Pipe Line Loss Calculator

The Basics

This program is based on the technical paper 'Steady Flow Analysis of Pipe Networks; An Instructional Manual' by Roland W. Jeppson, the book Hydraulic Pipeline Sytems by Bruce E. Larock, Roland W. Jeppson, Gary Z. Watters. Additional information has been drawn from the Crane technical paper 410 for fitting resistance coefficients. The program is intended for non-compressible fluids including slurry flows, even though use in the case of slurries should be with some caution.

The piping systems which can be analysied include simple straight runs of pipe, pipe runs with take off points (consumption points), closed pipe loop(s) with consumption points. The pipe system can be fed by pump(s), tank(s) or just by specifying an input flow. When specifying input flows care needs to be used in that the flows entering the loop(s) need to equal the flows exiting the loop(s).

The Program Interface

Data is entered using a simple graphic interface, NOTE; this interface is more of a line schematic of the pipe system, it is not intended to be a scaled or even a geometrically accurate layout of the pipe system. It is just a series of lines and nodes representing how the pipe is connected.

The Opening Screen:
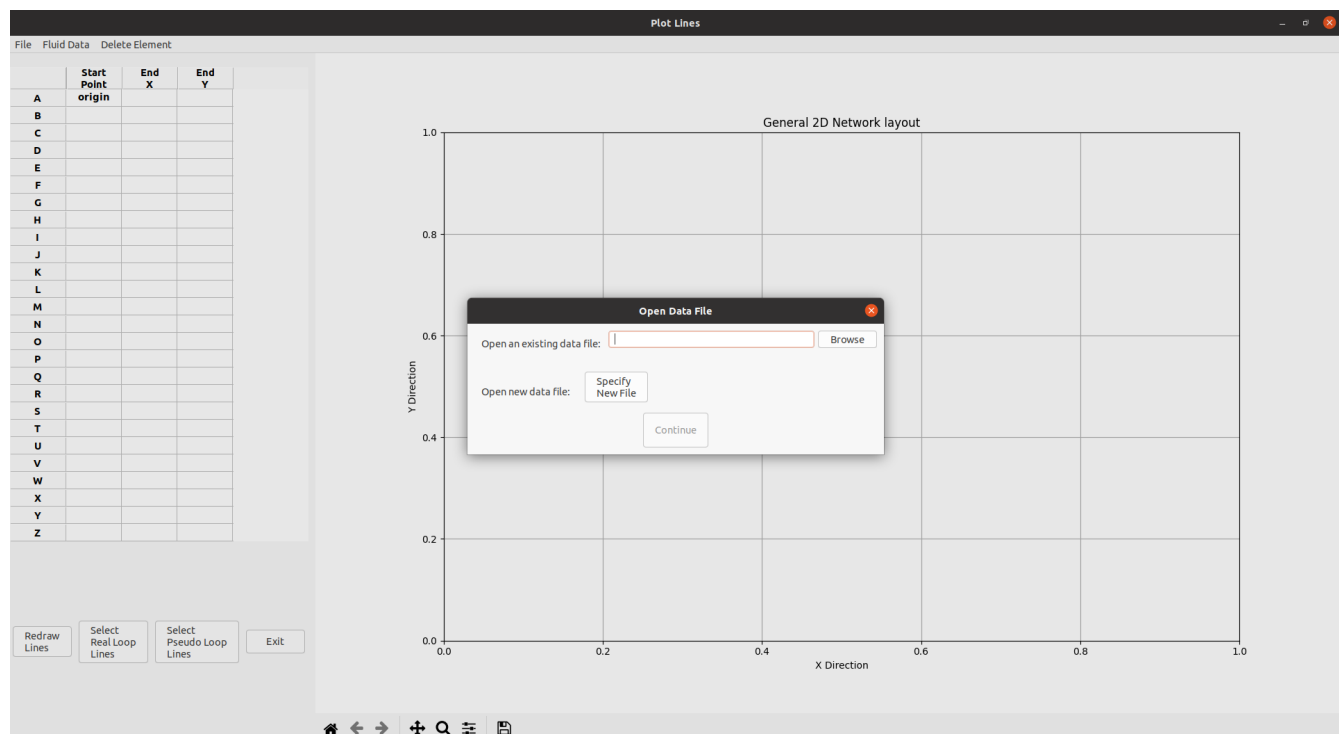
This what you see at start up, likely you don't have a data file to open so click the 'Specify New File' button. The next popup should be your standard save file screen, complete it as you would normally and save the file. The file path and name will now appear in the above 'Open Data File' form click the 'Continue' button. If you have a data file already use the 'Browse' button to locate the file then click 'Continue'. For now I will assume you have created a new file, for two reasons; 1) I am lazy and do not want to do this for two different options and 2) if you already have a data file your beyond this point anyway. You are now at the information entry screen, more on that later.So here we are later.

The Information Entry Screen
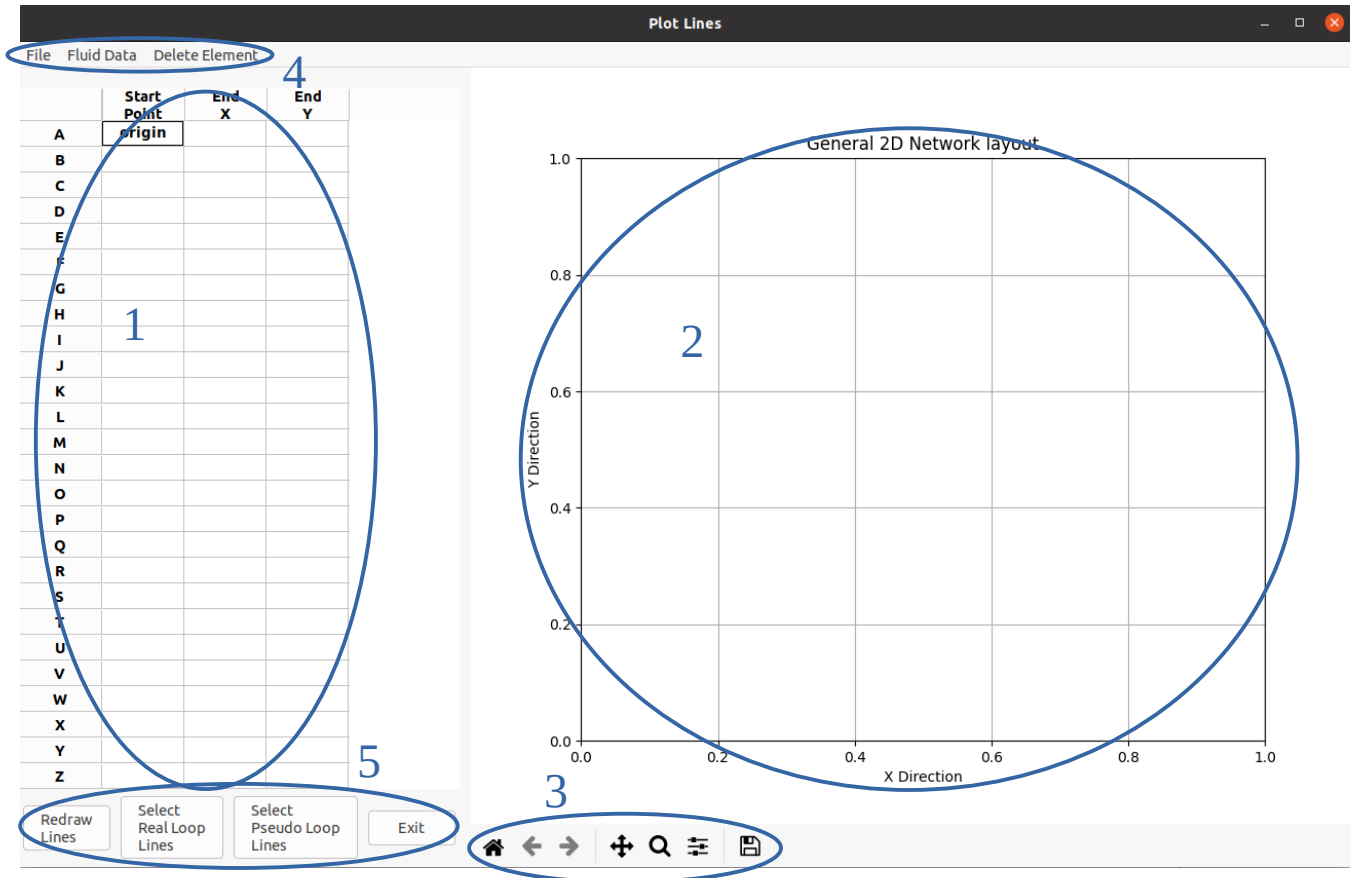

The areas of the screen are:
1 – data entry for the pipe end points
2 – the interactive graphics area
3 – buttons to manipulate the graphics area
4 – menu buttons
5 – command buttons

