Quasi-EQ portion of the program so we will wait until we get there to discuss it.

→Steady State Critical Conditions

This could have been more accurately called Steady State Temperature Critical Conditions as it gets this information from the temperature part of the Steady State. Again we have the same standard MineFire Results and Viewing dialog box with the 3 tabs across the top (Airways, Junctions and Options/Colors). The difference this time is that the Airways and Junctions list *only* list the Airways and Junctions in which there are Critical Conditions.

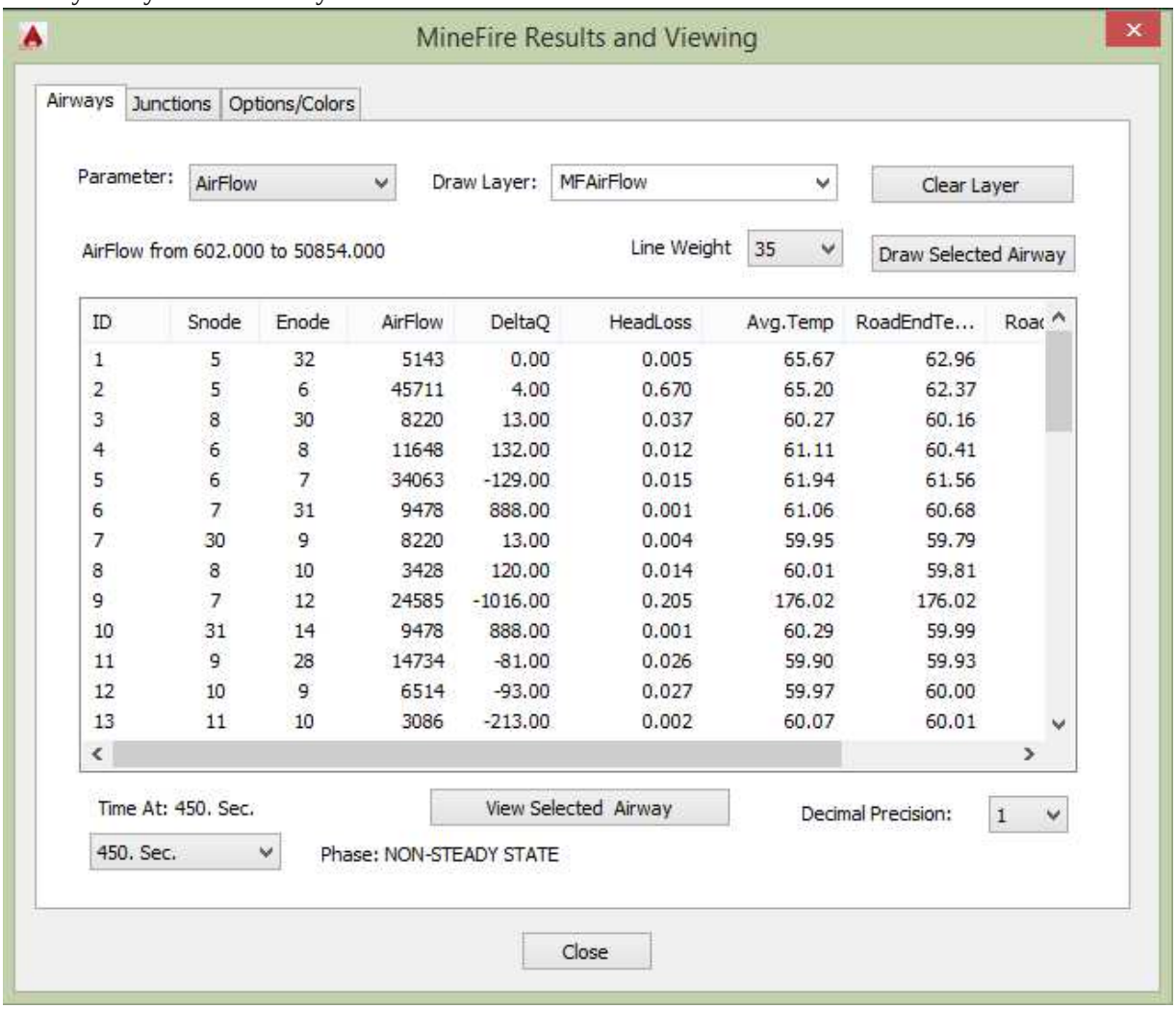


One quick way to help determine the Critical Conditions is to select a parameter on either the Airways or Junctions tab and see the minimum and maximum range. Remember the critical conditions are set by the first Run MineFire dialog under the Control Card 2 under Pressure Drop Warning Limit, Fume Warning Limit, CH4 Warning Limit, and Temperature Limit.

→Non-Steady Sate

This brings up the following dialog box, same interface as before, with one important difference. The difference is that down at the bottom left the Time drop down now has the time increments (or intervals) that are available in the output report.

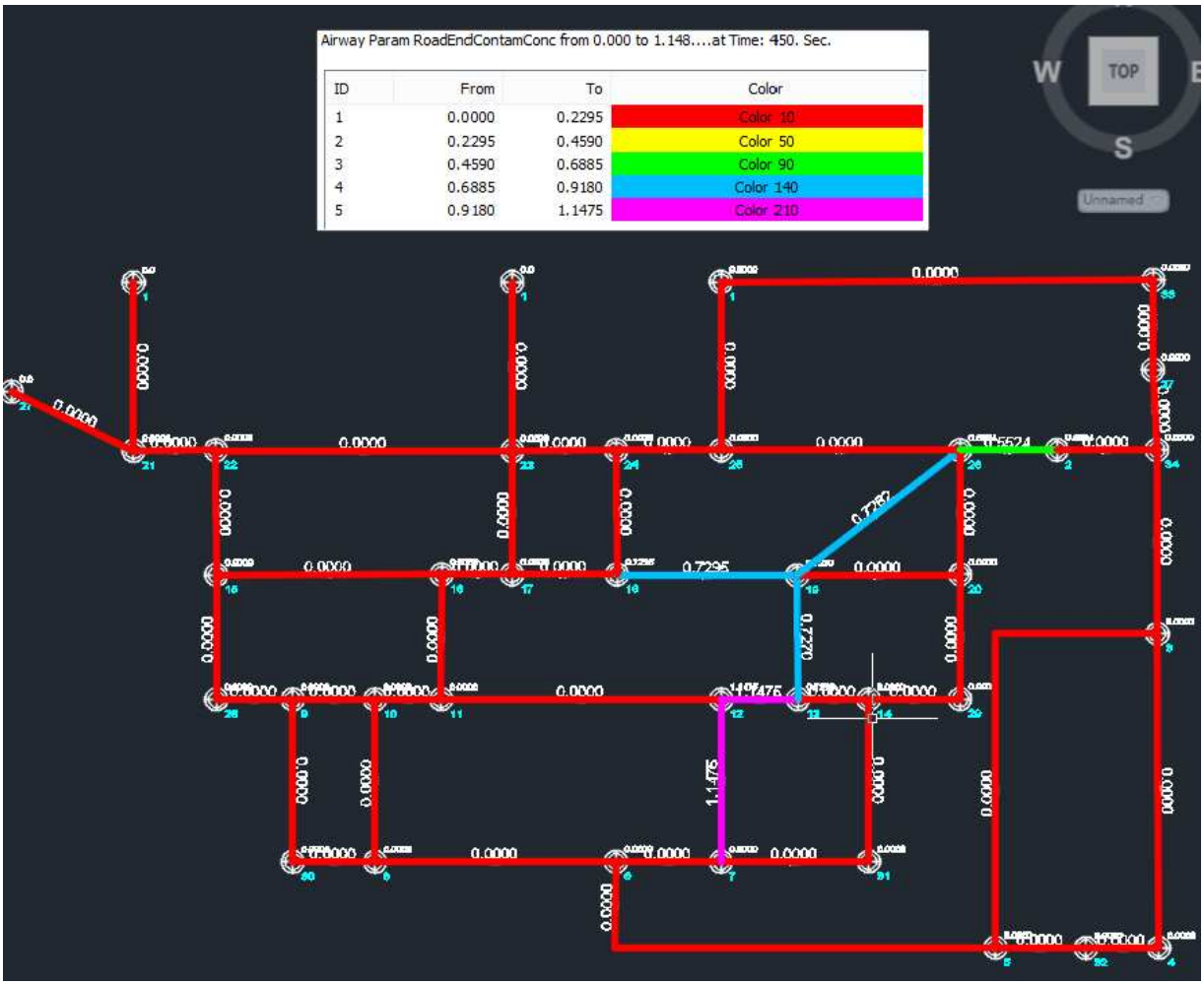
Important Note: Both the Airways tab and the Junctions tab have a time drop down so you could vary seeing the junction parameters and airway parameters at different times but it is most likely that you would want them to match. So if for example in the following dialog box you selected 450 Sec. for the Airways tab you would likely want to set the time interval to 450 Sec. on the Junctions tab as well.



This is really the same data that you would see in real-time, if you were to View Results at that same time interval. However, there is less data available than in real-time and the numbers have less decimal accuracy. Each time you select a different time interval for either the Airways tab or the Junctions tab the

information in the list changes and you can post numbers and color up the drawing at each time interval individually.

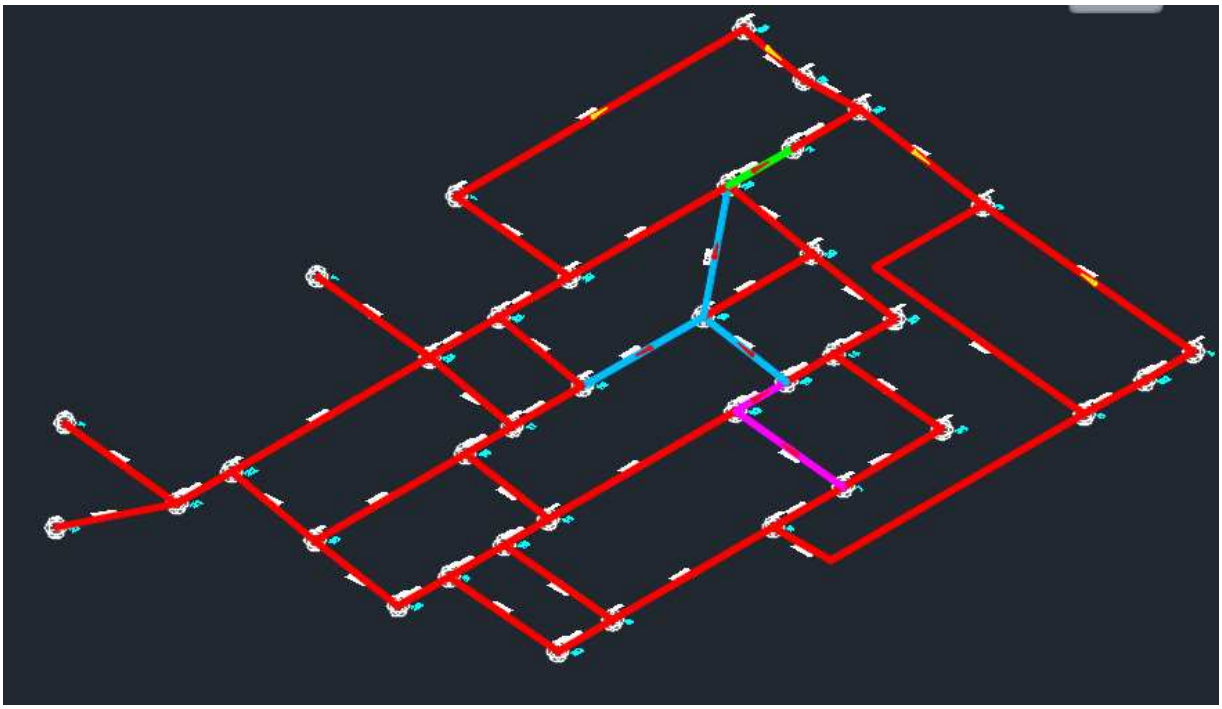
Example of Non-Steady fire_example3:
If you are not already looking at the dialog above select Non-Steady State from the Output Menu. Let us assume we want to see the fumes (RoadEndContamConc) on the airways and junctions at 450 seconds. So first select 450 Sec. from the time drop down on the Airways tab then select RoadEndContamConc from the Parameter drop down. Notice how the layer changes to MFRoadEndContamConc this is the layer that it is going to draw the 3D overlaying polylines on for the color ranges. Also you can see that the RoadEndContamConc ranges from 0.000 to 1.148 and if you scroll to the right on the main list you can see that it is mostly 0.00. Now set the Line Weight to 53 and the decimal precision to 4 then select the Junctions tab. Here on the Junctions tab select 450 Sec. from the Time drop down and the ContamConcentration parameter at the top and again 4 for the decimal precision. Now select the Options/Colors tab. Here we could modify the color ranges using Add/Delete/Change, but just go ahead and keep the defaults and click View Colors and Parameters on drawing. It will post the airway fumes, press Enter, it will post the junction fumes, press Enter it will color the Airways and press Enter to return to the MineFire Results and Viewing dialog. Go ahead and click Close. You should see something similar to the below. (Legend added here with the Windows Snipping Tool).



