PEL PIPER

User Guide

About this document

This guide is designed to assist the user in becoming quickly familiar with the capabilities of PIPER, its interface and how the program is used.

It has been produced to the recommendations of British Standard BS7649 – Guide to the design and preparation of documentation for users of application software.

Trademarks

All trademarks acknowledged.

.

Associated PEL Support Services Documents

PIPER Reference Guide: This document describes the technical basis of the fluid flow calculations performed within PIPER.

PIPER System Guide: This document describes the system specific aspects of PIPER that fall outside the scope of the PIPER Reference and User documents.

Contacting PEL Support Services

This program is developed, maintained and supported by PEL Support Services, ABB. We run a Hotline telephone and email service to answer any queries about PIPER.

Please let us have any suggestions on how you feel we could improve PIPER. You can contact us by any of the following routes:

By Telephone: ++44 (0)1925 74 1126

By Fax: ++44 (0)1925 74 1265

By E-mail: pel.support@gb.abb.com

By Post: PEL Support Services
ABB Ltd.
Daresbury Park
Daresbury
Warrington
Cheshire
WA4 4BT
United Kingdom

Owner:	M. G. Pass, ABB.
Approved By:	M. G. Pass, ABB.
Document Version / Issue Date:	Version 1.4 / 24 October 2005
Last Amended Date:	24 October 2005
Last Amended By:	G. Langford, ABB.

Change history

This table records the changes made to each new revision of this document.

Changes to approved issues are indicated by a double revision bar on the outer margin next to the text. This is an example.

Revision	Date	Description of change	
1.0	20 April 2000	First Approved Issue	
1.1	23 Oct. 2000	Second Approved Issue comprising the following:	
		Amended:	Section 2.1 – Title, introduction text and Figure 1 updated.
		Amended:	Section 2.6.2.3 – Text and figure updated for Valves Tab.
		Amended:	Section 2.7 – PIPER Toolbar format revised.
1.2	21 March 2001	Third Approved Issue (ABB logo added).	
1.3	10 October 2002	Fourth Approved Issue comprising the following:	
		Amended:	Industrial IT logo & paragraph added, “Eutech” removed, Front page modified.
1.4	24 October 2005	Amended	Removed Industrial IT logo & paragraph.

Contents

1. PIPER User Guide 7

1.1. Introduction **7**
1.1.1. PIPER System Specification 7
1.1.2. PIPER Calculation Assumptions 8
1.2. About this Guide **8**

2. The PIPER User Interface 9

2.1. The PIPER start up screen **9**
2.2. Conditions Tab **10**
2.3. Physical Properties Tab **12**
2.4. Pipe Editor Tab **14**
2.4.1. Additional information on input data required for Orifices and Valves 15
2.5. Summary Results Tab **16**
2.6. PIPER Menus **17**
2.6.1. File Menu 17
2.6.2. Tools Menu 17
2.6.2.1. Pipe Inner Diameter Calculator 18
2.6.2.2. Molecular Weight Calculator 19
2.6.2.3. K-value Calculator 20
2.6.2.4. Pipe Roughness Calculator 23
2.6.2.5. Calculator 24
2.6.2.6. Text Editor 24
2.6.3. Options Menu 25
2.6.3.1. Calculation Options 25
2.6.3.2. Show Calculation Progress 26
2.6.4. Calculate Menu 26
2.6.4.1. Additional information on using the Calculate option 26
2.6.5. Results Menu 27
2.6.6. Help Menu 27
2.7. The PIPER Toolbar **28**

3. PIPER Tutorial 29

3.1. General **29**
3.2. Solving the Network **29**
3.2.1. Defining a Physical Properties file 29
3.2.2. Accessing PIPER 29
3.2.3. Entering Conditions data 30
3.2.4. Entering Physical Properties data 30
3.2.5. Defining the Pipe fittings using the Pipe Editor 31
3.2.6. Calculating the Network Solution 33
3.2.7. Viewing Detailed Results 33
3.2.8. Printing Results 33
3.2.8.1. Print Detailed Results menu option 33
3.2.8.2. Print Summary of Calculation Results 35
3.2.9. On-screen Graphs 37
3.2.10. Saving Detailed Results 37

Tables

Table 1 The PIPER Toolbar 28

Figures

Figure 1 PIPER Start up screen 9
Figure 2 The Conditions Editor 10
Figure 3 Physical Properties Editor 12
Figure 4 The Pipe Editor 14
Figure 5 A typical Summary Results dialogue after a successful calculation 16
Figure 6 Pipe Inner Diameter Calculator dialogue 18
Figure 7 Molecular Weight Calculator dialogue 19
Figure 8 Fittings Loss (K-value) Calculator dialogue 20
Figure 9 Pipe Roughness Calculator dialogue 23
Figure 10 Calculator dialogue 24
Figure 11 Calculations Option dialogue 25
Figure 12 PIPER - Calculation example 29
Figure 13 Physical Properties Editor showing .PPB file data 30
Figure 14 Pipe Editor showing Contraction values 31
Figure 15 Pipe Editor showing Orifice values 32
Figure 16 Pipe Editor showing final values 32
Figure 17 Summary Results dialogue 33
Figure 18 Results Report 35
Figure 19 Physical Properties Report 36
Figure 20 Results Graphs 37
Figure 21 Save Results dialogue 37
Figure 22 Choosing a Fluid Flow Program 43

Appendices

Appendix A – Creating a .PPB file 39
Appendix B – In-cell Units Conversion 41
Appendix C – Choosing a Fluid Flow Program 42
Appendix D – Troubleshooting 44

1. PIPER User Guide

1.1. Introduction

PIPER is used to calculate the pressure change or flow of a liquid, two-phase mixture or gas through an unbranched closed piping system.