On the first line after a new point is entered in the grid (area 1) a line will be draw in the graphics (area 2). The line will be labelled with an upper case letter the end point will be labelled with a lower case letter. The line label will be the same as the row label in the grid, the node label will just be the first unused available letter. Again the end points do not and should not represent any scaling of the actual pipe layout, in-fact for ease of reading line and node labels it is recommended that line lengths be keep to $3-5$ units long, both X and Y coordinates are needed. From this point on the first point entered in the grid as a start point must be a node letter (or the word 'origin') which shows in the graphics area.

The X – Y coordinates can be entered either as coordinates or by entering a node letter the line is connection with.An example of data entered is:

	Start Point	End X	End Y
A	origin	4	5
B	a	2	-1
C	origin	b	
D			
E			

Which produces the graphics as:

General 2D Network layout

Node Label

Line Labels

Node Label
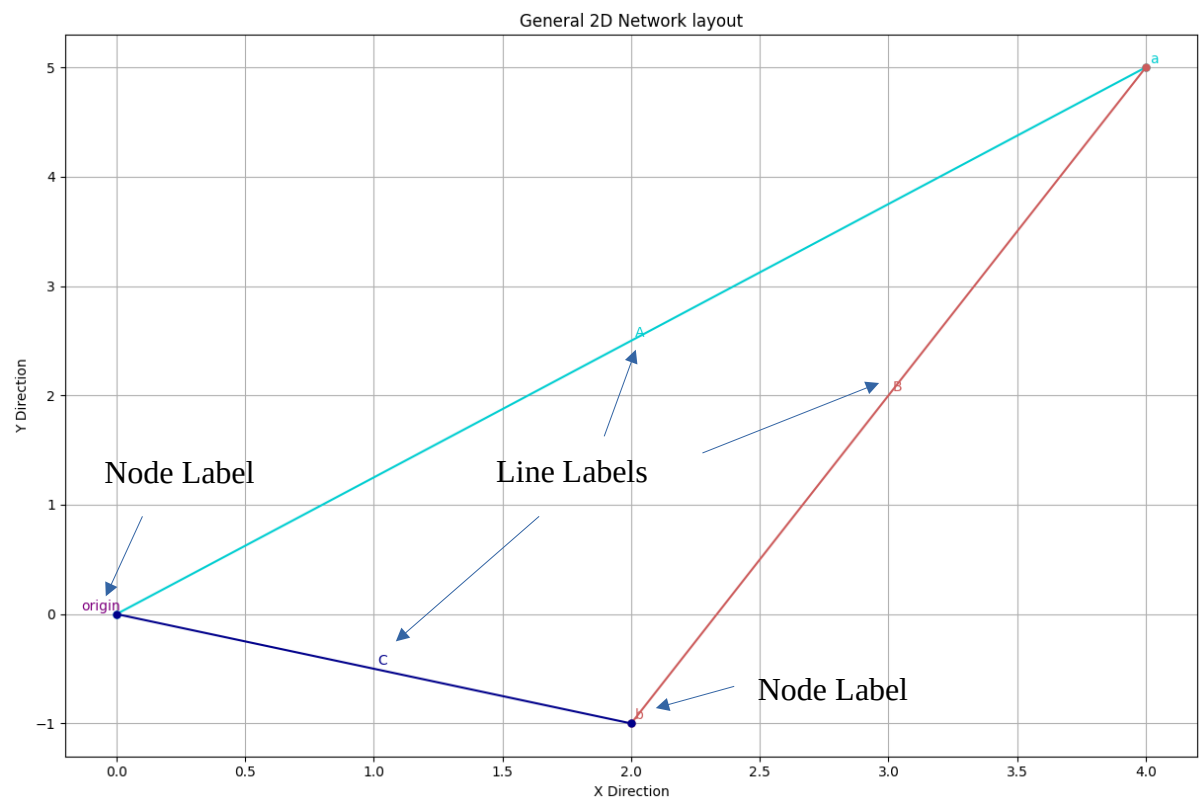
Now that you have a line and node drawn, clicking on the line letter will popup the line input form and it's may pages. Clicking on the node letter will bring up the node entry form. And yes more on these forms will follow later.

Its not later enough, so we will continue to look at the entry form. Area 3 manipulates the graphics area.

The `Home`, 🏠 `Forward` and ➡ `Back` ⬅ buttons

These are akin to a web browser's home, forward and back controls. `Forward` and `Back` are used to navigate back and forth between previously defined views. They have no meaning unless you have already navigated somewhere else using the pan and zoom buttons. This is analogous to trying to click `Back` on your web browser before visiting a new page or `Forward` before you have gone back to a page -- nothing happens. `Home` always takes you to the first, default view of your data. Again, all of these buttons should feel very familiar to any user of a web browser.

The ✛ `Pan/Zoom` button
This   button has two modes: pan and zoom. Click the toolbar button to activate panning and zooming, then put your mouse somewhere over an axes. Press the left mouse button and hold it to pan the figure, dragging it to a new position. When you release it, the data under the point where you pressed will be moved to the point where you released. If you press 'x' or 'y' while panning the motion will be constrained to the x or y axis, respectively. Press the right mouse button to zoom, dragging it to a new position. The x axis will be zoomed in proportionately to the rightward movement and zoomed out proportionately to the leftward movement. The same is true for the y axis and up/down motions. The point under your mouse when you begin the zoom remains stationary, allowing you to zoom in or out around that point as much as you wish. You can use the modifier keys 'x', 'y' or 'CONTROL' to constrain the zoom to the x axis, the y axis, or aspect ratio preserve, respectively.
With polar plots, the pan and zoom functionality behaves differently. The radius axis labels can be dragged using the left mouse button. The radius scale can be zoomed in and out using the right mouse button.

🔍 The `Zoom-to-rectangle` button
Click this toolbar button to activate this mode. Put your mouse somewhere over an axes and press a mouse button. Define a rectangular region by dragging the mouse while holding the button to a new location. When using the left mouse button, the axes view limits will be zoomed to the defined region. When using the right mouse button, the axes view limits will be zoomed out, placing the original axes in the defined region.

The `Subplot-configuration` button

Use this tool to configure the appearance of the subplot: you can stretch or compress the left, right, top, or bottom side of the subplot, or the space between the rows or space between the columns.

The `Save` button

Click this button to launch a file save dialog. You can save the graphics files with the following extensions: `png`, `ps`, `eps`, `svg` and `pdf`.

NOTE: after selecting any one of these buttons they need to be clicked again to deselect them.