As you can see from the above most of the lines are Red (Color 10) which is expected since most of the airways with fumes were 0.00. Airway 9 (from junction 7 to 12) is the airway where we said the fire is and that it is the more Magenta (Color 210) with the maximum fume concentration 1.1475. The Blue (Color 140) are the airways that the fume falls within 0.6885 and 0.9180. Finally the Green (Color 90) is the lowest non-zero branch at 0.5524.

Make sure you are closed out of the MineFire Results and Viewing dialog. Now maybe with this many decimal places the numbers are too cluttered or large in that case we could go to Output and Edit Schematic at the very bottom and change the Size of Node Parameters to 20 and the Size of Airway Parameters to 30 then click Update DWG Exit. You should progressively see all the numbers getting smaller until it is done with the whole drawing.

Just for fun go to Display→3D Orbit and wait a second depending on your computer, graphics card and version of AutoCAD and when you get the little obit mouse cursor twist it around a bit. Then press Esc to exit out of the 3D Orbit and go back to Display→3D Views and select SW Isometric. It should look like the below.



Now you can just go back to Display→3D Views and select Top to get you back to the top view. This example does not have significant elevation changes so there is little use for the 3D features. Normally you would layout the mine plan in Top view but if you have two junctions that are the same in the XY plane and only differ by elevation you could use the 3D Orbit to connect a branch between such nodes.

Now we may want to get rid of the colored thick polylines for fumes or view a different time interval or parameter. So we can go back to Output→Non-Steady State and select either the RoadEndContamConc parameter or select the MFRoadEndContamConc from the Draw Layer drop down and click Clear Layer. Or if you want to keep it for later you could use Display→Modify Layer and turn off or freeze the layer MFRoadEndContamConc. You will have to set some other layer to current first, maybe just layer 0.

The colored up fumes on the airways should disappear. However, the numbers posted on the airways and junctions will stay until you post another parameter. This is the same for all options under the Output menu that post parameters and numbers on the airways and junctions.

→Non-Steady Reversals
Same interface as Steady-State Reversals with the main difference being that you can select the time interval that you want to see if there are any reversals for.

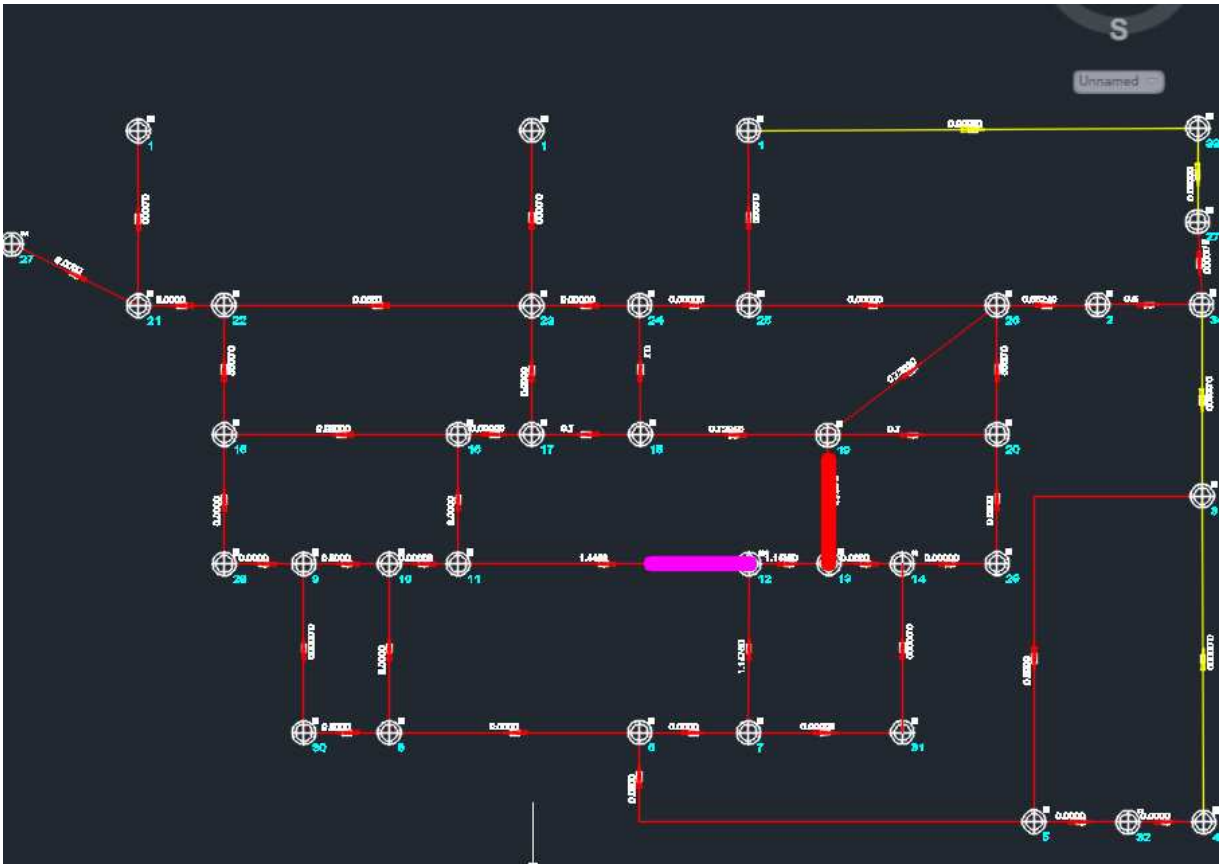
→Non-Steady Recirculation
Same interface as Steady-State Temperature Recirculation with the main difference being that you can select the time interval that you want to see if there are any recirculation for.

→Non-Steady Critical Conditions
Same interface as Steady-State Critical Conditions with the main difference being that you can select the time interval that you want to see if there are any critical conditions for. Also here you want to make sure that you are posting the Critical Conditions in the airways with the Critical Conditions on the junctions at the same time interval. It is possible to have a time interval that does not have critical conditions for one or the other (airways or junctions). The time interval drop down will reflect this and it will (on the Airways tab and Junctions tab) only show the time intervals for which there were critical conditions. For example, considering fire_example3, there could be critical conditions on the Airways tab at 150 Sec., 450 Sec., and 750 Sec. and so on, but on the Junctions tab there may only be critical junction conditions at 150 Sec., and 750 Sec. so that there were no critical conditions for the junctions at 450 Sec. so that interval would not be on the time interval drop down on the Junctions tab. It is likely that this would be a rare occurrence. Below is what it colors the drawing at 450 Sec., Fumes for Airways, Line Weight 60 etc..

Notice that it *only* colors the airways that have critical conditions and are in the main Critical Conditions list on the Airways tab.

→Non-Steady State Fume Front

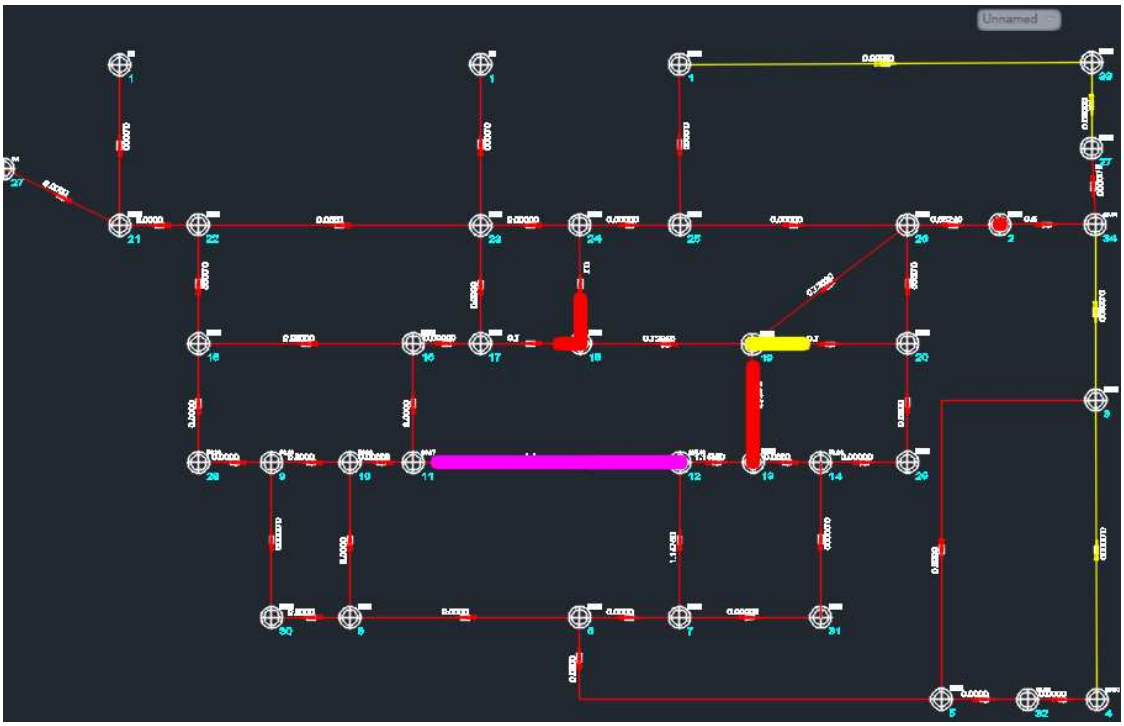
Same interface but a little more complicated. First your drawing should be to scale for using this option, meaning, the lengths should be that as calculated by Define New Branches. We have replaced the Airways tab at the top with the FumeFront tab. The parameters (Position, Temperature, Fumes and Methane) are used only for the color ranges and the number to post above the branch. The 3D polyline overlay will only draw the polyline to the Position on the actual branch polyline. As you select different time intervals you can see some airway ID's appearing and disappearing in the list. The Junction tab in this case just list the standard Non-Steady State junction information for the time interval chosen. So on the FumeFront tab set the Parameter to Fumes and the Line Weight to 100 and the time interval to 150 Sec. then choose the Options/Colors tab and click View Colors and Parameters on Drawing. You should see the below.



As you can see it shows the position of the fume front from the start node of each of the airways.

Important Note: At this point in the program we have not checked to insure that the branch has not been reversed, if it is the start node and end node would be flipped, so the program would show incorrect results showing the fume front from the start node which is actually the end node of a reversed branch. So make sure to check for reversals first. This may be changed in the future, but for now it would reduce the speed of the program so we need to check manually.