The program makes allowance for heat transfer through the pipe wall and change of phase at any point in the system. Variations of phase compositions and physical properties with temperature and pressure are accounted for during the calculations.

PIPER will not model a solid phase or a two liquid phase system.

The inlet stream to the system is defined by the inlet pressure, mass quality or temperature and overall stream composition which are supplied by the user.

Note. The mass quality is defined as the mass fraction of vapour in the stream.

PIPER has two modes of operation:

Specify flowrate and inlet pressure	Pressure drop is calculated. If choked, the maximum non-choked flowrate is calculated.
Specify inlet and exit pressures	Flow rate is calculated. If choked, all pressure discontinuities are modelled.

1.1.1. PIPER System Specification

Although the piping system in PIPER must be a single unbranched pipe, this pipe can be made up of the following:

- Straights
- Bends
- Contractions
- Enlargements
- Gate Valves
- Globe Valves
- Orifices.

The definition of the bends and straights elements allows for full description of the three dimensional geometry of the piping system.

In addition to defining the piping system, PIPER requires a detailed description of the variation of the fluid physical with the temperature and pressure. This is achieved by specifying a physical properties (.PPB) file produced using a product such as PhysPack. This allows the physical properties to be calculated as a continuous function of temperature and pressure throughout the system.

1.1.2. PIPER Calculation Assumptions

Equilibrium is assumed between phases, that is, no allowance is made for heat/mass transfer limitations within a phase, between phases or between the fluid and its surrounding.

A homogenous model for the two phase flow is implemented. This means that no allowance is made for slip between phases.

Note. PIPER uses strict IS units for its input and results. However, like all PEL programs, it can perform in-cell units conversion. See Appendix B for more information about this feature.

1.2. About this Guide

This guide is designed to assist the user in becoming quickly familiar with the capabilities of PIPER, its interface and how the program is used.

The chapters are organised as follows:

Chapter 1	–	An introduction to PIPER.
Chapter 2	–	Details the PIPER user interface.
Chapter 3	–	A tutorial to guide the user through a typical PIPER session emphasising the commonly used features. It is recommended that the user should read this chapter while running the program.
Appendix A	–	Detailed instructions for creating a physical properties .PPB data file.
Appendix B	–	Describes the in-cell units conversion facility.
Appendix C	–	Choosing a Fluid Flow Program.
Appendix D	–	Troubleshooting.

2. The PIPER User Interface

The PIPER user interface displays tabs that allow easy definition of the systems being modelled. It also offers many useful features including automatic unit conversion and a calculator.

On-line documentation is available within the program.

2.1. The PIPER start up screen

The PIPER start up screen consists of the following elements:

Menu Bar	Displays the menu options that are available.		
Toolbar Buttons	Descriptive text appears automatically as the cursor is held over the buttons.		
Tabs	The start up screen also displays four tabs. These are:		
	Conditions	Allows the conditions for the calculation to be entered.	
	Physical Properties	Allows a Physical Property (.PPB) data file to be input into PIPER.	
	Pipe Editor	Used to add the fittings to the pipe.	
	Summary Results	Displays the results of the calculation.	

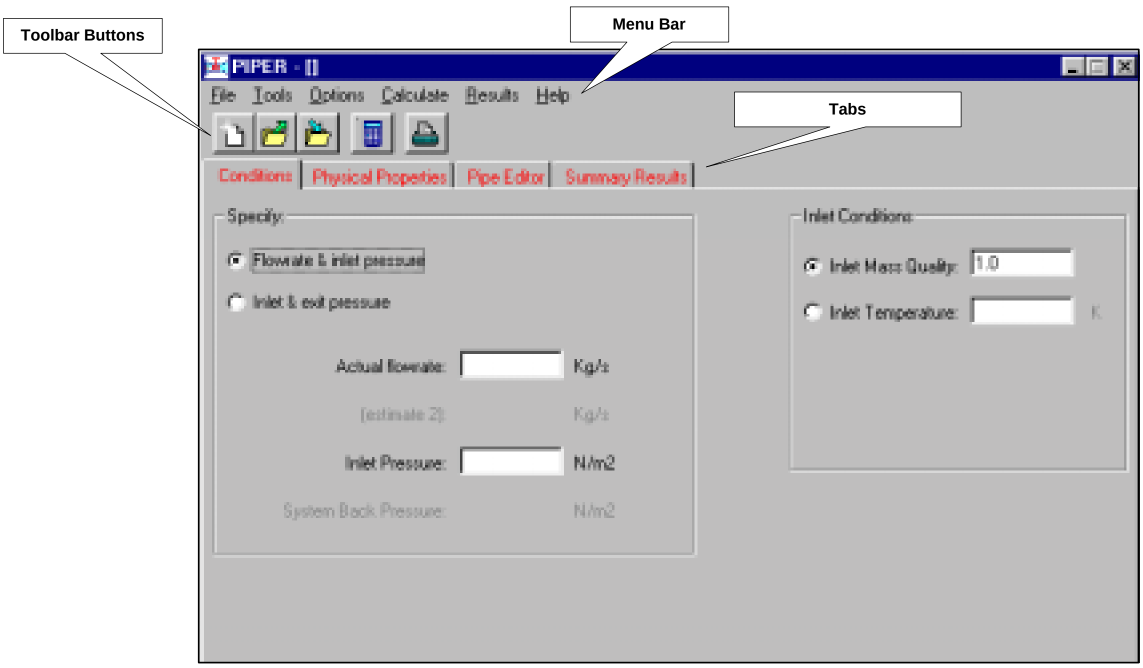


Figure 1 PIPER Start up screen

The following sections of this chapter contain detailed information on the functions available within these tabs.

2.2. Conditions Tab

The Conditions tab allows the conditions for the calculation to be entered.