The Menu buttons, have nothing to do with food.
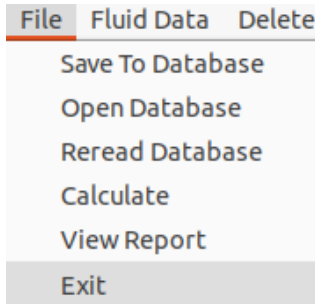
The File menu drop down gives you the option to:
- Save the currently input data to the database
- Open and existing database or create a new database
- Reread the database data, if it was changed and needs to update the screen
- To run the final calculations was all the required information is input
- To view the final calculated results in a PDF report which is generated automatically after the calculations are completed.
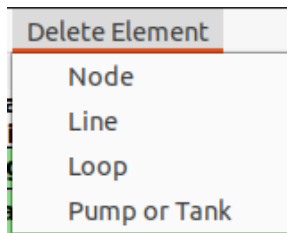
In order to remove certain elements from the graphics area you will need to use the Delete Element drop down menu.
To remove a node you would click the Node menu item this will introduce a pop up warning form, shown below (no waiting for this one). Once you acknowledge the warning you can proceed to delete a node. But be forewarned (oh scary stuff), deleting a node removes all the associated lines and anything associated with those lines. If its a matter that you just don't like the location of a node its easier to modify the coordinates in the grid area.
The same procedure can be followed for deleting Lines, Loops and Pumps or Tanks along with the same ominous warning.
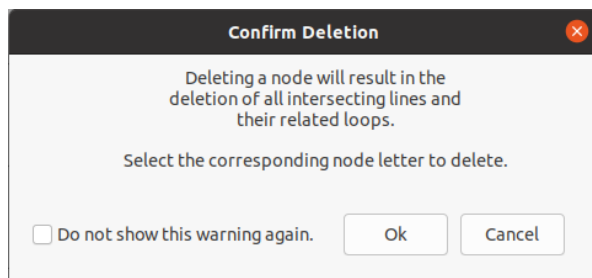
You can stop this warning from appear by checking the 'Do not show this warning again', box, neat eh!
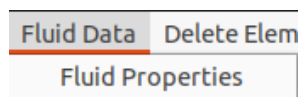
Selecting the Fluid Properties drop down from the Fluid Data menu pops up the fluid entry form. This is an annoying 2 step process but sometime life is like that. But its not as annoying as the pop up which will appear if you click the Calculate button before Saving this screen.

It comes preloaded with water, but you can make up your own homogeneous mix or slurry properties, just like a mad scientist.

	Density	Units	Kenimatic Viscosity	<= OR => Units	Dynamic Viscosity	Units	Concentration % by Wt	Concentration % by Vol
Homogenous Liquid (1)	62.3	lb/ft^3	1	centistokes	0	centipoise	0	100
Homogenous Liquid (2)	0	lb/ft^3	0	centistokes	0	centipoise	0	0
Solids	0	lb/ft^3		Save	Exit		0	

The last area to look at is area 5, the Command Buttons.

Redraw Lines

This button redraws the graphics and changes the line and node colors in the process.

Select Real Loop Lines

Selecting this command will put you into the mode to draw the 'real' or closed loops, used in the energy equations.

Select Pseudo Loop Lines

The pseudo loop button will allow you to select the lines forming a pseudo loop between tanks, pumps or control valves. All of this will, (you have it), be explained later.

Exit

The last button is the exit or close the application button. There are actually 3 ways to close the application, don't ask why there just are. The other two are the little icon in the top right corner of you screen, and the Exit option in the File drop down menu, which I forgot to mention previously.

Define the Pipelines

Well now it's latter enough and I can mention the line and node entry forms, final some of the good

stuff. The line entry form pops up when you select a line label in the graphics, the line selected shows up in the form title. It allows you to enter information specific to the pipe line selected. The opening screen gets general data pipe diameter, inside, pipe length and surface roughness of the pipe material. You can also select if there is a pressure relief valve or a back pressure valve in the line by clicking the appropriate check box.

Click on line label 'A' and enter diameter 4", length 1000' and roughness .0102". then File --> Data Complete.

If you are entering a consumption line, either one which supplies flow to the circuit or removes fluid flow you would leave the form blank.

This is the required additional information needed if you click either the Add Pressure Relief or Back Pressure valve boxes. For a Back Pressure valve you would be entering the Upstream pipe length.

I am sure you noticed that this form actually has 7 different pages; but because I am lazy, as previously stated, we will look at only one other page.

This page like the remaining 5 is for the entry of the fittings which maybe on the pipe line, you merely enter the number of fittings present. In some cases such as reducers or welded elbows you will need to specify more details, but I have confidence you'll get it.

If the fittings form is greyed out and does not allow data entry it is because the 'General Pipe Information' has not been entered. You must have a pipe diameter and length entered to proceed with the fittings selection.

NOTE: Once all the data is entered into all the various pages you **must** click the 'File' menu button and select 'Data Complete' to properly save your changes. You will also note a change to the grid area of the main form. The line that was selected will now be high-lighted green indicating the information has been entered and completed for this line.
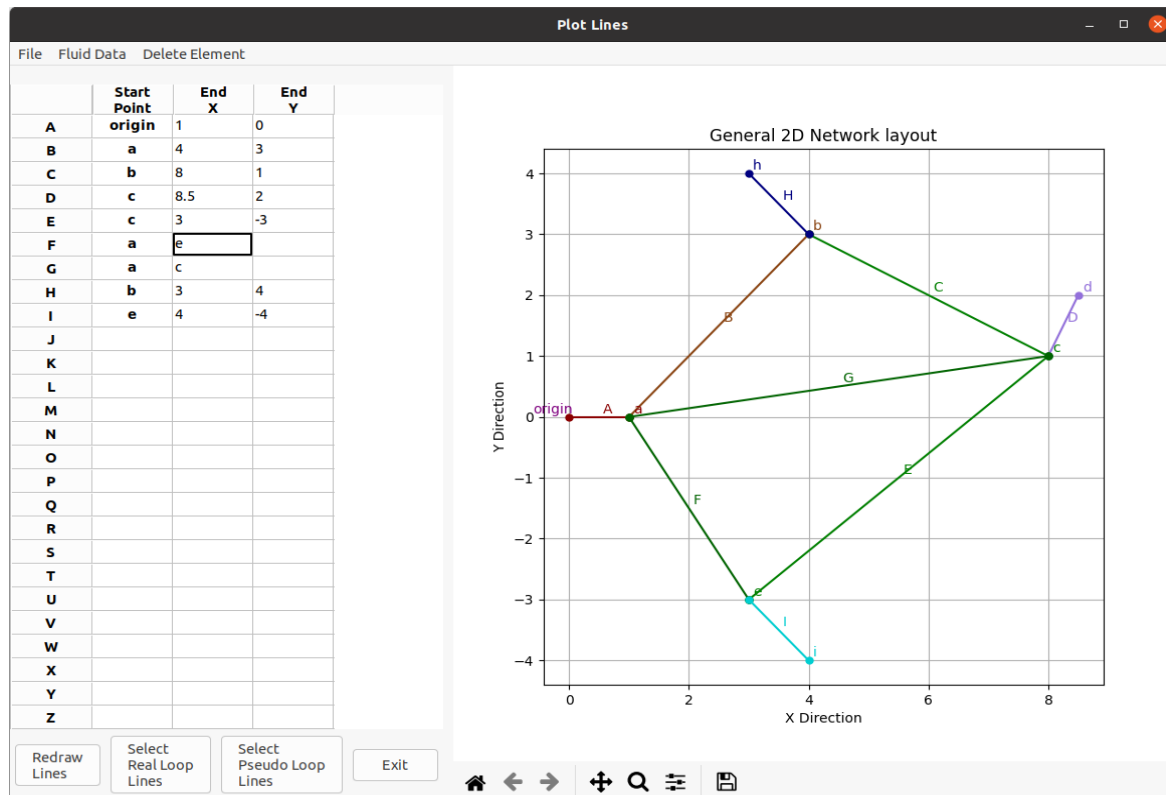
Now the node information entry form. This gets a little trickier, first I added some more points to my graphics to help explain, it now look like this:

Define the Intersection Nodes

Lines 'A', 'D','I' and 'H' at corners of the graphics will be used as consumption lines, 'A' & 'I' will supply a fluid flow and 'D' & 'H' will consume or drain the fluid flow. So lets first select node 'a' to get the node entry form.