Now repeat the process for each time interval and watch what happens. See below.
450 Sec.



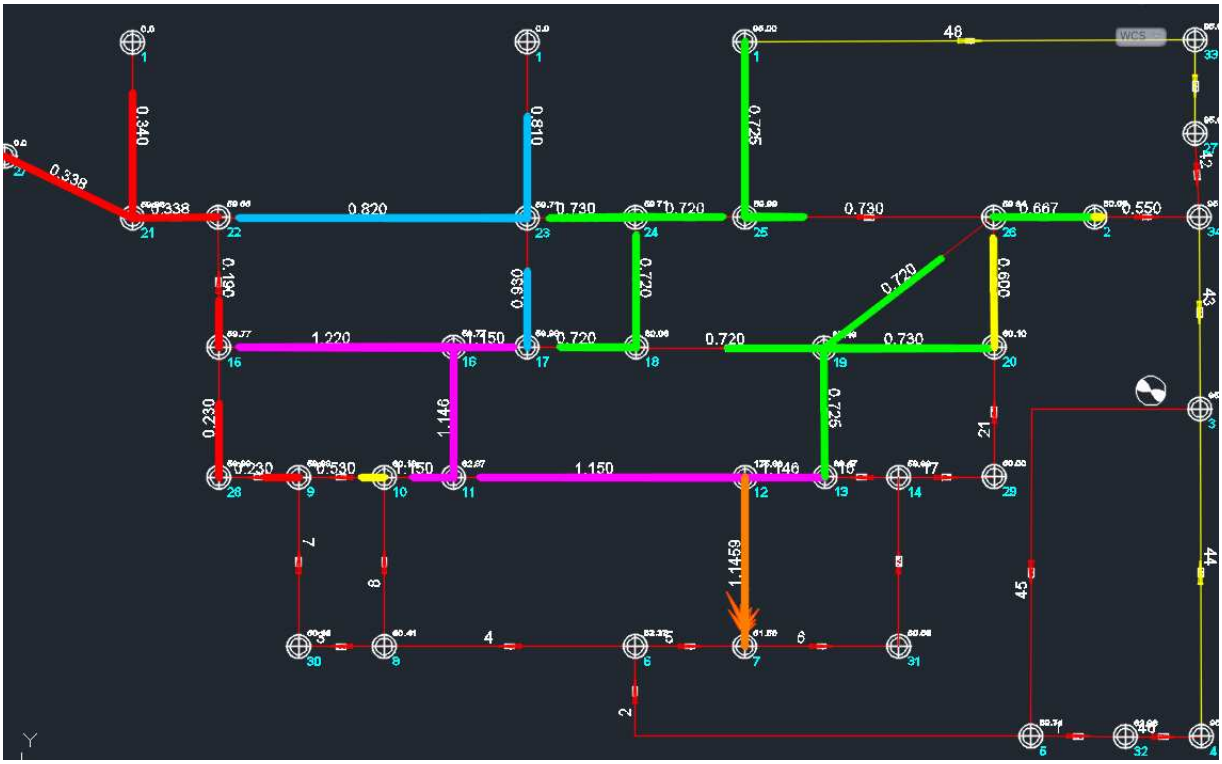
750 Sec.

1050 Sec.

You can play around with this to see the progression of the fume front. What we did above was not entirely accurate as we did not Clear the Fume Front layer at each time interval. Looking at the fume front at specific time intervals gives a snap shot of where it is at a certain time as it can over time clear out from some airways and move into others. You could also use Branch Menu→Get Polyline Length to test the lengths of the thick fume front polyline or if you need to know the distance while looking at the drawing.

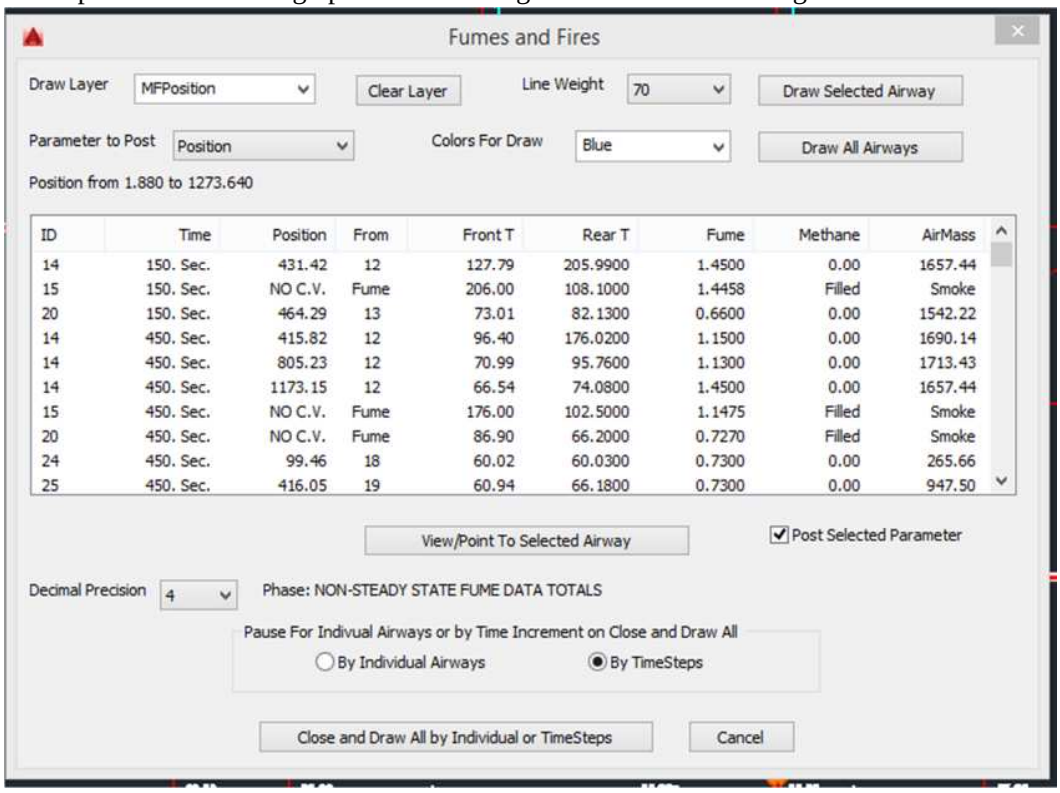
→Non-Steady State Fume Data Totals

As you can see from the previous Non-Steady Fume Front it only shows the Fume Front in the airways at the point in time you chose for the time interval. This loses the previous airways which are filled with fumes. So this option and the next Non-Steady Fume Data Totals Animated shows a more accurate progression of the Fume Front. This brings up the standard dialog once again, however, it list all the totals of the Fume Front from beginning to end for the time interval chosen. So that at any time interval you can see where the fume front is and was and the concentration at that time interval. If you choose the last time interval you will get the total from beginning to end of the simulation. As well as being able to color the drawing by the parameter chosen and post values on the airways with fumes. Below is what it would look like for fire_example3 at 1650 seconds and colored by fumes with the fume concentrations posted on the airways.



→Non-Steady State Fume Data Totals Animated

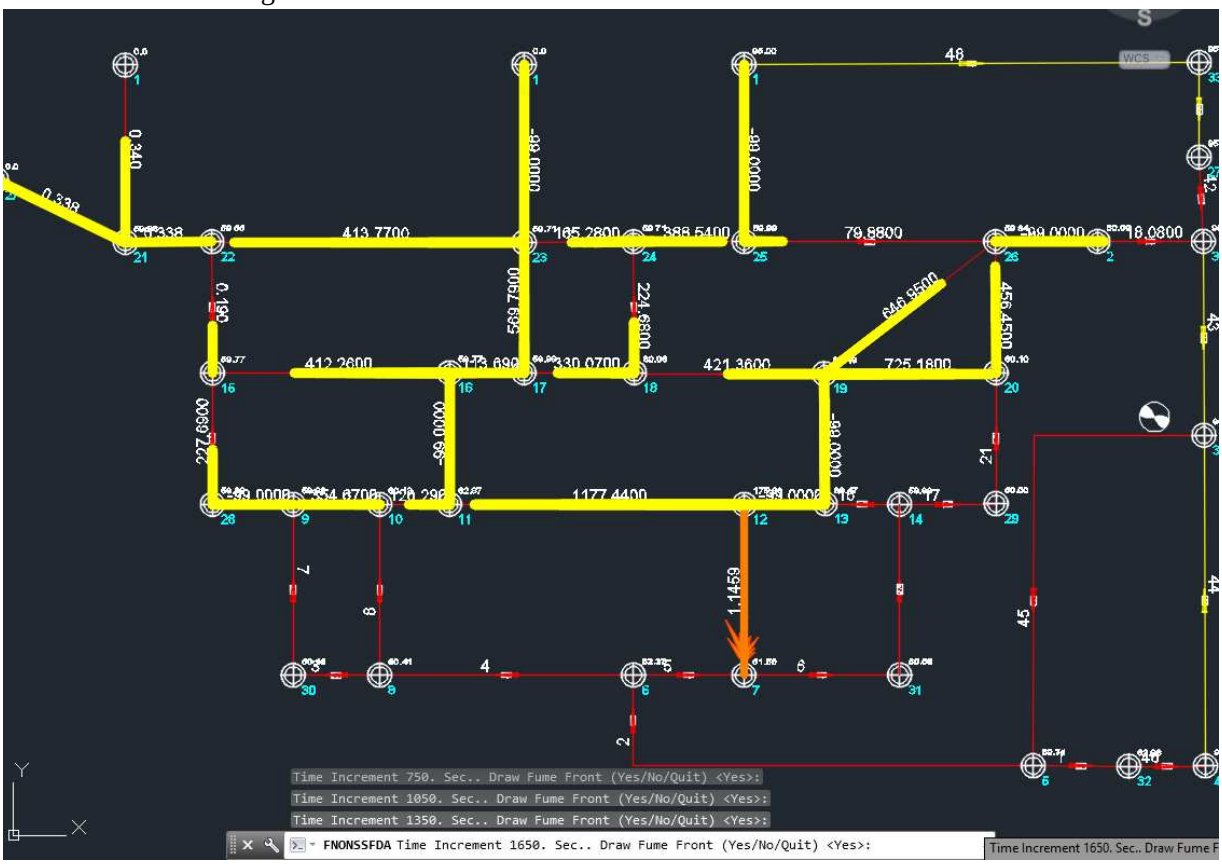
This option does not bring up the usual dialog interface but something similar to it as shown below.



Here we no longer have the time interval drop down as all the time intervals are in the list from beginning to end of the simulation for the fume data totals. You can draw a selected airway and post the parameter at that time interval or if you are in a hurry you can Draw All Airways which will end up mostly just showing the last state for parameters and where the fume front ended up. The parameters are not colored by range and the color and line weight here are just the color and line weight you want to see the fume front drawn in. The real, so called animated, portion of this interface comes when you click the button Close and Draw All by Individual or TimeSteps. The TimeSteps is going to be much quicker than the individual of course. Clicking this button for fire_example3 you are ask on the command prompt:

Time Increment 150 Sec. Draw Fume Front (Yes/No/Quit) <Yes>:

Since Yes is the default you can just press Enter to watch the fume front progress and get something that looks like the following at 1650 Seconds.