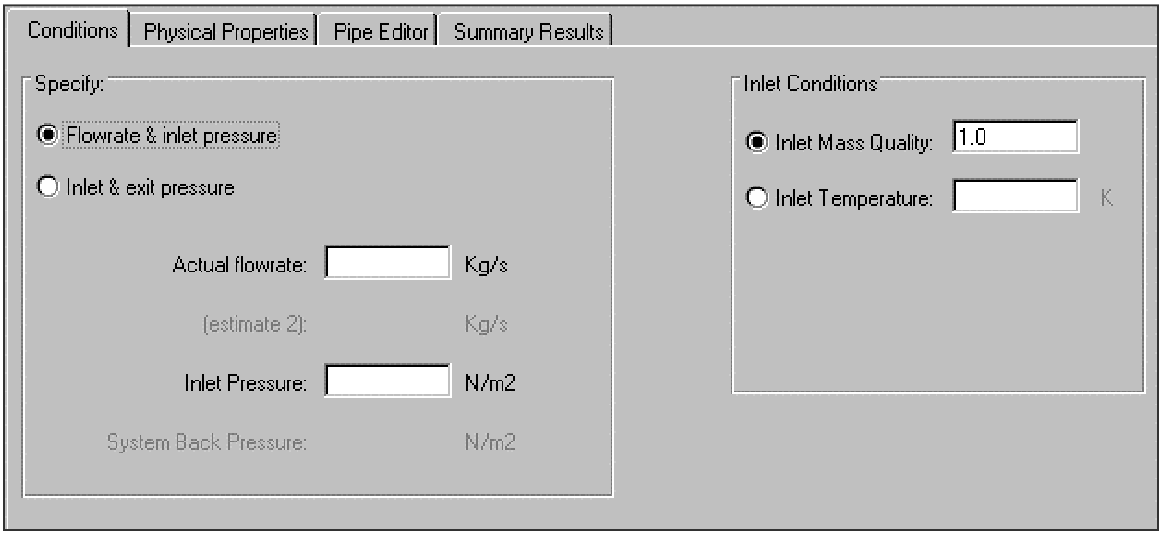


Figure 2 The Conditions Editor

Usage

Specify: Specify the 'Flowrate & inlet pressure' **OR** the 'Inlet & exit pressure'.

Flowrate & inlet pressure

Selecting this means that the calculation takes the following form:

- Calculation of the pressure drop at a specified flowrate
- Calculation of the maximum non-choked flowrate if the specified flowrate is choked.

Data input selections available are:

- Actual flowrate (Kg/s)
- Inlet Pressure (N/m^2)

In both cases the results also provide:

- Inlet and outlet conditions at the solution flowrate.
- Detailed conditions and physical properties for each element in the pipe system.
- Flow and physical property warning messages.

Note. PIPER is capable of examining vacuum as well as pressurised systems. This means that the outlet conditions are not limited to atmospheric pressure. Always check the results when using this calculation mode to ensure that the outlet pressure does not go below 1 atm when examining systems that exhaust into the atmosphere.

Inlet & exit pressure

This is only active when 'Flowrate & inlet pressure' is not selected.

- Calculation of the flowrate with specified inlet and back pressures. If the calculated flowrate is choked, PIPER calculates the maximum non-choked flowrate.
- All the pressure discontinuities within the system are modelled at the maximum non-choked flowrate if the system is choked.

Data input selections available are:

- Flowrate (estimate 1) Kg/s
- Flowrate (estimate 2) Kg/s (this is optional)
- Inlet pressure (N/m^2)
- System Back Pressure (N/m^2).

Inlet Conditions: Specify the 'Inlet Mass Quality' **OR** the 'Inlet Temperature (Temperature or Mass Quality may be supplied for a multi-component two-phase flow).

Inlet Mass Quality Remember the following points when entering data:

- Mass Quality must be supplied for a single component two-phase flow.
- For all cases, a Mass Quality of 0.0 or 1.0 denotes a saturated liquid at bubble point or a saturated vapour at dew point respectively.

Inlet Temperature Temperature must be supplied for a subcooled liquid or a superheated gas flow.

2.3. Physical Properties Tab

As PIPER works with a rigorous physical properties model, physical property data must be input into PIPER by means of a Physical Property (.PPB) data file produced using PhysPack.

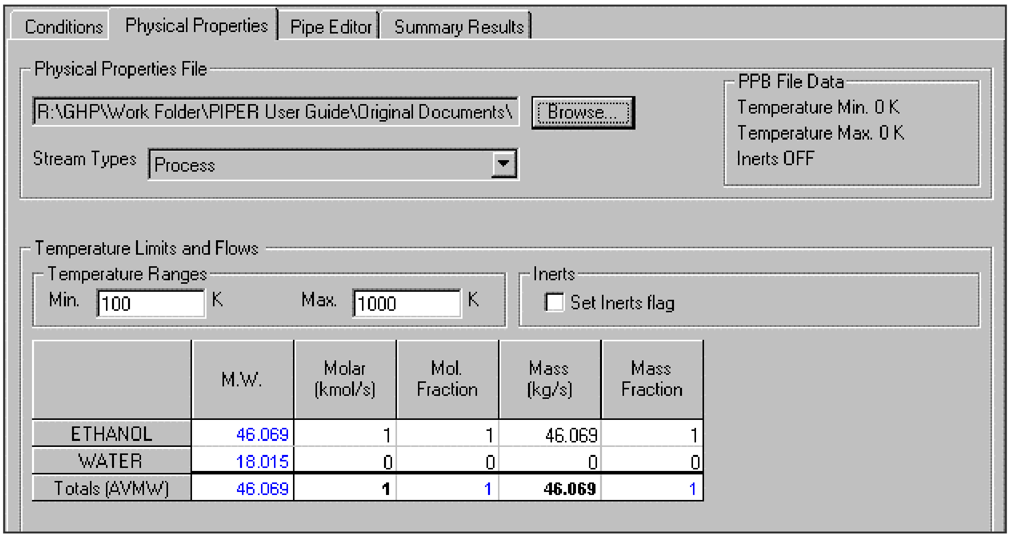


Figure 3 Physical Properties Editor

The .PPB file supplies the following information to PIPER:

- Chemical components present in a system
- Physical properties required by PIPER
- Methods selected to calculate the physical properties
- Data coefficients for each component and property.