At the top of the Node form is the node and its coordinates, next is the elevation of the node, you can select the units in the drop down box on the right.

The selection below lets you specify the direction of flow into the node along the line specified. Don't worry if you guess wrong for the flow direction, the program will correct it, during calculaion. If it is a consumption flow check the check box then input the flow and specify the units of flow.

The lower box area is where you indicate if the node is just a junction point, a pump/tank supply point or a reservoir tank. The next figure shows how this form can be completed and the changes it makes to the graphics.

Node "a [1.0, 0.0]" Flow Information.

Node Elevation: [0] feet ▼

Flow Into Node		Flow Out Of Node	External Consumption Flow		Units for Flow
○	line "B"	● Not Yet Specified	☐		US GPM ▼
○	line "G"	● Not Yet Specified	☐		US GPM ▼
●	line "A"	○ Not Yet Specified	☑	1.1	ft^3/s ▼
○	line "F"	● Not Yet Specified	☐		US GPM ▼

Specify type of node point;
- ● Intersection of Multiple Lines As List Above
- ○ Supply Tank / Reservoir
- ○ Centrifugal Pump and Supply Tank

[Save]   [Exit]

OK so a lot happened once the data was entered into the node form and the Save button was clicked.

In the grid the cells containing the coordinates for point 'a' and any cell containing the node letter 'a' where highlighted green. Since the consumption line 'A' has been specified as such it was highlighted in yellow.
The lines in the graphics have had arrow heads attached to indicate the direction of flow.

Plot Lines

File   Fluid Data   Delete Element

	Start Point	End X	End Y
A	origin	1	0
B	a	4	3
C	b	8	1
D	c	8.5	2
E	c	3	-3
F	a	e	
G	a	c	
H	b	3	4
I	e	4	-4
J			
K			
L			
M			
N			
O			
P			
Q			
R			
S			
T			
U			
V			
W			
X			
Y			
Z			

[Redraw Lines]   [Select Real Loop Lines]   [Select Pseudo Loop Lines]   [Exit]

General 2D Network layout
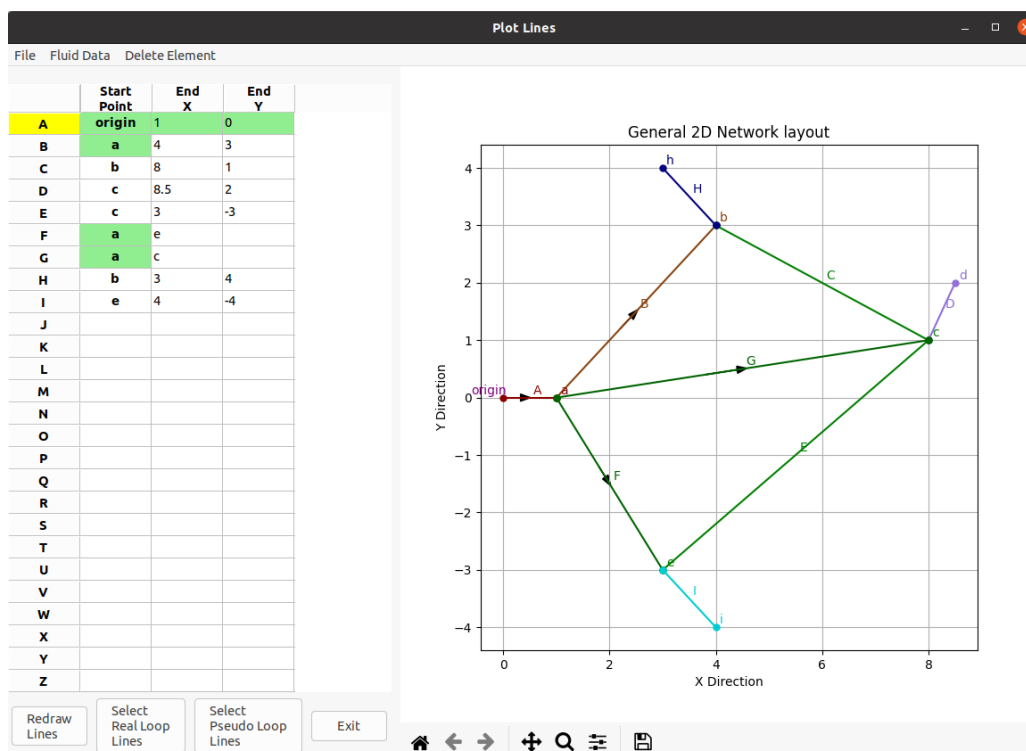
Lets repeat the process for point 'b'. The node form for the 'b' is completed as shown.
You will notice when the form is opened that line 'A' flow has already been "Specified at node 'a' ".
You need only specify those which are 'Not Yet Specified'.

Node "b [4.0, 3.0]" Flow Information.

Node Elevation: [0] feet ▼

Flow Into Node		Flow Out Of Node	External Consumption Flow	Units for Flow
●	line "B"	○ Specified at node "a"	☐ [ ]	US GPM ▼
●	line "C"	○ Not Yet Specified	☐ [ ]	US GPM ▼
○	line "H"	● Not Yet Specified	☑ [1.0]	ft^3/s ▼

Specify type of node point;
● Intersection of Multiple Lines As List Above
○ Supply Tank / Reservoir
○ Centrifugal Pump and Supply Tank

[Save]   [Exit]

Once 'Saved' the grid section will look like this and the graphics will update to show arrow heads on lines 'H' and 'C'. Remember line 'H' is a consumption and is high-lighted yellow.

	Start Point	End X	End Y
A	origin	1	0
B	a	4	3
C	b	8	1
D	c	8.5	2
E	c	3	-3
F	a	e	
G	a	c	
H	b	3	4
I	e	4	-4

	Start Point	End X	End Y
A	origin	1	0
B	a	4	3
C	b	8	1
D	c	8.5	2
E	c	3	-3
F	a	e	
G	a	c	
H	b	3	4
I	e	4	-4

Lets finish up the last nodes actually the last node flow into node 'c' is set at 1.0 ft^3/s and flow out of node 'e' is set at 1.0 ft^3/s. The last node need only be opened and confirmed, this step is there to confirm the node elevation. The grid section should now look like this;

And the graphics area should look like this

The cells for lines 'B','C', 'E', 'F', and 'G' are not high-lighted yet as the piping has not been defined. As each is defined it will be high-lighted. Go ahead and enter the line data as follows:
Simple click on each line label and eneter the data as shown then remember to click the menu item "File" --> "Data Complete" to save it. The grid should now look like this:

Line	Pipe ID	Length	Absolute Roughness
B	6"	1000'	.02"
C	6"	800'	.02"
E	6"	2000'	.02"
F	6"	1000'	.02"
G	1"	1500'	.02"

	Start Point	End X	End Y
A	origin	1	0
B	a	4	3
C	b	8	1
D	c	8.5	2
E	c	3	-3
F	a	e	
G	a	c	
H	b	3	4
I	e	4	-4
I			

Pipe & Fittings for Line G   — □ ⊗

File

General Pipe Information	Ball, Butterfly, Plug, Globe Valves	Diaphragm, Gate Valves	Check Valves	Fittings	Welded Elbows	Entry Exit Losses

Pipe Material	mm	inch
Plastics	0.0015	0.00006
A53 A106	0.0610	0.00240
Concrete Smooth	0.0400	0.00157
Concrete Rough	2.00	0.07874
Copper Tube	0.6100	0.02402
Drawn Tube	0.0015	0.00006
Galvanized	0.1500	0.00591
Stainless Steel	0.0020	0.00008
Rubber Lined	0.0100	0.039

Diameter: 1   inches ▼

Length: 1500   feet ▼

Absolute Roughness: .02   inches ▼

This table shows suggested design values for the Absolute or Specific roughness, for various pipe materials. These values are sited from;

- The Hydraulic Institute, Engineering Data Book.
- Various vendor data compiled by SAIC, 1998
- F.M. White, Fluid Mechanics, 7th edition

Add a Pressure Relief Valve ☐
or Back Pressure Valve ☐

Define the Closed Loops (or energy equations)

The last step in defining our model is to define the closed energy loops, in this example there are 2 potential loops. In this example there are 5 line flows to be determined, there are 4 junction nodes which have already been defined; however, of the 4 node equations only 3 are independent. The 4 junction nodes provide 3 independent equations and the as yet undefined loops provide 2 more equations.