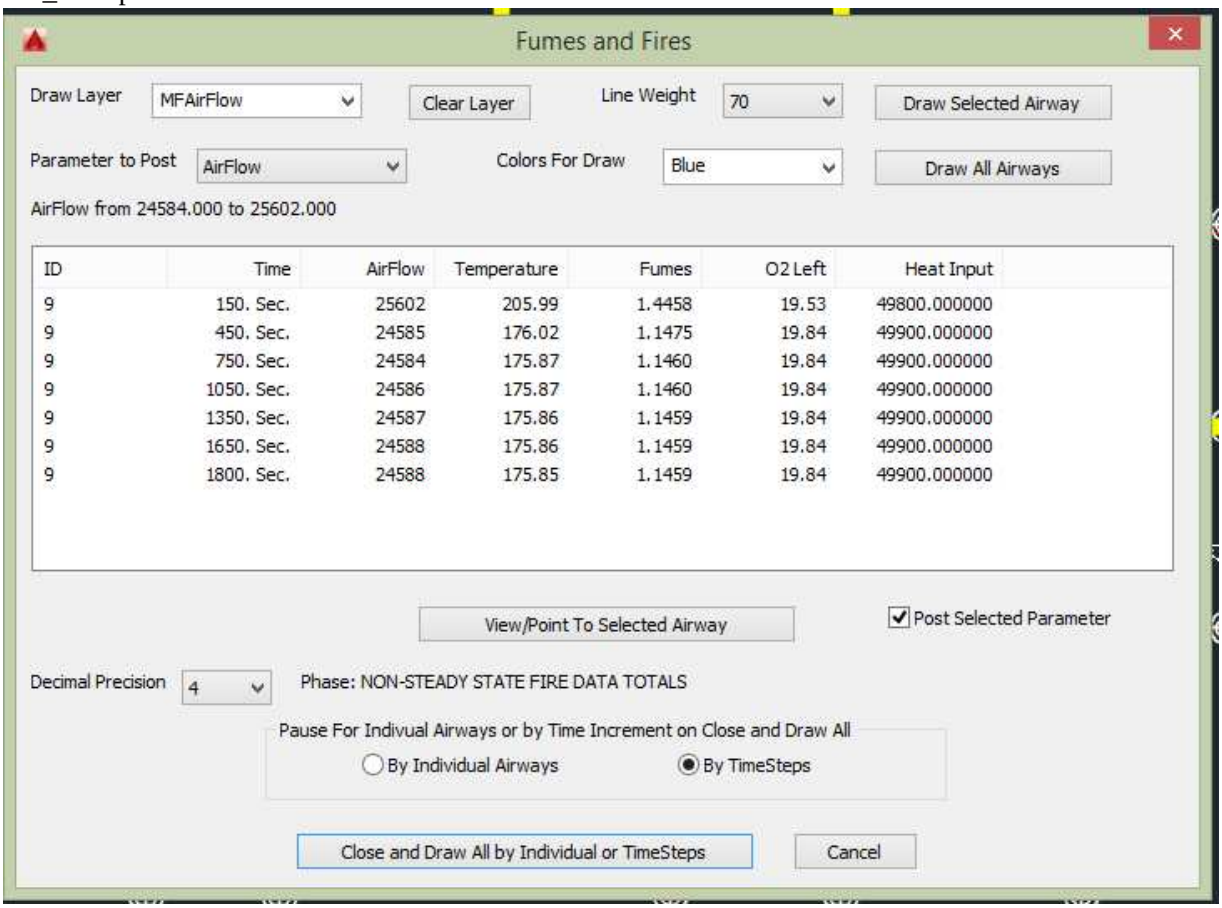
After it draws the fume front it will post the parameters and again ask your permission on the AutoCAD command prompt, such as:

Parameter Fumes Time Increment 150 Sec.. Post Numbers (Yes/No/Quit) <Yes>:

This way you can stop the process at any point in time of coloring or posting. If you tell it to Quit during coloring you still have to tell it to quit for posting the parameters so that it is easy to do one or the other or both at any time you choose. Key here is to watch the command prompt and what it is asking as well as watching the fume front progress.

→Non-Steady State Fire Data

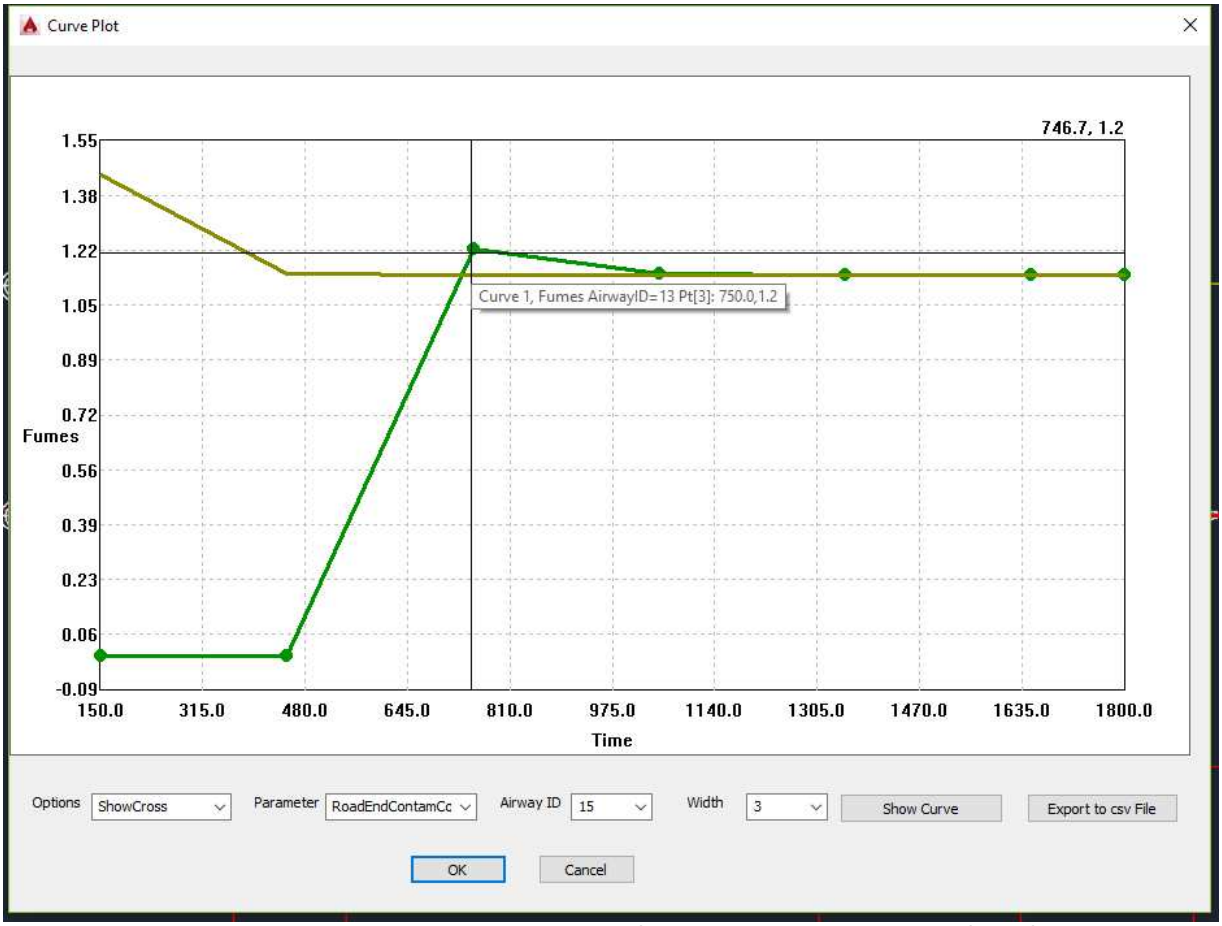
This option brings up the same dialog as Non-Steady Fume Data Totals Animated. But its main purpose is to let you see what is going on in the fire branch or branches for all the time intervals. The animated part may be useful but the parameters are more to focus on here. Below is what it looks like for fire_example3.



→Non-Steady State Curve Plot

This option brings up a dialog box that will allow you to plot any parameter versus time for one or more airway ID's. You can plot more than one parameter as well but the results maybe so different that you just get a straight line at the top or bottom for that airway parameter combo. The data comes from the output file under NON-STEADY STATE SIMULATION OUTPUT and the Time At in the MFIRE Verbose output file, under the title of "TEMP. AND CONCENTRA. AT AIRWAY ENDS, HEADLOSS IN AIRWAYS" in the non-steady output. The parameters are:

AirFlow = AIRFLOW
DeltaQ = DELTA Q
HeadLosss = HEADLOSS
AverageTemperature = AVE. T
RoadEndTemperatuer = T AT END
RoadEndCH4Conc. = CH4
RoadEnfContamConc = FUMES



The above plot shows Fumes (RoadEndContamConc.) for airway 15 and airway 13 from fire_example3. Hovering over the point shows the curve number, parameter, AirwayID, and the x,y coordinates. The options dropdown has a few little features to manipulate the curves shown. The most handy is ShowMark and ShowCross, and perhaps the zooming in and out. The width dropdown determines the thickness of the curve line and size of the marks (points) shown. Once you select the Parameter, AirwayID and Width you can click Show Curve to show it in the graph above. If you click Export to csv File the program will export the data that corresponds to the CURRENT parameter and AirwayID selected not what is shown in the graph unless of course the graph is the same airwayID and parameter you have selected when you click the export button.

→Quasi-EQ Basic

This option brings up the same interface only this time it is from the output file section labelled BASIC DATA FOR AIRWAYS IN THE NETWORK and shows the OUTPUT OF THE QUASI-EQUILIBRIUM SIMULATION PART. This information differs from the Steady-State Basic in that it is the information about the network when it has reached the Quasi-Equilibrium state.

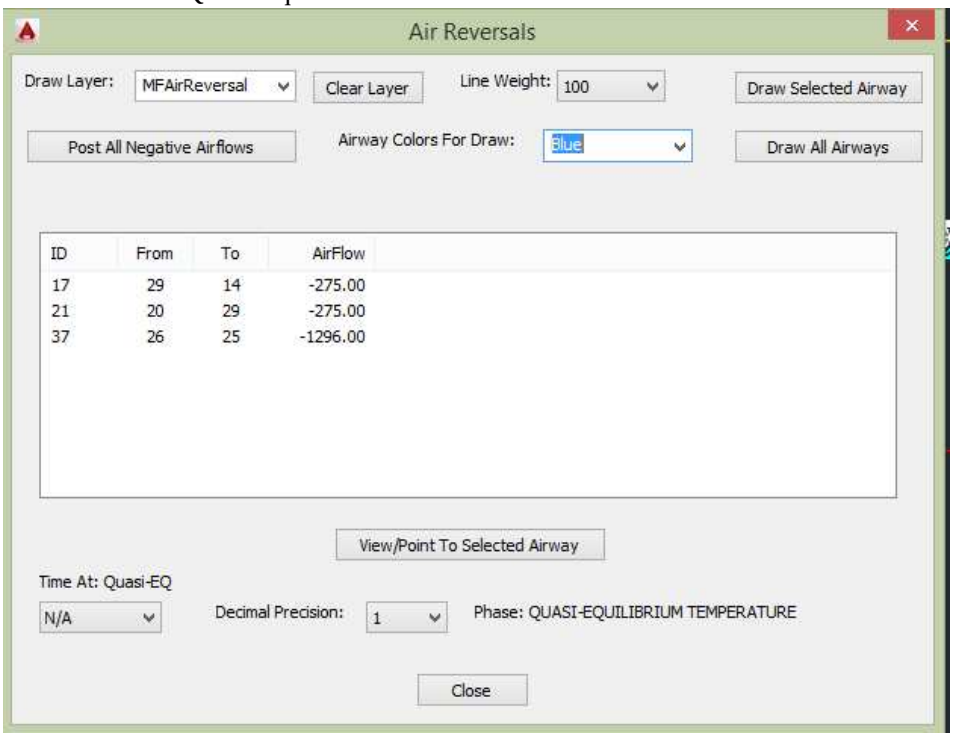
→Quasi-EQ Temperature

This option brings up the same interface only this time it is from the output file section labelled TEMP. AND CONCENTRA. AT AIRWAY ENDS, HEADLOSS IN AIRWAYS and shows the OUTPUT OF THE QUASI-EQUILIBRIUM SIMULATION PART. This information differs from the Steady-State