The following properties must be specified in the PPB file for PIPER irrespective of the flow to be modelled:

- Molecular Weight
- Critical Pressure
- Critical Temperature
- Liquid Density
- Vapour Density
- Liquid Enthalpy
- Vapour Enthalpy
- Liquid Viscosity
- Vapour Viscosity
- Ratio of Specific Heats
- Vapour Liquid Equilibrium.

Note. The user must verify the physical property model they have created before running PIPER (see Appendix A for more information).

The physical properties package Properties Plus produced by Aspen Technology Inc. can also be used to generate an Aspen Physical Properties Datafile (.APPDF) for use by PIPER.

Usage

Physical Properties File:

Physical Properties File 'Browse'	Mandatory. Use the 'Browse' button to select the required physical properties .PPB or .APPDF file.
Stream Types	Mandatory. Select the stream type from the pull down list and enter the composition of the inlet stream. • This can be done in terms of mass or molar flowrate, mole fraction or mass fraction. • PIPER provides the option to normalise the data if the fractions do not add up to 1.00.

Temperature Limits and Flows:

Temperature Ranges	Mandatory. The minimum and maximum temperature ranges that are expected when modelling the flow through the system. • The user should ensure that the physical property model has been verified over this range. • Limits can be extended during the calculation if they prove to be inadequate but cannot be extended beyond the range of the underlying physical properties file.

Inerts:

Set Inerts flag	Optional. Checking this box means that you must enter the threshold level of composition below which no warning messages are produced when the inerts switch logic is implemented. The inerts switch is used by PhysPack to allow calculations to proceed when data is missing or out of range for components with low concentrations. Typically this is implemented when calculating properties for mixtures that contain supercritical components. **Note.** This option is normally left unchecked.

2.4. Pipe Editor Tab

The Pipe Editor tab is used to add the fittings to the pipe.

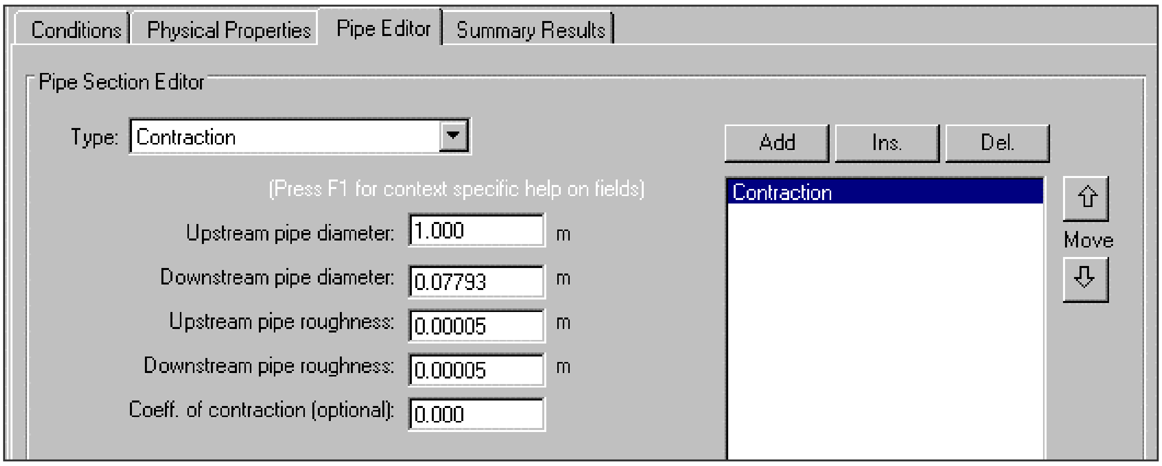


Figure 4 The Pipe Editor

Usage

Pipe Section Editor:

To **Add** a section: Select a section from the ‘Type:’ box in the top left of the Editor. The appropriate data entry fields appear below the box so the relevant details for each pipe section can be entered.

To **Edit** a section: Click on the section in the Section list. The data associated with that section is then displayed in the data entry fields on the left of the Editor.

Type

Mandatory.

Select a section type from the following list:

- Bend
- Contraction
- Enlargement
- Gate Valve*
- Globe Valve (Cast Body)*
- Orifice*
- Straight.

* See section 2.4.1 for more information on Orifices and Valves.

The data entry fields displayed vary depending on which section type is selected. Help is available for any field by pressing F1 while that field is activated.

Usage

Conduit section comment	Optional. Use this field to record any explanatory comments for the selected section.
Add	The **Add** button (located above the selection list on the right of the screen) adds a section to the list on the right-hand side of the Editor. The new section is added at the end of the list if the pipe already has sections.
Ins.	The **Insert** button adds the current section to the section list below the item currently highlighted.
Del.	The **Delete** button removes the section currently highlighted from the section list.
Move	The highlighted section can be moved **Up** or **Down** the list using the arrow buttons. This allows the sections that comprise the pipe to be rearranged.

2.4.1. Additional information on input data required for Orifices and Valves

Each time an orifice, a globe valve or a gate valve is modelled, the calculation is split into two separate parts:

1. A choke test is performed by modelling the flow as a contraction from the upstream pipe area to the minimum available area for flow within the fitting. (For an orifice this area is taken to be the orifice area).