Let's define the loop defined by lines 'E', 'F' and 'G'.
First initiate the process by clicking the   button, it will change to

Select
Real Loop
Lines

Cancel
Real Loop
Selection
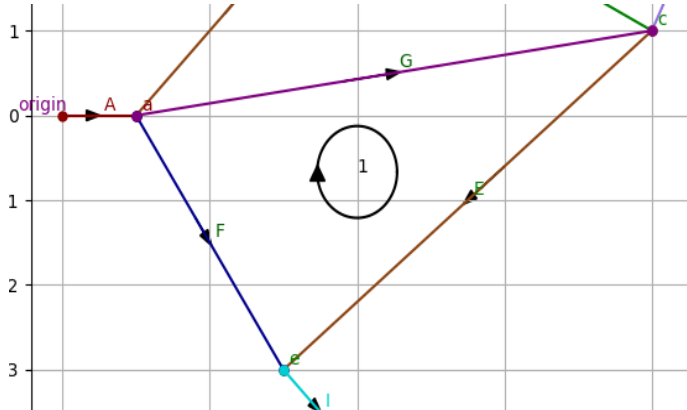
 Now all you need do is select the lines by clicking on the line label for each line. When the line is selected it's colour well change to black. Once the last or loop closing line is selected, the lines will revert to coloured and a circular arrow will appear with a number defining the loop. The order in which the lines are selected does not matter the program will arrange them into an order which is clockwise. Do the same for loop 'B', 'C', 'G'. You could have selected the lines 'B', 'C', 'F' and 'E' instead but it is recommended that you keep the loops small and repeat as few lines as possible.

If all has gone well your screen should look something like this:
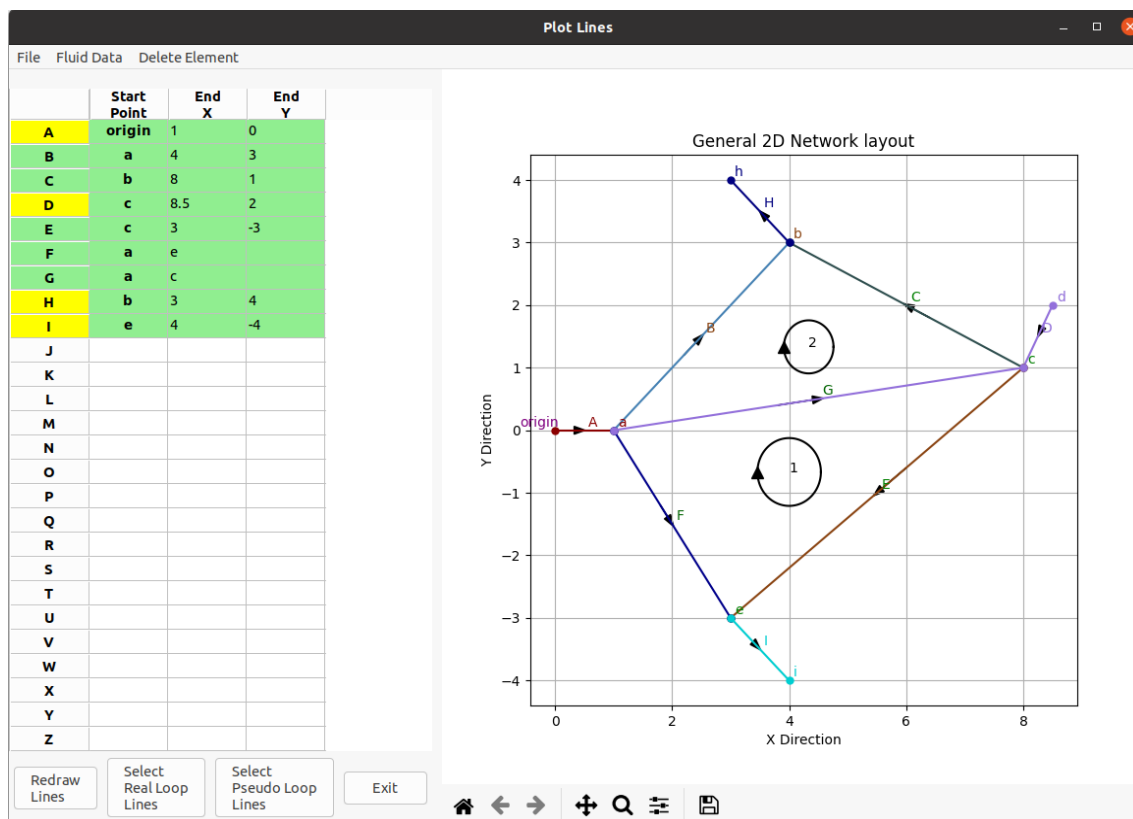
the cell defining the node coordinates should be shaded green
2. all the lines which have undefined flows and which have had the diameter, length and roughness defined will be shaded green
3. any line which is just a consumption point will be shaded yellow

You form should now show 2 loops as a minimum and the cells shaded as above. Now you can click the menu item File --> Calculate. But first you should use the File -->Save to Database menu item.
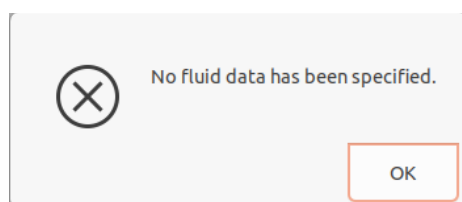
Now because the fluid is not defined you just get:

So click the menu item Fluid Data --> Fluid Properties, just accept the default for water by clicking the Save button.

Now go ahead and click the File --> Calculate menu option. One of two things could happen, the good thing is you will see this screen:

Data Completed

Line Label	Flow ft^3/s
B	0.46047932239818934
C	0.5395206776018107
E	0.46047323550011443
F	0.6395267644998858
G	-6.086898075015992e-06

Save Results   Exit

The printed information includes;
- pump data
- control valve data
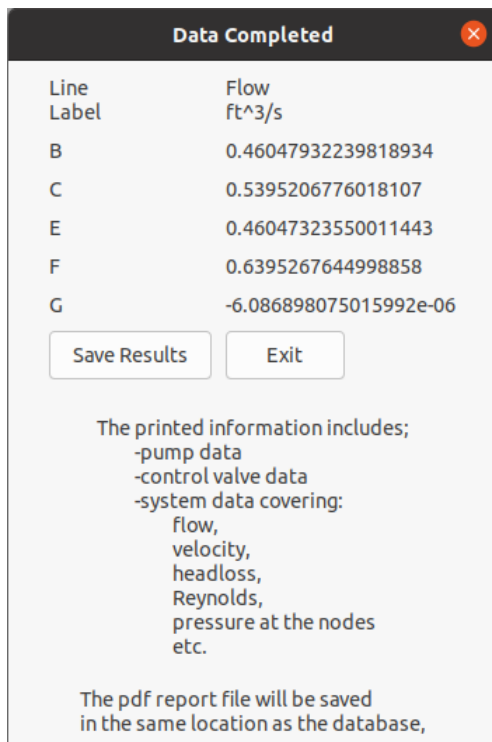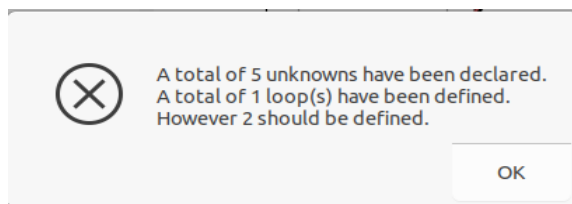- system data covering:
  - flow,
  - velocity,
  - headloss,
  - Reynolds,
  - pressure at the nodes
  - etc.

