This short guide will show you within 60 seconds how easy it is to use PEW and how quickly you can perform calculations and display the results both graphically and in a table.

We will calculate the flowrate of water in 500ft of 2” unlined cast iron pipe up a 10ft hill for pressure drops between 0.5 and 3 bar diff.

Starting PEW

1. Click Start > All Programs > PEL > PEW. The main PEW window appears.

   There are the usual menus and tool bars at the top of the screen.

   The first thing we need to do is to select the type of calculation.

2. Click the Add a new case button on the toolbar to open the Calculation type window. The different types of calculation appear in the left pane; click Fluid Flow. The available calculations for Fluid Flow appear on the right pane; click Incompressible and then click OK.

   The Incompressible Flow form opens.

Next, we need to input the pipework data

3. Click Flow to calculate the flow rather than the pressure drop (the default).

4. Double-click the Pipework Length box to select the default value of 10m. Type 500 followed by a space followed by ft. Press Enter on the keyboard or click another input box to do the conversion.

5. Double-click the Diameter box to select the default value. On the Tools menu, click Pipe Inner Diameter Calculator. Select 2” and 40/STD/40S and then click OK. The result is pasted back into the Pipe Diameter.

6. Double-click the Roughness box to select the default value. On the Tools menu, click Pipe Roughness Calculator. Select Cast iron, concrete, timber and then click OK.

7. Double-click the Static head loss box to select the default value and type in 10 ft. Don’t forget the space!

8. Move to the Fittings section of the dialog. In the Number of Items column, type in 1 for one 90 degree circular bend. At the bottom of the form, type in 5 miscellaneous losses and then return to the Inputs section.

Now we need the density & viscosity of water. We will use the Physical Property Calculator to generate them.

9. Click in the Density box. Next, on the Tools menu, click Calculate Physical Properties. When the Calculator appears, if any components appear in the worksheet click Clear Worksheet to clear them. Next, click Add Component. This will bring up the Select Components window; enter water in the Search for Name box, select WATER in the results list and then click Add to stream to add water to the Calculator and then click Close. Enter a temperature of 20°C and a pressure of 1 bar and click Calculate. The Calculator returns a density of 999.48 kg/m3 and a viscosity of 0.9983. Click OK to return these values to the Incompressible Flow form.
Finally, we need to enter the pressure drop.

10. Double-click the **Pressure drop** box to select the default value and type in **0.5 bar diff**.

So now we are ready to do the calculation.

11. Click the **Calculate** button on the toolbar and the results appear in blue on the right hand side. You should get a value of 1.372kg/s for the Mass flow.

We need to repeat the calculation over a range of pressure drops between 0.5 and 3 bar. We’ll do it at 1, 2 & 3 bar. But we want to save each set of results not overwrite them so we need to copy the calculation before we make the changes.

12. Click the **Copy** button on the toolbar. This produces a copy of the first case. Change the title from “Copy of No 1” to “No 2”.

13. Double-click the Pressure drop and change the value to **1 bar diff**.

14. Click the **Calculate** button on the toolbar to re-calculate the flowrate. You should get 2.608 kg/s for the flowrate.

15. Repeat the last 3 steps for pressure drops of **2 and 3 bar diff**. You should get flowrates of 4.095 & 5.175 kg/s.

Now we are ready to plot the graph.

16. Click the **Create a graph** button on the toolbar.

17. In the **Graph – select calculation type** dialog, click **Incompressible** and then click **OK**.

18. In the **Graph – select X axis** dialog, click **Pressure drop** and then click **OK**.

19. In the **Graph – select Y axis** dialog, click **Mass flow** and then click **OK**.

And there’s your graph. Now let’s create a Summary Table.

20. Click the **Create a summary** button on the toolbar & enter **Demonstration** as the summary title.

21. Click the **Add a case to the current summary** button on the toolbar.

22. In the **Summary – select case** dialog, select all 4 cases, and then click **OK**.

23. Click the **Add a variable to the current summary** button on the toolbar.

24. In the **Summary – select variable** dialog, select **Pressure drop, Massflow, Velocity, & Reynolds number** as the columns for the table, and then click **OK**.

And there’s your Summary Table!