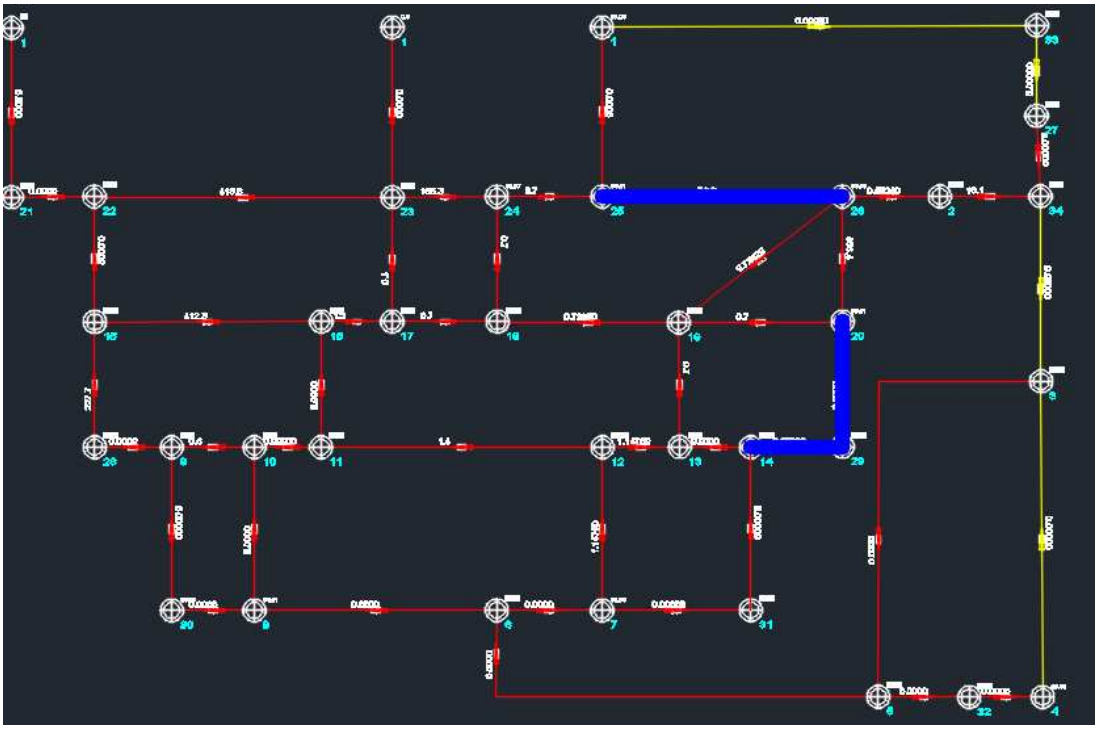
Temperature in that it is the information about the network when it has reached the Quasi-Equilibrium state.

→Quasi-EQ Temperature Reversals

This option brings up the same interface as Steady State and Non-Steady State reversals only this time it is from OUTPUT OF THE QUASI-EQUILIBRIUM SIMULATION PART. This information differs from the Steady-State Temperature Reversals in that it is the information about the network when it has reached the Quasi-Equilibrium state. For fire_example3 it looks like the following. There are more reversals in the Quasi-Equilibrium State.

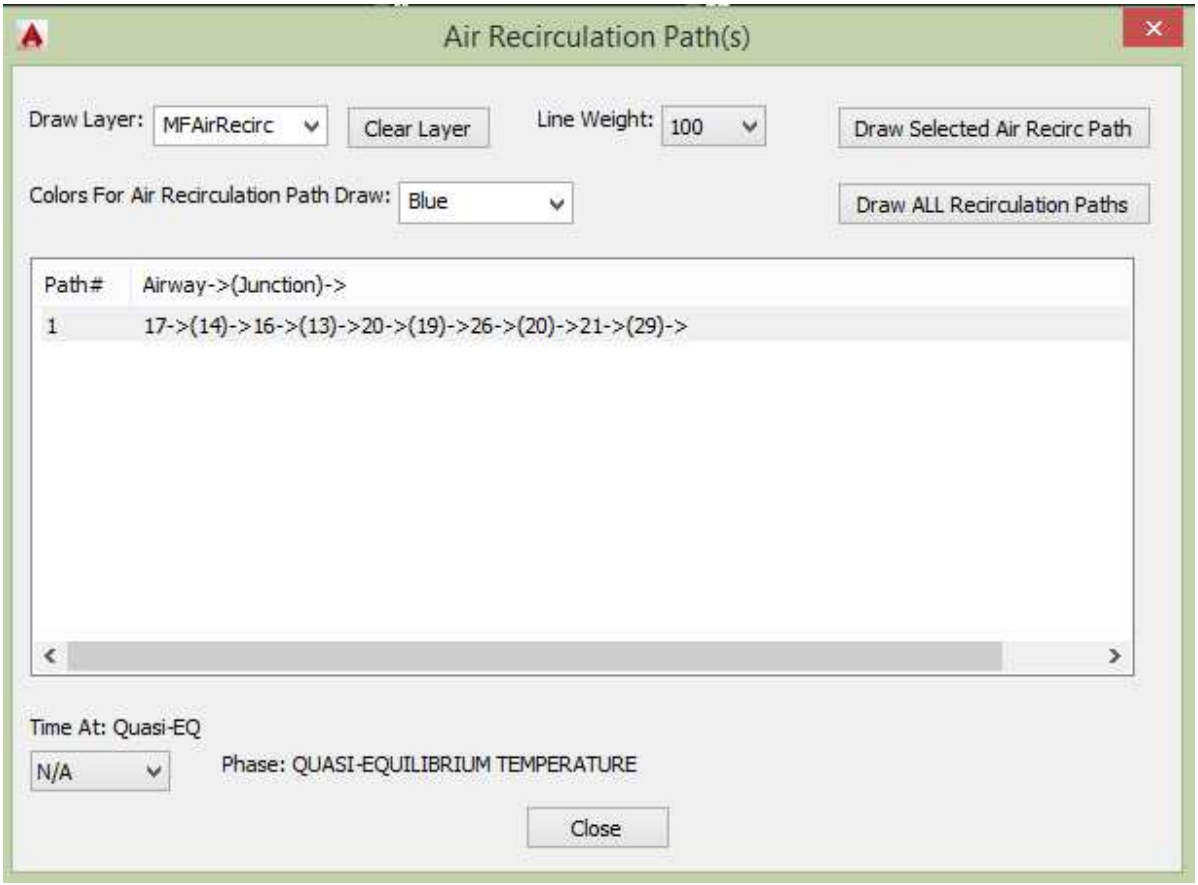


If we Draw All Airways the map should look like the below. Don't forget to post the negative Airflows, especially if you want to reverse them with either Output→Reverse Selected Neg. Branches or Modify Branch Parameters.

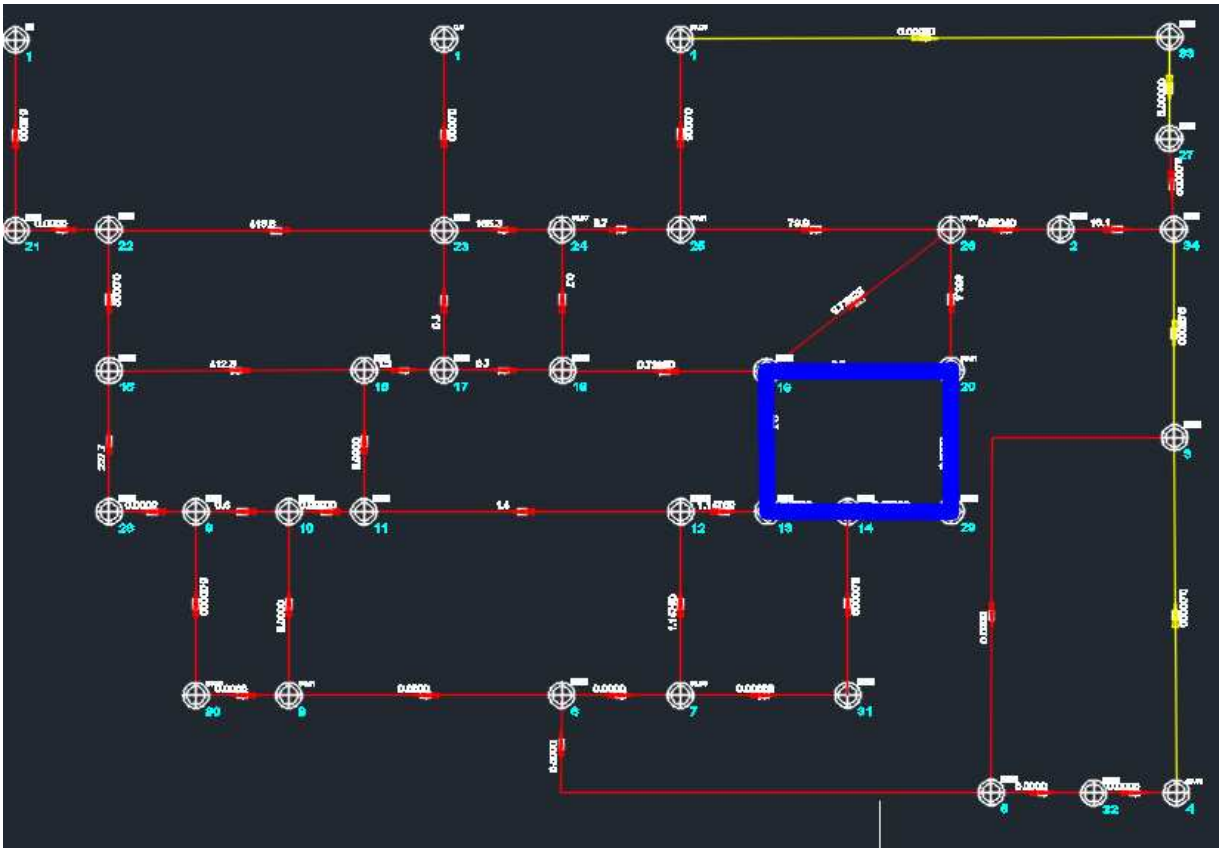


→Quasi-EQ Temperature Recirculation

This is the same interface as for all the other recirculation path menu options, however, we saved showing and discussing it in detail until here since the fire_example3 only has a recirculation path for the Quasi-Equilibrium portion. This option reads from the RECIRCULATION PATH section of the Quasi-Equilibrium part of the program. Likewise the Steady State Temperature Recirculation and Non-Steady State Recirculation read from the same RECIRCULATION PATH sections of their locations in the report. Below is what it looks like for fire_example3 and you can see there is only one recirculation path had there been more we could draw them individually with different line weights and colors or all at once with the same line weight and color.



So in the above as always we have a layer that it is going to draw the 3D polyline(s) on. The default is MFAirRecirc. However, you could type in any layer name you want, just be sure to remember the layer name you type in. This is true for all the dialogs that have a layer drop down. I have set the Line Weight to 100 and the color to Blue. Now if you select it from the list and hit Draw Selected Air Recirc Path you should get the drawing colored up as follows. If you had more than one recirculation path you could use different Line Weights and colors for different parts of the mine. I suspect that in most cases you would just want to select Draw ALL Recirculation Paths, which would have given the same result without having to select from the list first.