The pdf report file will be saved in the same location as the database,

It shows the line labels and the line flows in ft^3/s. You can elect to save the results or exit the form and make changes to the graphics.

The flow for line 'G' shows as a negative this means that the direction shown in the graphics is wrong and is reversed. If you go to the graphics and select either node 'a' or 'c' and reverse the direction of flow for line 'G', then rerun the calculation the flows will be the same except line 'G' will no longer be negative.

If you didn't get the above screen, the second thing which may happen is not too bad and can usually be fixed.
 (NOTE: for this example one of the 2 loops was deleted to force this message).
It looks like this;

A total of 5 unknowns have been declared.
A total of 1 loop(s) have been defined.
However 2 should be defined.

OK

It means that you have not fully defined your data. The program tries to take a best guess at the problem. In this case it says you should have 2 loop defined, but only 1 has been shown.

But sometimes it just indicates a warning;
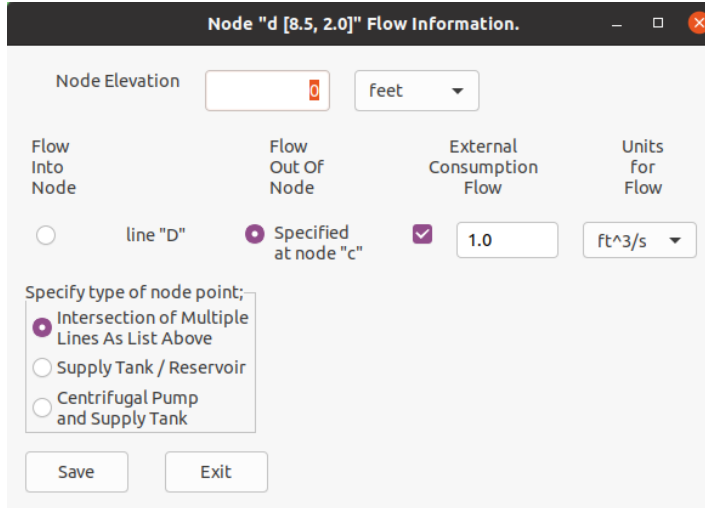
'Unable to iterate network to a solution, total number of iterations completed = #.
 Based on presented information system cannot be solved.'

This is not good and it means there is something wrong with the data entered i.e. you screwed up. You will need to go through the forms/data and try to determine what went so terribly, terribly wrong. Sometimes this can be repaired by using the menu item File --> Reread Database.

Ok on a high point we can now continue on with other features of the program, pumps and tanks.

Entering Pumps and Tanks

You define a pump or tank using the node form at a specified node, the only stipulation is that the node must be at the end of a single line, i.e. it cannot be at a node with multiple intersecting lines. Lets start with a simple tank and locate it at node 'd'. Open the node definition screen by clicking on the new node 'd'. Define the items as shown;
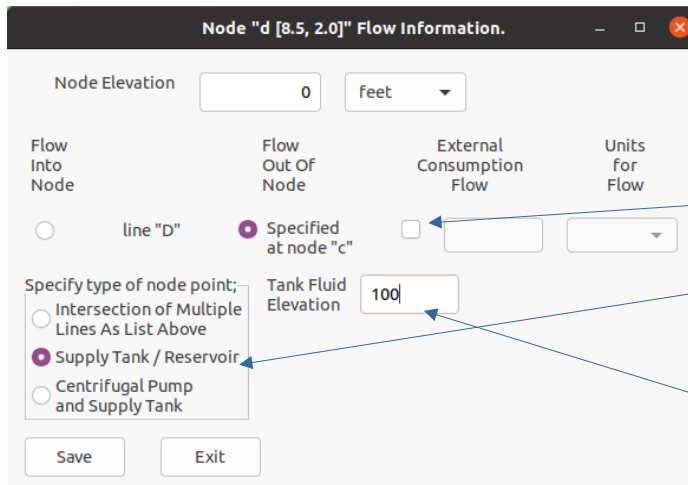
Change the original Node form for node 'd' to look like this.

2) Specify a fluid level in the tank

3) make sure the flow direction is selected in this case I elected it to flow into the circuit

1) remove the consumption flow by clicking the check box
2) select the bullet next to 'Supply Tank / Reservoir
3) The 'Tank Fluid Elevation' text box will appear and you can enter the fluid elevation.

The Pipe and Fittings form for line 'D' well need to be changed to include a pipe size of 6" with a length of 200' and a roughness of .02"
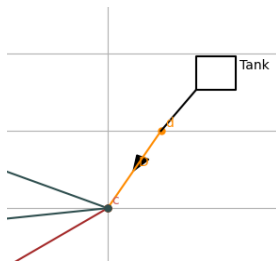
You will note that apart from the flow arrow appearing on the graphics a tank figure has been added.
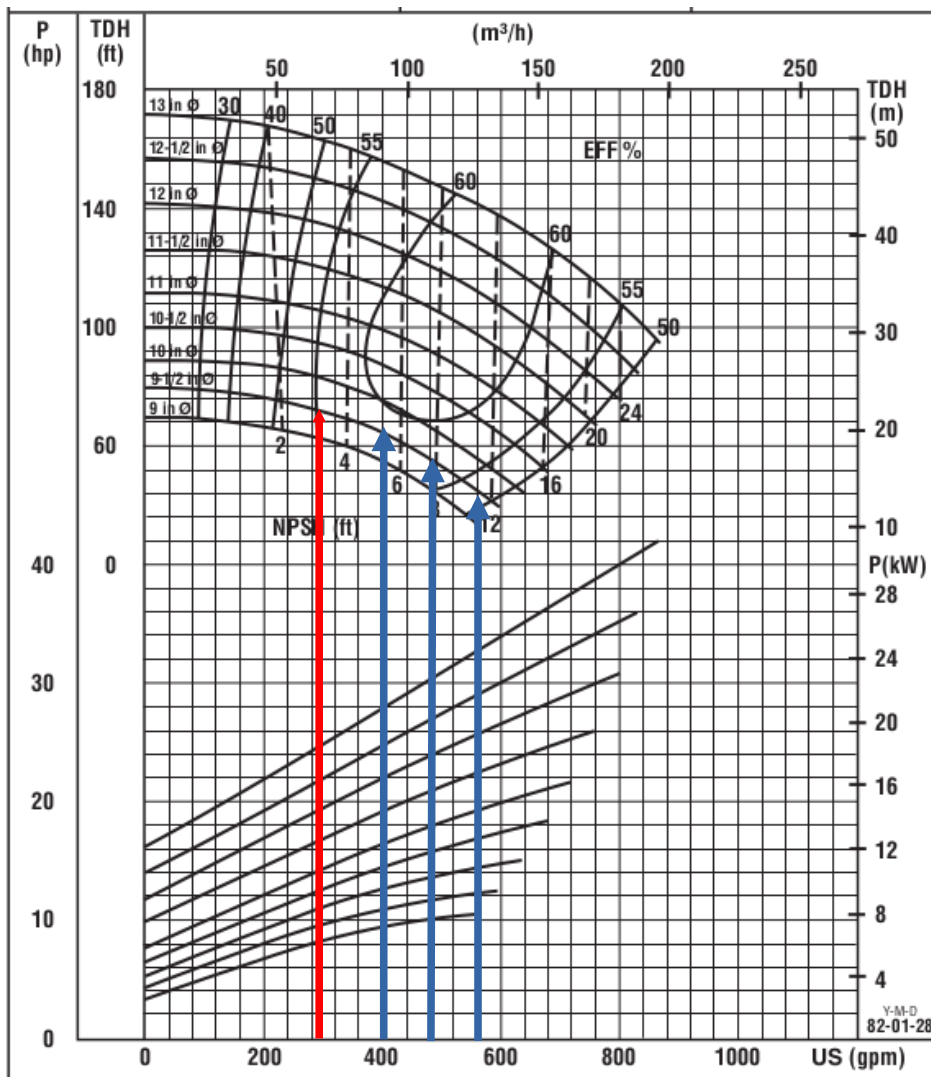
Next lets change node 'origin' to include a pump, but first some pump data is needed. In order to proceed a pump curve will be needed for a constant speed and set impeller size. You will need to select 3 points from the pump curve which define flow and corresponding head.