2. If the flow is not choked, the pressure drop through the fitting is calculated using the K-value to represent the frictional loss.

The K-value is not used in the first calculation and the minimum available area for flow is not used in the second calculation because these data items are independent as far as PIPER is concerned. However, they must be consistent so as to correctly model the fitting (the degree of valve opening is used only in the calculation of K-value).

If the minimum available area for flow is equal to the upstream pipe diameter then no initial choke test is performed but, as with other section types, a choke may be detected while calculating the pressure drop through the fitting.

2.5. Summary Results Tab

The Summary Results tab is activated after a successful calculation.

Options on the 'Results' menu can be used to view, print and save detailed results from the calculation.

The calculation results displayed after a successful calculation are:

- Total flowrate or pressure drop
- Summary description of the calculation and any warning messages
- Graphs of mass quality (two-phase) and static pressure along the pipe
- Tables of conditions and physical properties along pipe.

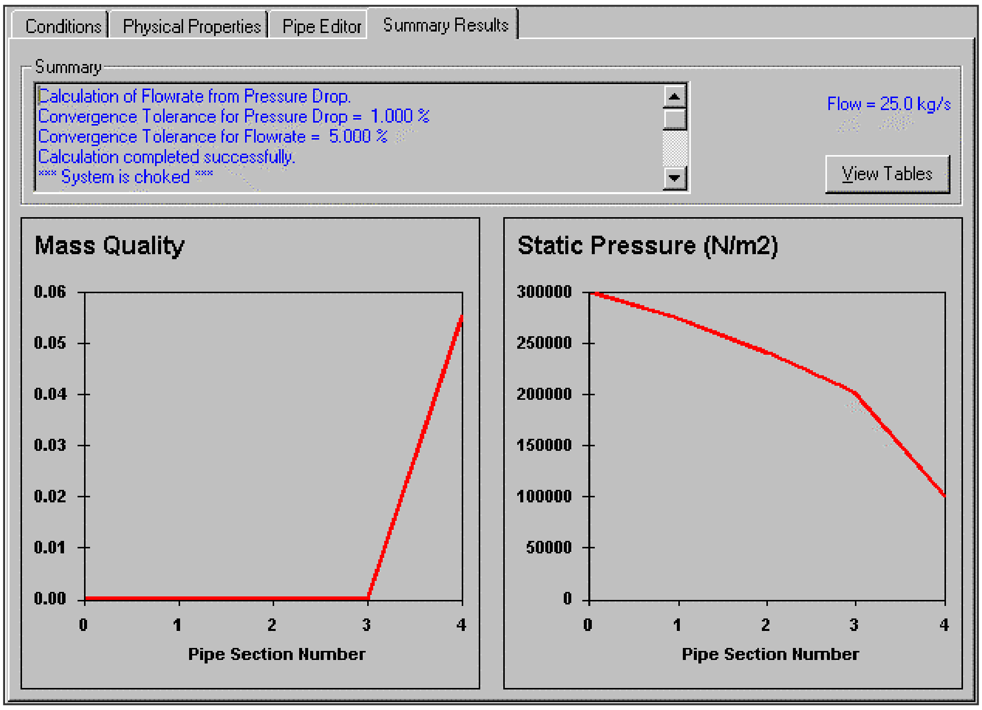


Figure 5 A typical Summary Results dialogue after a successful calculation

The 'View Tables/View Graphs' button allows switching between graph and table views.

The level of diagnostic data output from a calculation can be set by the user. See section 2.6.3 for more information on Diagnostic Output Levels.

2.6. PIPER Menus

This section lists the various menus and describes the options that are available.

2.6.1. File Menu

The File menu allows you to access the following options:

Menu Option	Definition
New	Creates a new file.
Open	Allows an .SCD file to be opened via the ‘Open .SCD File’ dialogue box.
Save	Saves the file (PIPER files are saved with a .SCD extension).
Save As…	Saves the file as a new name.
Print Setup	Displays the ‘Print Setup’ dialogue.

2.6.2. Tools Menu

The Tools menu allows you to access the following options:

- Pipe Inner Diameter Calculator
- Molecular Weight Calculator
- K-value Calculator
- Pipe Roughness Calculator
- Calculator
- Text Editor.

These are detailed in the following sections.

2.6.2.1. Pipe Inner Diameter Calculator

This dialogue is used to calculate the pipe inner diameter.

Select a standard pipe size then the available schedules for that pipe size. The details for that combination are displayed at the bottom of the dialogue box.

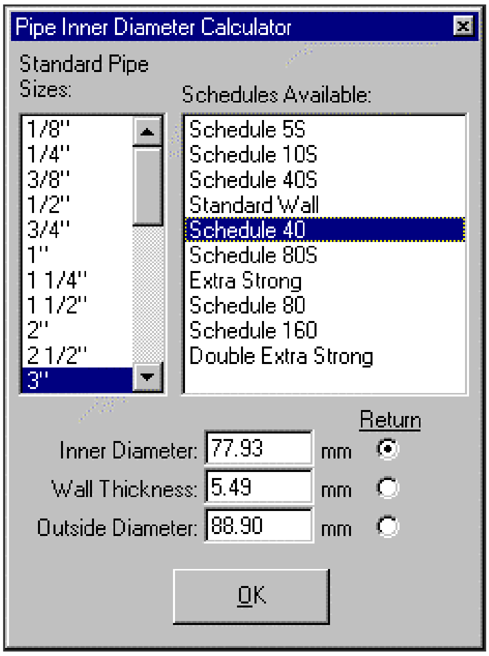


Figure 6 Pipe Inner Diameter Calculator dialogue

The following selections can be made from this dialogue:

Usage

Standard Pipe Sizes	Lists the pipe sizes available from 1/8″ to 36″.
Schedules Available	Lists the pipe schedules available for the pipe size selected.
Inner Diameter	Displays the Inner Diameter of the selected pipe.
Wall Thickness	Displays the Wall Thickness of the selected pipe.
Outside Diameter	Displays the Outside Diameter of the selected pipe.
Return	Use the radio button to indicate which selection is to be returned to the program. The default is Inner Diameter.
OK	Pressing the OK button returns the chosen value to the program. The value is pasted into the cell that was highlighted when the Pipe Inner Diameter Calculator was selected.