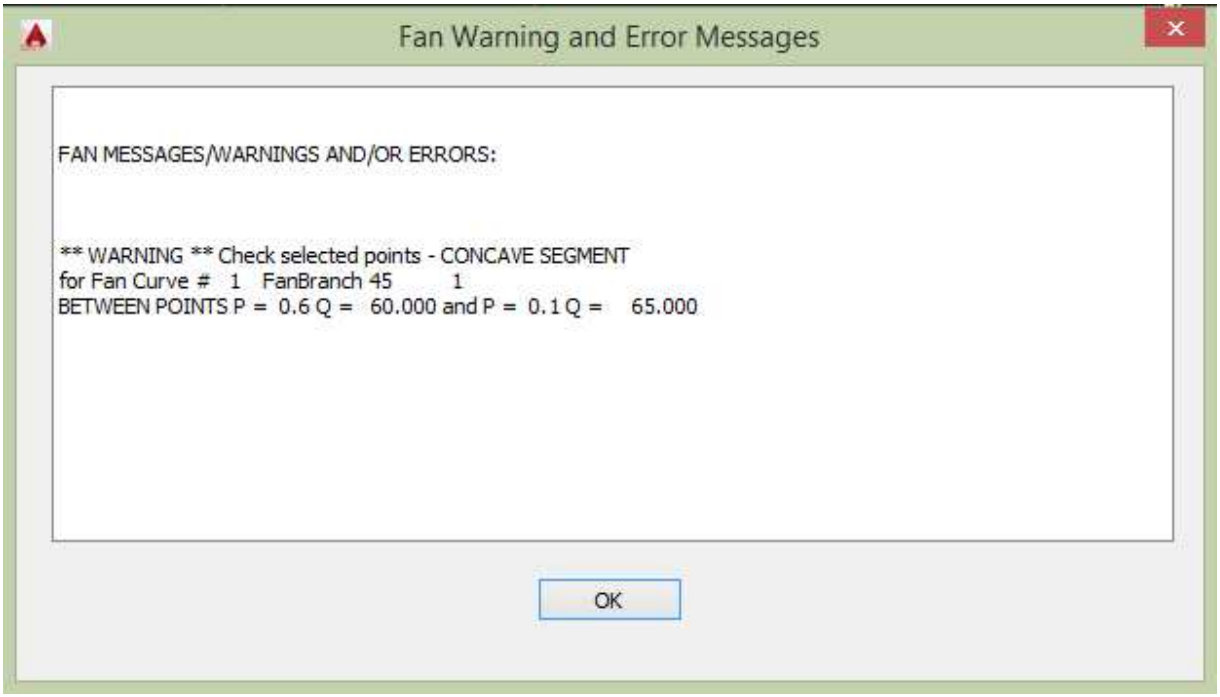


→Quasi-EQ Critical Conditions

This is the critical conditions taken from the Quasi-Equilibrium portion of the output file. Again we have the same standard MineFire Results and Viewing dialog box with the 3 tabs across the top (Airways, Junctions and Options/Colors). The difference this time is that the Airways and Junctions list *only* list the Airways and Junctions in which there are Critical Conditions. See also Steady State Critical Conditions and Non-Steady State Critical Conditions.

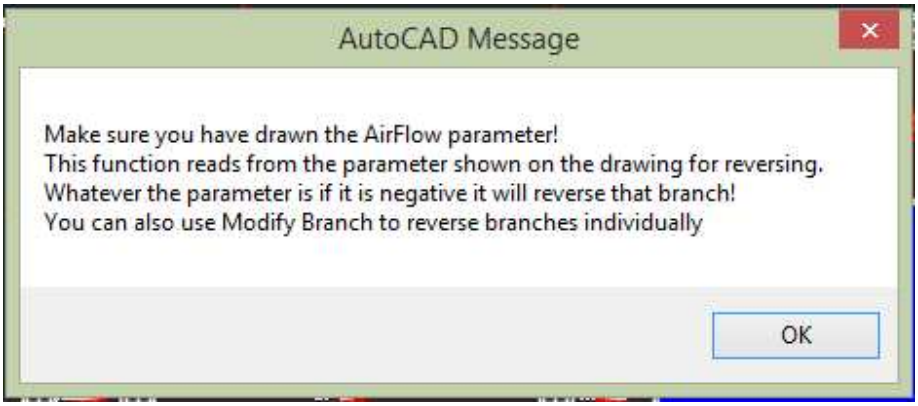
→Fan Warning/Error Messages

This option does a check on the fan curves that are entered into the system to detect any linear or concave segments and reports them if there are. It only does this check after you press Run MFIRE 3.0 from the Run MineFire dialog box. So it is not valid to select this option before you run the program. If it finds a problem you should likely attempt to correct it, however, some warnings may not be a problem as is the case for fire_example3 which has a concave segment (as shown below). It tells you first the type of problem then the fan curve number and name of the fan curve where the problem exist and finally the two points where the problem exist.



→Reverse Selected Negative Branches

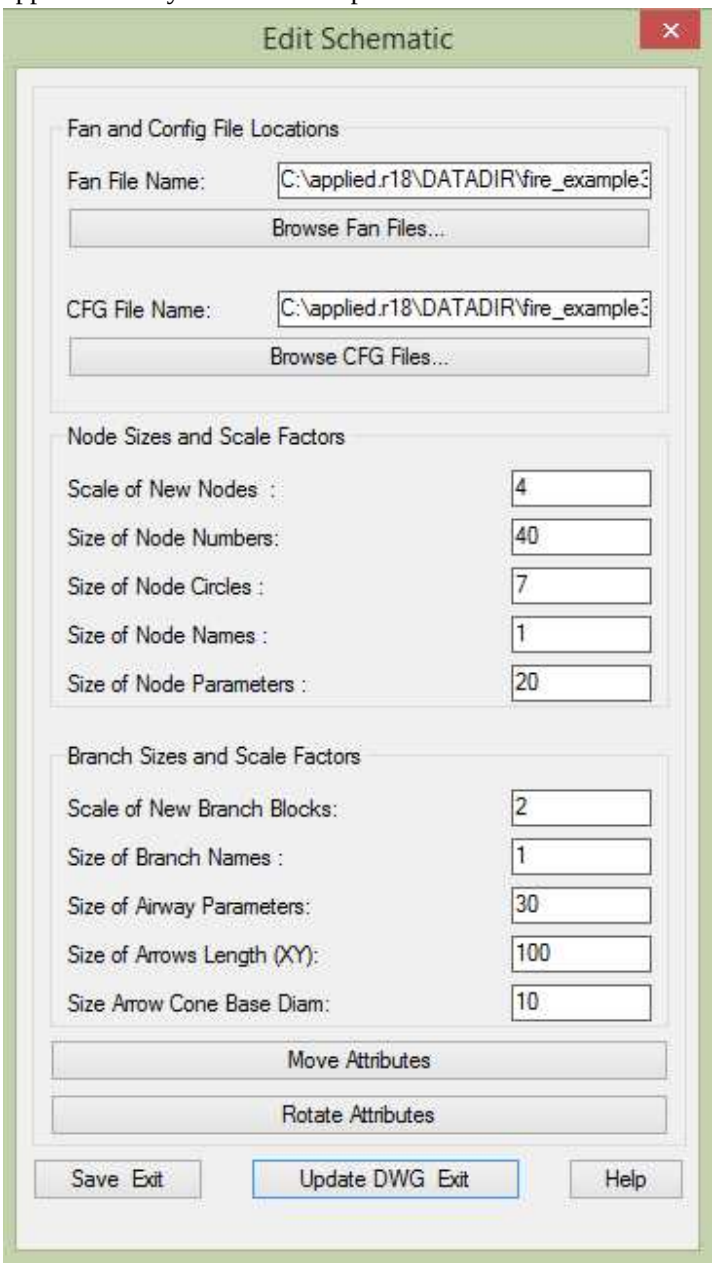
This option is intended to let you draw a closed polygon around airways/branches that have a negative airflow and reverse them so that the next time you run the program you will not get negative airflows. As such, it likely only applies to any reversed branches under the Steady State Network that persist throughout the simulation. However, as it gets the information from the drawing if you have posted some other parameters (i.e.DeltaQ) it will reverse all branches that have a negative value posted above the branch attribute block. When you select this option you will get a warning as follows describing what was said above.



Once you click OK, if you have not drawn the Airflow parameter for the phase you want to reverse, then just press Esc when it ask you to Create a four sided polygon which contains the branches to reverse: Pick Lower Left Corner. Otherwise draw a polygon around the branches you want to reverse and close it by pressing Enter. You will get a dialog telling you how many branches were reversed.

→Edit Schematic

This option has been mentioned throughout the course of this manual and may be useful at several different stages in the design from start to finish and everywhere in between. The following dialog box appears when you select this option.



Edit Schematic is used to change the size of parameters, node numbers, circles, names, arrows. It is also used to set the defaults names for the fan files and the CFG File (Branch code configuration file). The information is stored with the drawing so that you can have one fan file and one CFG file for several different scenarios.

Scale of New Nodes:

This represents the scale factor of the new nodes that have not yet been inserted. It does not affect the size of existing nodes.

Size of Node Numbers:
This represents the size in drawing units of the node numbers. This value is used for existing nodes as well as upcoming nodes.

Size of Node Circles:
This represents the size in drawing units of the node circles. This value is used for existing nodes as well as upcoming nodes.

Size of Node Names:
This represents the size in drawing units of the node name. If the node name is --- it will not be shown. Also it will not be shown if Hide Name is checked in Define or Modify Node.

Size of Node Parameters
This represents the size, in drawing units, of the parameters. The parameter is selected in the MineFire Results and Viewing dialogs. This value is used for existing parameters as well as upcoming parameters.

Scale of New Branch Blocks:
This represents the scale factor of the new branch blocks that have not yet been inserted. It does not affect the size of existing branch attribute blocks.

Size of Branch Names:
This represents the size, in drawing units, of the branch name. Unlike node names the branch names will be shown even if they are ---. The branch names will not be shown if Hide Name is checked in Define or Modify Branch.

Size of Airway Parameters:
This represents the size, in drawing units, of the airway parameters. The parameter is selected from the MineFire Results and Viewing dialogs. This value is used for existing parameters as well as upcoming parameters.

Size of Arrows Length (XY):
This represents the size, in drawing units, of the arrows/cones. This is in the XY direction. This value is used for existing arrows as well as upcoming arrows.

Size Arrow/cone Base Diam:
This represents the size, in drawing units, of the base or circle of the arrows/cones. This value is used for existing arrows as well as upcoming arrows.