Reading 3 arbitrary points for a 9-1/2" dia impeller (blue arrows).

The red arrow reveals the actual calculated operating point.

Flow (USGPM)	TDH (ft)
400	65
480	57
560	44

Click on the node 'origin' label and modify the node form by clicking the Centrifugal Pump and Supply Tank bullet. Doing so pops up the text boxes for the added information. Specify the data from the pump curve and add a fluid level to the tank. Make sure the flow direction is correct that is out of the pump node. This time when you save the node data a small pump and tank will be drawn on the graphics.

You will also need to specify the line information for the pump discharge line 'A', 6" diameter, 1000' long and roughness of .02". You will know this because the cell for line 'A' initially is yellow, but once the diameter, roughness and length are defined it will revert to green.

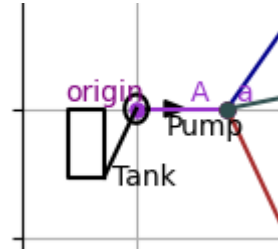If you try to run the calculation at this point you will get the warning;

A total of 8 unknowns have been declared but only 7 equations have been specified.

There is a potential for 1 pseudo loops, 0 have been defined

OK
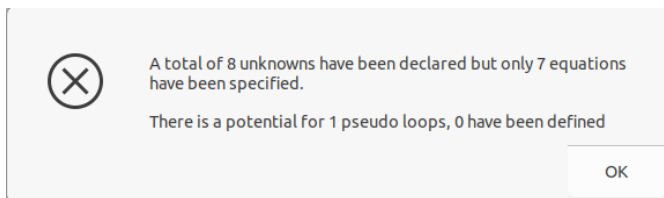
When you add a pump to the system you introduce and added unknown. This is where the pseudo loop comes to play. Selecting a pseudo loop is the same as a closed loop except you click the 'Select Pseudo Loop' button. A pseudo loop needs to run between two supply points (tank, pump or Control Valve). If there was only one tank, pump/tank of control valve in the system you would not be able to select a pseudo loop.

Once you have clicked the 'Select Pseudo Loop' a warning about pseudo line selection pops up. It just means you have to start the selection at a pump or tank discharge line. (Control valves come later.)

The first selected line must connect to a tank, pump or contain a control valve

OK
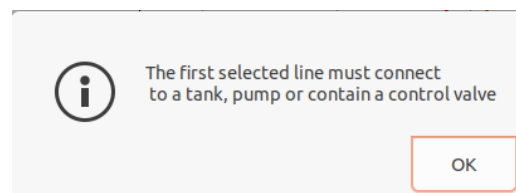
Select lines 'A' , 'G', and 'D'. When you select the last line with the pump/tank or tank connected to it the lines revert to colour and the dotted loop arrow line is drawn. Again the pseudo loop must start and end with a tank or pump/tank node.
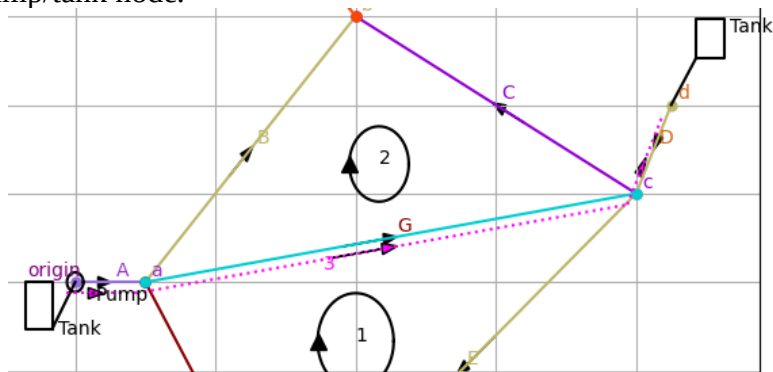
You can now run the calculation.

When you do if all is good you will get the form to the left. The numbers may vary depending on the values you entered for: elevations, pipe roughness, etc.

Data Completed ⊗

Line Label	Flow ft^3/s
A	0.8681190121007205
B	0.35305160740222863
C	0.6469483925977714
D	1.1318809878992797
E	0.48487358817772824
F	0.5151264118222718
G	-5.900712378009594e-05

Save Results   Exit

The printed information includes;
  -pump data
  -control valve data
  -system data covering:
     flow,
     velocity,
     headloss,
     Reynolds,
     pressure at the nodes
     etc.

The pdf report file will be saved
in the same location as the database,
using the database file name.

PRV and BPV Entry

First some information regarding **P**ressure **R**elief **V**alves (not safety valve) and **B**ack **P**ressure **V**alves.

A PRV's purpose is to maintain a constant pressure on its downstream side, independent of upstream pressure. Only two exceptions to the maintenance of this downstream pressure exist:

1. if the upstream pressure becomes less than the valve setting. If this occurs, the valve has no effect on flow conditions except to create a local loss; generally its effect is then like a globe valve in dissipating additional head beyond the friction loss in that line.

2. if the downstream pressure exceeds the pressure setting of the valve so that flow would occur in the upstream direction if the PRV were not present. Occurrence of this and PRV acts as a check valve, preventing a reverse flow in the line. Then the PRV allows the pressure immediately downstream from the valve to exceed its pressure setting.

A BPV's purpose is to maintain a constant pressure upstream, independent of the value of the downstream pressure. This type of valve can be used at higher elevations of a network to maintain pressure in that area. Again there are exceptions

1. should the upstream pressure become less than the pressure setting, the valve can not maintain the pressure setting and it will shut down the flow in its line.

2. should the flow want to reverse direction from the positive flow direction through the valve, the valve opens completely and acts as a local loss device.

Some of the requirements for adding a valve to the graphics;

1. if you add a valve to a line which is associated with either a closed loop(s), the loop(s) will be removed. PRV and BPV cannot be part of a closed loop.

2. If you try to add a valve to a line which is part of a pseudo loop(s) the loop(s) will be removed. PRV and BPV are usually part of a pseudo loop but flow direction is important in selecting the pseudo loop.

3. As mentioned flow direction is important in selecting pseudo loops containing a valve. In the case of a PRV the pseudo loop arrow must approach the valve from the downstream direction. In the case for BPV the direction must approach from the upstream side. No worries the program will prompt you with a node which must be on the next line selected, it will also let you know if your selection is wrong.

Lets add a PRV to the line 'F' by opening the Pipe and Fitting form, by clicking on line label 'F'. Now select the check box to 'Add a Pressure Relief Valve' and a few things will happen;
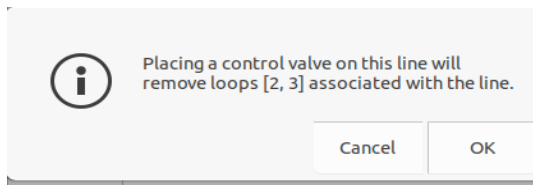
this warning will appear and after clicking 'OK' the one loops will be removed from the graphics, and some text boxes will appear on the Pipe and Fittings form.