2.6.2.2. Molecular Weight Calculator

This calculates the molecular weight of a compound given its formula. The following rules are used to interpret the formula:

- Elements are given their usual atomic symbol, for example, He for Helium and O for Oxygen. The first character must always be upper case and the second (if there is one) lower case. This enables the Molecular Weight Calculator to distinguish formulae such as PO and Po. The calculator regards the first of these as Phosphorous and Oxygen and the second as the element Polonium.

- No other symbols are recognised, for example, common groups like benzene rings, ethyl groups etc.

- Elements can be followed by numbers and are separated by spaces, dots or colons.

- Brackets can be used as required to any level. For example, CH3(CH2)10CH3 would be one way of describing Dodecane (see Figure 7). However, note that in this example the 10 refers to the CH2 group, not the succeeding CH3. Thus water of crystallisation must be specified as Na3SO3(H2O)5.

- The atomic weights used by the Molecular Weight Calculator are taken from Perry's Chemical Engineers Handbook.

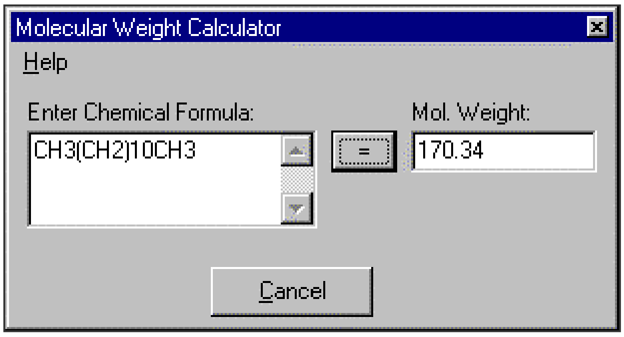


Figure 7 Molecular Weight Calculator dialogue

The following selections can be made from this dialogue:

Usage

Enter Chemical Formula	Enter the chemical formula then click on the '=' button to calculate the molecular weight.
Mol. Weight	Outputs the molecular weight for the formula entered. The molecular weight calculated is automatically pasted into the cell that currently has focus when the dialogue is closed. If this cell does not accept the value it is pasted to the clipboard instead.
Cancel	Pressing the Cancel button pastes the calculated value into the cell that was highlighted when the Molecular Weight Calculator was selected.

2.6.2.3. K-value Calculator

The Fittings Loss (K-value) calculator consists of a number of tabs where the fitting's details of a pipe are built up. The front sheet of the dialogue contains a summary of the following tabs:

- Tee Junctions
- Bends
- Valves
- Expansions/Contractions
- User Defined (Process Equipment)
- Manual Adjustment.

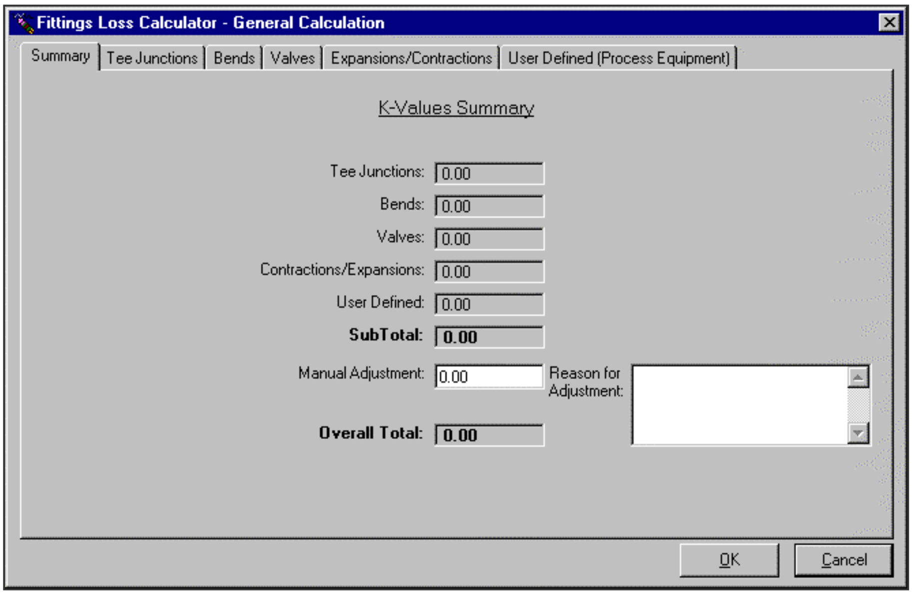


Figure 8 Fittings Loss (K-value) Calculator dialogue

The following selections can be made from this dialogue:

Usage

Summary Tab:

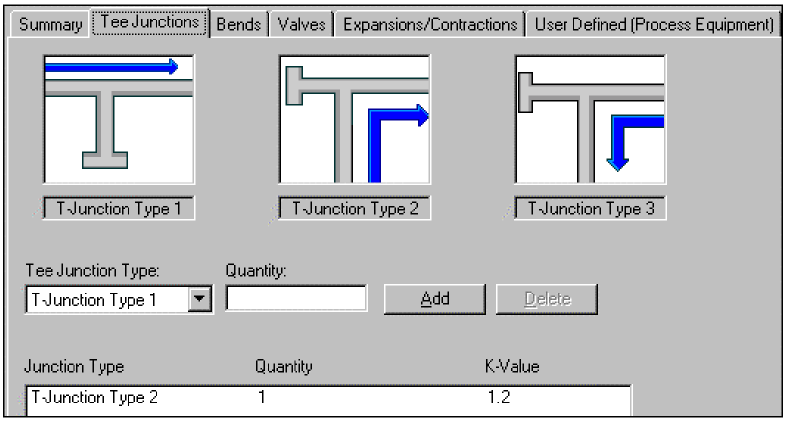
- Sub Total	Outputs the sub total before a manual adjustment is added.
- Manual Adjustment	The manual adjustment field is for entering miscellaneous fittings. It is also used for manually adjusting the model in the validation stage or for studying the effect of changes, for example, changing a control valve position.
- Reason for Adjustment	Text describing the reason for adjustment can be entered in this field.
- Overall Total	Outputs the overall total combining the sub total and the manual adjustment to produce the fittings loss value for the fitting.

Tee Junctions Tab:

The Tee Junctions calculation takes into account any blanked off junctions. This can also be a line where the dead leg is isolated at a valve further downstream.

Add a Tee Junction:	Select a Tee Junction Type, enter the Quantity and click on **Add**. The entry is added to the list at the bottom of the sheet.
Remove a Tee Junction:	Select the Tee Junction then click on **Delete**.

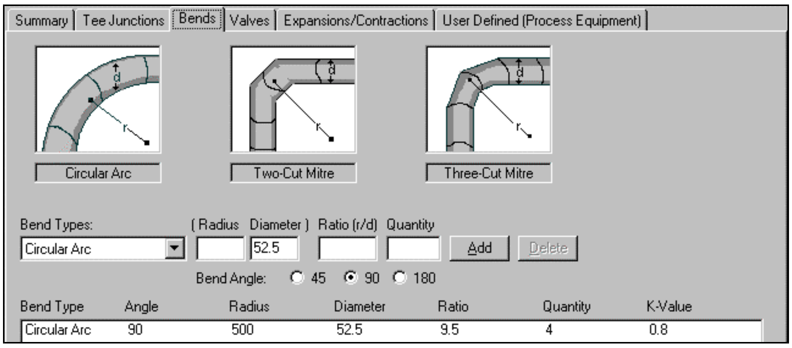
An example of a typical Tee Junction tab display is shown below:



Bends Tab:

Bends are entered and deleted using the same method as for Tee Junctions.

An example of a typical Bends tab display is shown below:



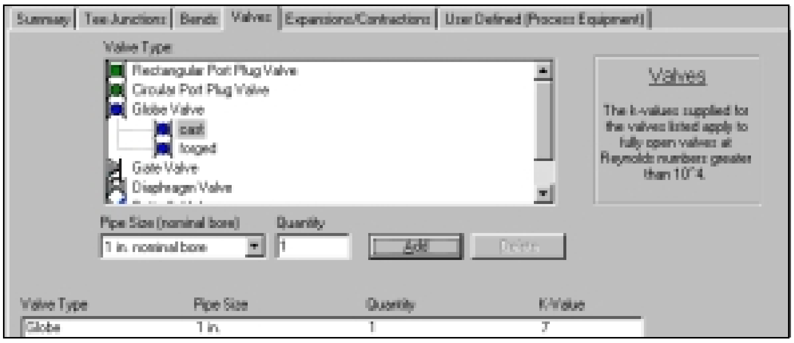
Valves Tab:

Valves are entered by selecting a valve type then double-clicking on it.

This displays the various categories of valve available for that type, for example, the Globe Valve type has two categories – cast valves and forged valves (see below).

Add a Valve:	Click on the category required and select a valve from the 'Pipe Size (Nominal Bore)' drop down list. Specify how many of these valves are required in the 'Quantity' box then click on **Add**.
Delete a valve:	Select the valve from the list and click on **Delete**.

An example of a typical Valves tab display is shown below:

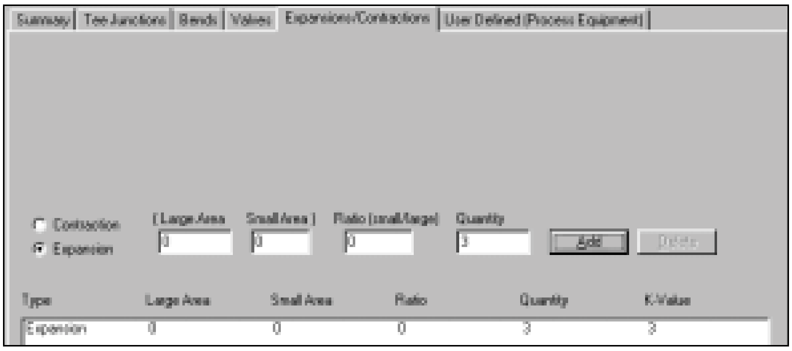


Contractions/Expansions Tab:

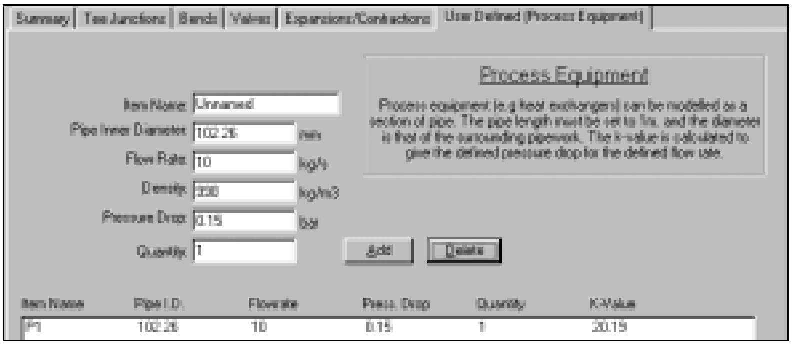
Select either the Expansion or Contraction radio button then follow the same method as for Tee Junctions.