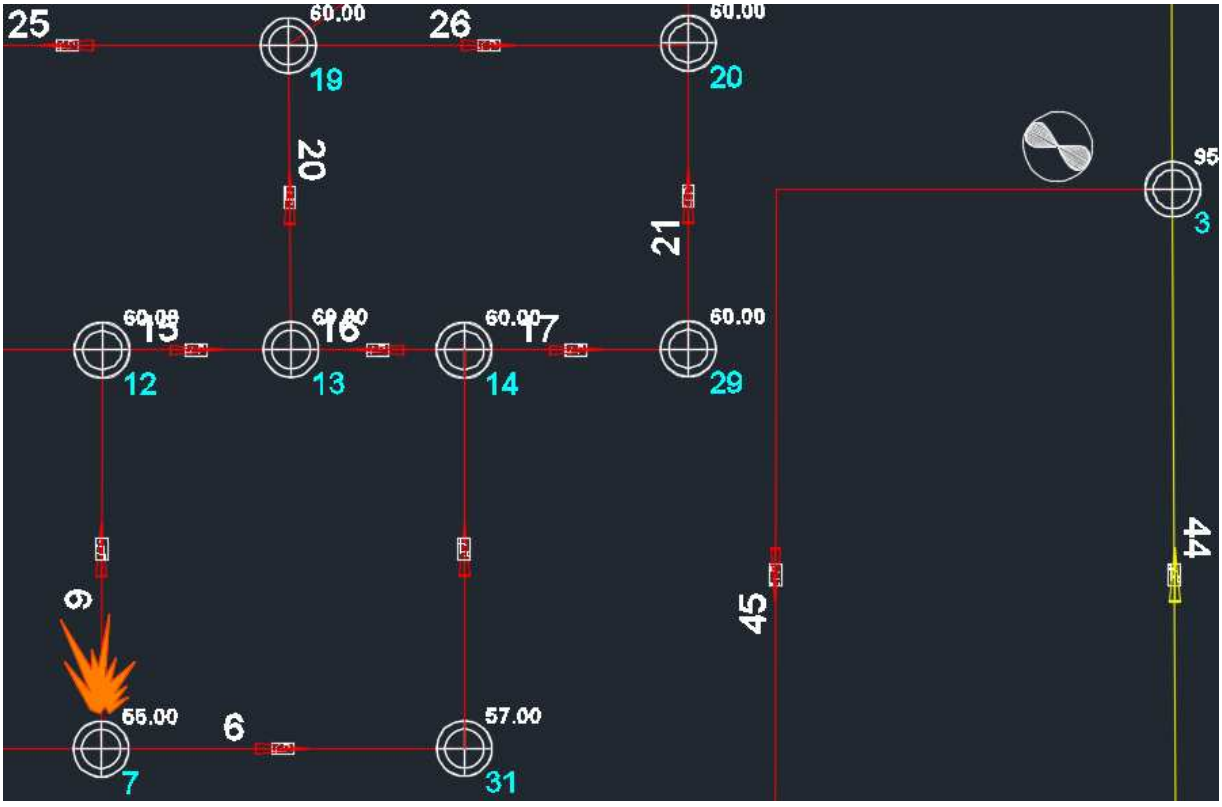
The **Move Attribute** button allows you to select a parameter, node number or name and move it.

The **Rotate Attribute** button allows you to select a parameter, node number or name and rotate it.

Symbols Menu

→Fire Symbol

Select this option if you want to insert a fire symbol in your drawing. When you have selected this option, you will see a set of cross hairs on the screen and will be prompted for the insertion point of the Fire Symbol. Move the cross hairs to the point where you want the center of the Fire Symbol. Then scale and rotate it as you wish. You can also scale and rotate it after it has been inserted. Typically you would insert this somewhere on the branch/airway that has the fire source either in the Time Table or where you intend to place the fires in real-time or a fire branch. You can insert a fire symbol at any time as longer as you are not running any of the other menu options, which goes for inserting anything from this menu. The below screenshot shows the section of fire_example3 with a Fire Symbol on branch 9 and Fan Symbol on branch 45.



→Title Block

Select this menu if you wish to insert a title block in your drawing. After you select this option, you will see a set of cross hairs on the screen and will be prompted for the insertion point of the title block. Move the cross hairs to the point where you want the upper left corner of the title block and click the left mouse button. After you enter the insertion point of the title block, a series of seven prompts will request the scale, angle, drawing name, number, designer, location, scale, etc. For each prompt, you can either accept the default by pressing the ENTER key or type new information. The title block is inserted is inserted with the size specified for the x and y scale factors. This size may be completely out of proportion to the size of your drawing. It may be so small that you can hardly see it, or it may be so large that it overwrites much of your drawing. If it is the wrong size, you may scale it after it is inserted. The title block is a MineVent symbol which means that you may scale it, move it, etc. as a complete entity. If you need to change the block text you can use the AutoCAD ddatte command. You can use Edit Schematic to move

or rotate the block attribute text if needed. You can also explode the block and then manually edit the text. However, after you have exploded the symbol, you can no longer treat it as a single entity.

→North Arrow

Select this menu item if you wish to insert a North Arrow symbol in your drawing. When you have selected this menu, you will see a set of cross hairs on the screen and will be prompted for the insertion point of the North Arrow. Move the cross hairs to the point where you want the bottom of the North Arrow. Next you will be prompted for the scale and the rotation angle of the North Arrow. Enter the number of degrees of rotation to orient the arrow in the correct direction for your drawing. Zero degrees gives a vertical arrow. Positive degrees rotate the arrow counter-clockwise from the vertical, negative degrees rotate it clockwise. For example if the north direction of your map was directly toward the upper right corner, you could enter a rotation angle of either -45 or 315.

The North Arrow block is inserted with the size specified for the x and y scale factors. This size may be completely out of proportion to the your drawing. It may be so small that you can hardly see it, or it may be so large that it overwrites much of your drawing. If it is the wrong size, you may scale it after it is inserted.

The north arrow is a MineVent symbol which means that you may scale it, move it, etc. as a complete entity. If you need to change the block text you can use the AutoCAD ddatte command. You can use Edit Schematic to move or rotate the block attribute text if needed. You can also explode the block and then manually edit the text. However, after you have exploded the symbol, you can no longer treat it as a single entity.

→Fan

This option will prompt you for the location (center of fan symbol) and outside diameter of the fan symbol. Key the diameter but use the cursor to locate the symbol.

→Regulator

This option will prompt you for the location (center of regulator symbol) and outside diameter of the regulator symbol. Key the diameter but use the cursor to locate the symbol.

→Stopping

This option will prompt you for the insertion point location (center of stopping symbol) , x and y scale factors and rotation angle.

→Door

This option will prompt you for the insertion point location (center of door symbol) , x and y scale factors and rotation angle.

→Drop Door

This option will prompt you for the insertion point location (center of drop door symbol) , x and y scale factors and rotation angle.

→Overcast

This option will prompt you for the insertion point location (center of overcast symbol) , x and y scale factors and rotation angle.

→Check

This option will prompt you for the insertion point location (center of check symbol) , x and y scale factors and rotation angle.

→Scale Block

This option allows you to change the map scale symbol. You will be prompted for the X and Y axis scale factors and the angle through which to rotate the symbol. Next you enter the four dimensions of the scale legend. The dimensions are multiplied by the scale factors to create the symbol.

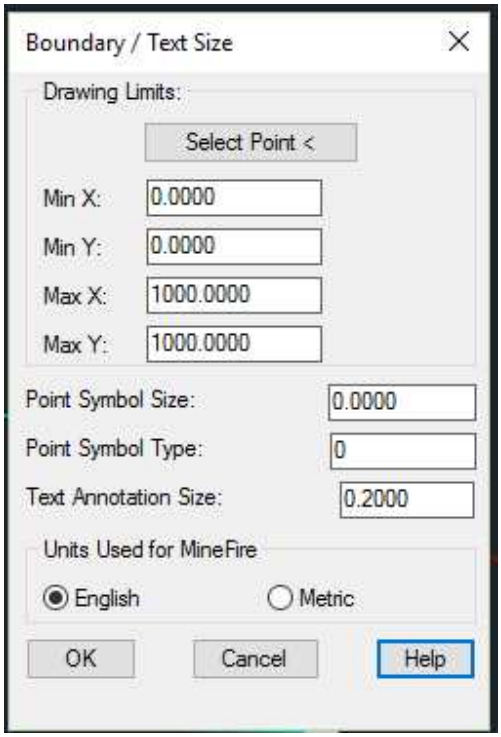
→Escapeways

This option will prompt you for the location (center of escapeway symbol) and outside diameter of the escapeway symbol. Key the diameter but use the cursor to locate the symbol.

Setup Menu for Metric, English, and Import Export Feature

→Drawing Configuration

You should run this option when you start a new drawing or when the drawing defaults need to be changed. The following dialog box allows you to change the drawing limits, point and text sizes and whether to use imperial or metric units. To change the size of all existing text, nodes and branch arrows, use Edit Schematic.



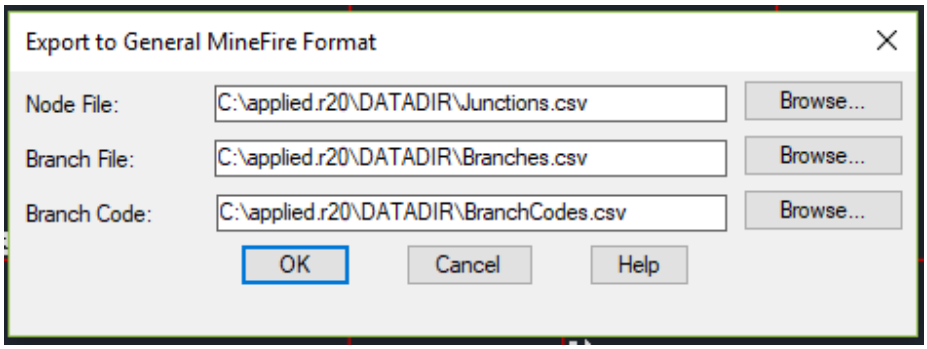
This option also lets you choose what units you would like to work in for new or existing drawings. **It is very important that the units you are working in are the same as what is selected in this dialog box. Units are NOT drawing dependent. So if you are working on a drawing in English then switch to a metric drawing you need to run this option and select the appropriate Units for the drawing in question.**

Important Note: The Verbose output (Output→Show Output File (NotePad) will always show the units in Imperial but everything else under the Output menu should be in the units specified above.

→Export General MineFire and Import General MineFire

These options allow you to Import data from any other program using the general format and saving it as a comma separated file (.csv). Most can use Microsoft Excel or Access. You can use the Export feature and load the files into Excel to see the format. You can also make changes in the file and re-import it to update and existing drawing and even add branches. The most important things for importing into a new drawing are the start node, end node and resistance. However, you would want to get the Type and Code correct and match them up with the other file. You may need to create some sort of look up table to do this. But if you have large drawings you want to Import into MineFire the import feature could save you considerable time. All the information should be in the exported file except that fire branches do not update the fire data so those branches need to be re-entered manually.

Below is the Export Dialog. The Import is identical except for the heading. It is okay to not have a Branch Code file. But nodes and branches are required.

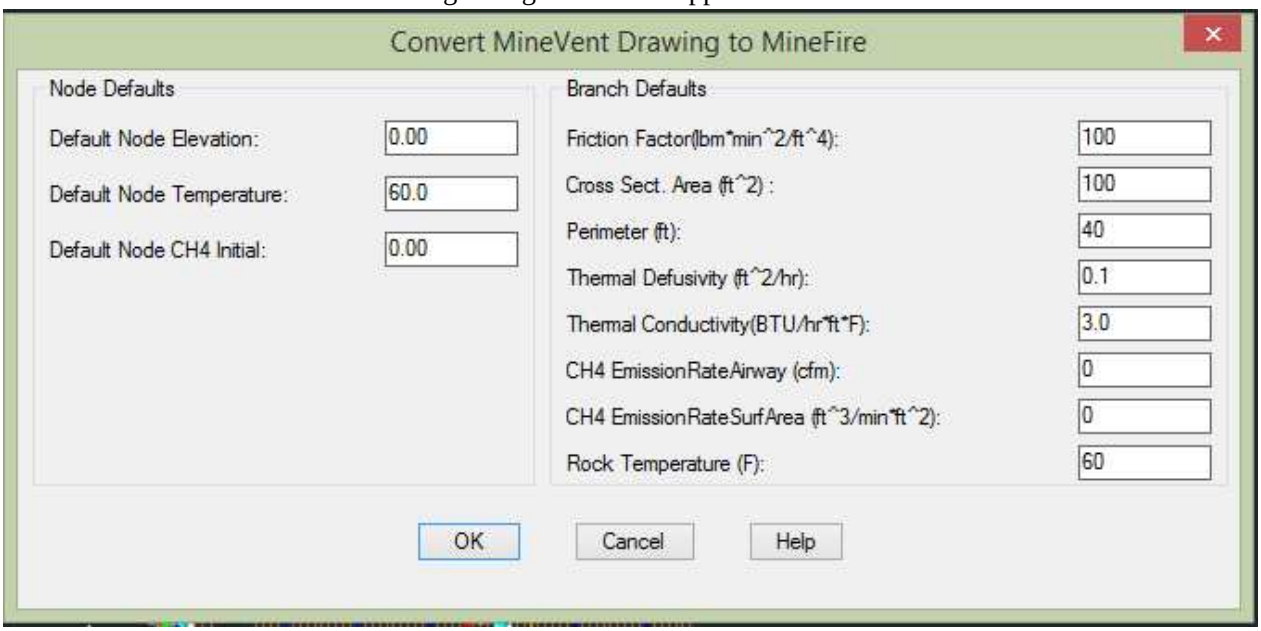


File Menu

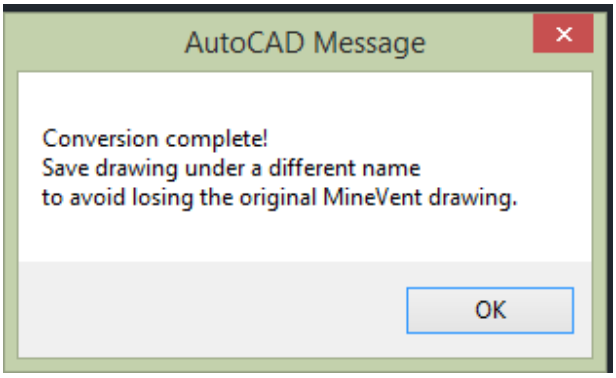
The only thing special here is Convert this MineVent File to MineFire. The rest are just useful AutoCAD functions and allow you to go back to the ICAMPS Startup menu.