Specify the valve set pressure say 55' and the length of pipe downstream call it 500'. Select the 'File' --> 'Data Complete' and try to rerun the calculations. You get a warning information is missing, that's because a loop was removed and something needs to replace it. Lets start with the valve pseudo loop, click 'Select Pseudo Loop Lines' and select line 'F' after acknowledging the warning about having to select the line containing the valve. You are informed that the next line selected must terminate at node 'e', acknowledge this warning and select line 'E'. This loop must end with a tank or pump as any pseudo loop so select line 'D' as the final line. A new pseudo line from the valve to the tank should appear. You have now replaced the deleted closed loops and your final screen should look like this;
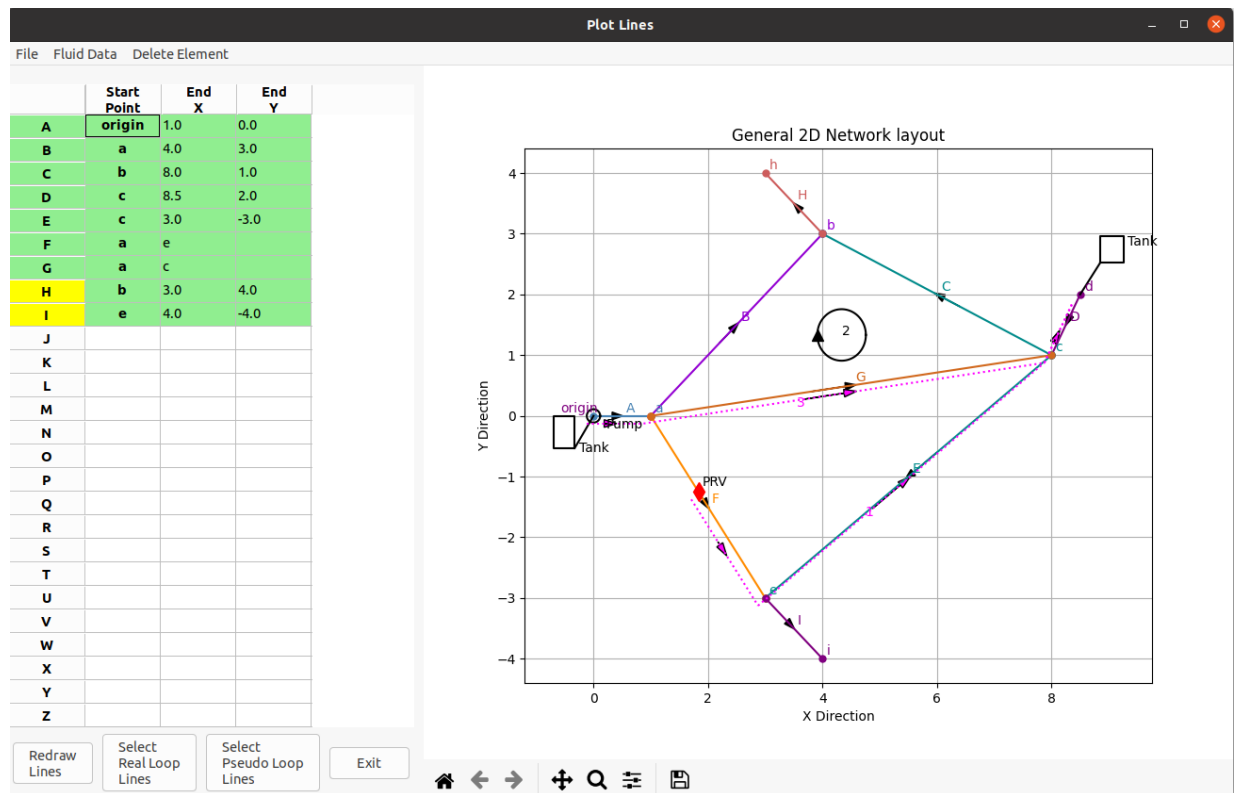
The final test comes when you select the File --> Calculate menu item and get this window and then save the results.

Data Completed ✕

Line Label	Flow Ft^3/s
A	1.1106184746896481
B	1.0785135411547233
C	-0.07851354115472325
D	0.8893815253103508
E	0.9682109220814759
F	0.03178907791852409
G	0.0003158556164010943

[Save Results] [Exit]

The printed information includes;
 -pump data
 -control valve data
 -system data covering:
  flow,
  velocity,
  headloss,
  Reynolds,
  pressure at the nodes
  etc.

The pdf report file will be saved in the same location as the database, using the database file name.

To save the report click the 'Save Results' button and you will see the following message, it tells you where your report has been saved, it should be the same name and location as the database file you entered at the start of this example, with a .pdf extension.

The report data has been saved as /home/paul/Desktop/manual.pdf and can be viewed using the view command in the File drop down menu.

[OK]
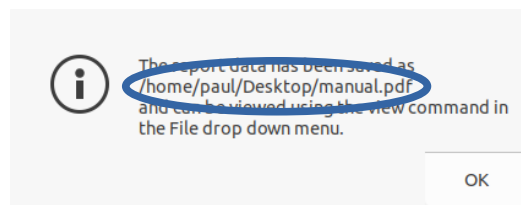
The Final Report

The attached report is divided into 10 areas over 3 tables;

- Line data table;
1. this is the information entered by you for each line label
2. the final calculated flows in three different units
3. the total line loss for the given flow
4. head loss converted to psig based on fluid density
5. the fluid velocity in the line
- Node data table;
6. node label
7. node head
8. node pressure based on fluid density
- Pump data table
9. The pump information is based on the theoretical calculation of the pump curve and may not be an exact point on the pump curve, but as you see here it is only off by a few feet of TDH for the calculated flow. The curve indicates about 70' TDH for the flow of 290 USGPM (see red arrow on previous pump curve).
- Control Valve data table
10. When a control valve is introduced its information will appear here. For PRV valves the downstream pressure should approximate the set pressure and for BPV the upstream pressure should be the same as the set pressure.
- Fittings data table by Line Label
11. When fittings as specified in the Pipe and Fittings form the information will appear here, if there are no fittings specified for a line then the table will not be shown.

Calculated Data for expl_2.db

Line Label	Pipe Dia inches	Pipe Length	Flow ft^3/s	Flow USGPM	Flow m^3/s	Head Loss feet	Head Loss meters	Pressure Drop psig	Velocity ft/s	Velocity m/s
A	6.0	1000.0	1.114	499.83	0.032	27.311	8.324	11.8	5.66	1.73
B	6.0	1000.0	1.075	482.44	0.03	25.459	7.76	11.0	5.47	1.67
C	6.0	800.0	-0.075	-33.61	-0.002	0.117	0.036	0.05	0.38	0.12
D	6.0	200.0	0.886	397.83	0.025	3.476	1.059	1.5	4.51	1.37
E	6.0	2000.0	0.968	434.68	0.027	41.418	12.624	17.89	4.93	1.5
F	6.0	1000.0	0.032	14.15	0.001	0.03	0.009	0.01	0.16	0.05
G	1.0	1500.0	0.007	3.25	0.0	25.575	7.795	11.05	1.33	0.4

1   2   3   4   5

Node Label	Elevation feet	Elevation meters	Head feet	Head meters	Pressure psig	Pressure kPa
origin	0.0	0.0	149.039	45.43	64.4	9.34
a	50.0	15.15	121.728	37.1	52.6	7.63
d	0.0	0.0	100.0	30.48	43.21	6.27
c	50.0	15.15	96.153	29.31	41.55	6.03
e	20.0	6.06	54.735	16.68	23.65	3.43
b	50.0	15.15	96.27	29.34	41.6	6.03

6   7   8

Pump Node	Head feet	Head meters	Flow USGPM	Flow m^3/hr
origin	59.039	17.995	499.798	113.524

9

10

Line Label	Valve Type	Set Pressure feet	Set Pressure psig	Set Pressure kPa	Upstream Pressure feet	Upstream Pressure psig	Upstream Pressure kPa	Downstream Pressure feet	Downstream Pressure psig	Downstream Pressure kPa
F	PRV	55	23.76	163.85	121.73	52.53	362.18	54.73	23.62	162.85

Fittings for line B	Quantity
Ball Valve Full Port	1
Ball Valve Reduced Port	3
Plug Valve 3-Way Straight	6

11