→Convert this MineVent File to MineFire

Open a MineVent file then select the MineFire Menu to get to this option. The first thing this option will do is tell you that the MineFire defaults have not been set because MineFire uses a different defaults brand for Edit Schematic than does MineVent. Click OK and you will be presented with the Edit Schematic dialog where you can set the sizes of Nodes, Node Numbers, Circles etc... The defaults should be sufficient for most real world mines, but if after converting you find that it does not suit your preferences then you could re-run Edit Schematic after the conversion. However, if it is a problem with the Scale of New Nodes or the Scale of New Branch Blocks then you would have to reconvert the drawing by re-opening it and then choosing this menu option again. So best to just set your preferences and click Save Exit. Then the following dialog box should appear.



This dialog is meant to set the default information of branches in MineFire because there is data in MineFire that is not in MineVent. The most troubling of these at this point in the default Node Elevation, because this would not update all the branches and lengths associated with that elevation. As well as any intermediate “bends” in the branch polyline which also have elevations. There may be a fix to this in a future release, but for now you would have to do it manually. MineVent does not use Node elevations so it is likely that 0.0 is a good number to use. Once you click the OK button the conversion process should start. This may take some time and your AutoCAD screen my go black depending on the size of the drawing, so please be patient until you are represented with the following dialog.



This tells you that everything went as planned and to save the drawing under a different name. For safety exit out of AutoCAD and re-open the new drawing or just re-open the new drawing. Some versions of AutoCAD leave the same name even if you do a Save As.

The next thing to do would be to add your fire, probably best to use the Branch Menu→Time Table option. Then copy your MineVent fan file to the same drawing name as your new MineFire drawing (i.e. <new_mfire_dwgname>.fan. The CFG file from MineVent should work fine. However, you should likely make a copy of it as well to avoid conflict by changing it for MineFire.

Appendix A

MFIRE 2.20 Manual Translations for MFIRE 3.0 and ICAMPS

This appendix is to help see the change in variable names from the original MFIRE 2.20 Manual and the variable names used in MFIRE 3.0. The variable names used in MFIRE 3.0 are what were used in creating ICAMPS MineFire as they are more descriptive and less short and cryptic. This should help the user if they want to use some of the examples in the MFIRE 2.20 manual, which is also available for download from www.oa-mining.com along with this manual.

New Parameter Name	Old Parameter Name	Limits	Default
NumOfAirways	NB		None
NumOfFans	NFNUM		None
NumOfFires	INFLOW		None
OmitJunctions	NVPN	'0' or '1'	None
NetworkOnly	NETW	'0' or '1'	None
ReferenceTemperature	TR	>=10 Deg. F; <=110 Deg.F	75 Deg.F Note 1
MaxTemperatureIterations	MADJ	>= 5; <=80	10 Note 1
MaxDynamicIterations	ITN	>=5; <=80	10 Note 1
TemperatureOnly	NTEMP	'0' or '1'	None
TimeIncrement	TINC	>0	15 seconds Note 1
TimeSpan	SPAN	>0	None
OutputDetail	IOUT	1 = Normal report 0 = Brief report -1 = Detailed report --2 = More detailed report	None
TimeIntervalForReport	TOUT		
TransientState	CONCT	'0' or '1'	
ReferenceDensity	DR		Note 2

Note 1: In the current version of MFIRE, an error message is displayed if these parameters are omitted or outside the acceptable limits. Insert the appropriate default value into the input file to replicate the default value substitution of the previous version of MFIRE.

Note 2: Default calculated based on reference temperature. If specified, the ratio of the specified reference density to the calculated reference density must be less than 0.7.

Airway Cards

New Parameter Name	Old Parameter Name	Limits	Default
Number	NO	>=1; <9990	None
StartJunction	JS	>=1; <=999	None

New Parameter Name	Old Parameter Name	Limits	Default
EndJunction	JF	>=1; <=999	None
Type	NWTYP	0 = Normal airway 1 = Fan airway -1 = Fixed quantity airway 10 = Airway contains fire source	None
Resistance	R	> 0	Note 1
FlowRate	Q	>= 1000	Note 2
FrictionFactor	KF	>= 10.0; <= 1000.0	100.0 Note 3
Length	LA	>= 10.0; <= 3000.0	500.0 Note 3
CrossSectionalArea	A	>= 0.1; <= 500.0	100.0 Note3, Note 4
Perimeter	O	>= 0.1; <= 100.0	40.0 Note3, Note 5
ThermalDefusivity	HA	>= 1.0E-5; <= 10.0	0.1 Note 3
ThermalConductivity	HK	>= 1.0E-5; <= 50.0	3.0 Note 3
CH4EmissionRateAirway	CH4V	None	None
CH4EmissionRateSurfArea	CH4PA	None	None
RockTemperature	TROCK	None	None

Note1: If the specified resistance is less than or equal to zero for a normal airway (type = 0), the resistance is calculated based on the Friction Factor, Length, Cross-sectional Area, Perimeter and Reference Density. In fan airways that do not calculate pressure based on a fan curve (fix flow), the resistance represents the pressure in the airway.

Note 2: If the initial airway flow rate is less than 1000, the flow rate is set to 1000.

Note 3: If the value for these parameters is omitted or if the value specified is outside of the acceptable range, the parameter is set to the average default value. If average values are not specified in the network specification, the above default values are used.

Note 4: If a value of less than or equal to 0.01 is specified initially (before default substitution) for the cross-sectional area and the value of the perimeter is within valid limits, the value of the cross-sectional area will be calculated based on the value provided by the perimeter. If the value of the perimeter is not within limits, an error message will be displayed.

Note 5: If a value of less than or equal to 0.01 is specified initially (before default substitution) for the perimeter and the value of the cross-sectional area is within valid limits, the value of the perimeter will be calculated based on the value provided by the cross-sectional area. If the value of the cross-sectional area is not within limits, an error message will be displayed.

Junction Cards

New Parameter Name	Old Parameter Name	Limits	Default
Number	JNO	>=1; <999	None
Temperature	T	>= 35; <= 125	None
Elevation	Z	None	None
CH4InitialConc	CH4C	>= 0; <= 100	None

Fan Card 1

New Parameter Name	Old Parameter Name	Limits	Default
AirwayNo	NOF	Note 1	None
DataCount	MPTS	>= 2; <= 10	None
CurveFittingMethod	NSWT	1 = Least Square 2 = Spline 3 = Auto select	None

Note 1: The airway number specified must exist in the mine structure of the fan data will be flagged as isolated.

Fan Card 2

Air flow data and pressure data occur as paired data items (up to five pairs per card; maximum total of ten data pairs)

New Parameter Name	Old Parameter Name	Limits	Default
AirFlowData	QF	>= 0; Note 1	None
PressureData	PF		

Note 1: Air flow data must be specified in increasing order of magnitude.

Fan Card 3

New Parameter Name	Old Parameter Name	Limits	Default
FanCurveBoundry	NPLOT	1 = Curve is extended following it's gradient at both ends; 2 = Left boundary extended following gradient, right boundary is sent to zero; 3 = Both boundaries are sent to zero	None

Contamination (Fire) Card

New Parameter Name	Old Parameter Name	Limits	Default
AirwayNo	NCENT	Note 1	None
ContamFlowRate	CONT	>= 0	None
ContamConcentration	CONC	>= 0; <= 100	None
HeatInput	HEAT	Note 2; Note 3	None
O2ConcLeavingFire	O2MIN	>= 0	None
ContamPerCuFtO2	SMPO2	>= 0	None
HeatPerCuFtO2	HTPO2	>= 0	None
StandardAirFlow	QCENT	>= 0; Note 2	None
TransitionTime	TPR	>= 1.0E-5	None

Note 1: The airway number specified must exist in the mine structure.

Note 2: The heat input and standard air flow are limited to prevent abnormal temperature output from the fire source.

Note 3: If a negative heat input is specified, the contamination source is treated as a fix capacity cooling station.

Control Card 2

New Parameter Name	Old Parameter Name	Limits	Default
AvgValueDataPresent	NAV	<= 0 – Average value inputs have been omitted >= 1 – Average value inputs present	None

New Parameter Name	Old Parameter Name	Limits	Default
StartJunction	JSTART	Note 1	None
StartJunctionTemperature	TSTART	>= -40; <= 120	None
TimeEquillibrium	TIME	> 0	10.0 Note 2
FumeCriteria	CRITSM	>= 1.0E-4; <= 1.0	0.005 Note 2
CH4Criteria	CRITGS	>= 1.0E-4;; <= 1.0	0.01 Note 2
TemperatureCriteria	CRITHT	>= 1.0E-4; <= 1.0	0.1 Note 2
PressureDropWarningLimit	WRNPR	>= 1.0E-10; <= 0.5	0.01 Note 2
FumeWarningLimit	WRNSM	>= 1.0E-10; <= 1.0	0.05 Note 2
CH4WarningLimit	WRNGS	>= 1.0E-10; <= 10.0	1.0 Note 2
TemperatureLimit	WRNHT	>= 1.0E-10; <= 1000.0	100.0 Note 2

Note 1: The airway number specified must exist in the mine structure.

Note 2: In the current version of MFIRE, an error message is displayed if these parameters are omitted or outside the acceptable limits. Insert the appropriate default value into the input file to replicate the default value substitution of the previous version of MFIRE.

Average Value Card

New Parameter Name	Old Parameter Name	Limits	Default
AverageDiffusivity	HAAVR	>= 1.0E-5; <= 10.0;	0.1 Note 1
AverageConductivity	HKAVR	>= 1.0E-5; <= 50.0	3.0 Note 1
AverageFrictionFactor	KFAVR	>= 10.0; <= 1000.0	100.0 Note 1
AverageLength	LAAVR	>= 10.0; <= 3000.0	500.0 Note 1
AverageCrossSection	AAVR	>= 0.1 <= 500.0	100.0 Note 1
AveragePerimiter	OAVR	>= 0.1 <= 100.0	40.0 Note 1

Note 1: Default values are loaded into these parameters if an out or range input is specified or if the average value inputs are omitted (AvgValueDataPresent <= 0).