For *exit* losses:	Select an expansion with a small/large area ratio of zero.
For *entry* losses:	Select a contraction with a small/large area ratio of zero.

An example of a typical Contractions/Expansions tab display is shown below:



User Defined (Process Equipment) Tab:	The easiest way to model process equipment (for example, heat exchangers and filters) is as a section of pipe with a fitting loss coefficient. The pipe length needs to be short so that the pressure drop is solely due to the fittings – 1m is generally used. The values for mass flow, pressure drop etc. can be obtained from the process datasheet. Static head changes between inlet and outlet should not be taken into account as the node information deals with this. An example of a typical User Defined (Process Equipment) tab display is shown below:



Note. The K-value is automatically posted into the cell that is currently active when the K-value dialogue is closed. This value is pasted to the clipboard if the cell does not accept the value.

2.6.2.4. Pipe Roughness Calculator

This calculator is used to select a Surface Type and Absolute Roughness. Clicking on OK then adds the selection to the calculation by pasting the selected value into the cell that was active when the calculator was called.

The units of measurement for roughness depend on the program calling the Pipe Roughness Calculator. Those for PIPER are in mm. For other programs (for example, FLONET) the Pipe Roughness Calculator displays and returns the relative roughness based on the relevant pipe diameter.

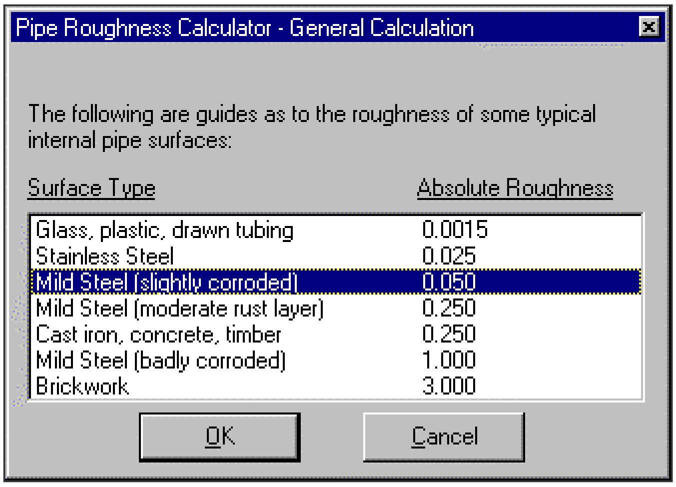


Figure 9 Pipe Roughness Calculator dialogue

2.6.2.5. Calculator

Displays the Windows calculator to assist with your calculations. On-line help is also available directly from this dialogue.

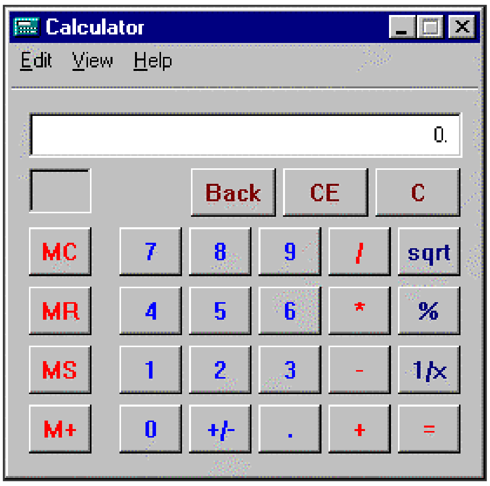


Figure 10 Calculator dialogue

2.6.2.6. Text Editor

Displays the Notepad text editor.

2.6.3. Options Menu

The Options menu allows you to access the following options:

- Calculation Options
- Show Calculation Progress.

2.6.3.1. Calculation Options

This allows additional calculation control parameters to be selected when a network is solved.

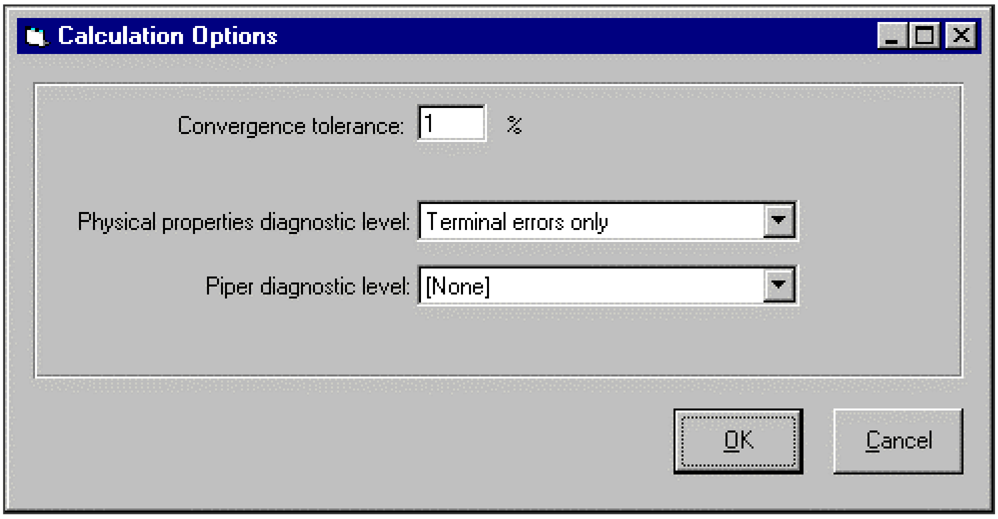


Figure 11 Calculations Option dialogue

The following selections can be made from this dialogue:

Usage

Convergence tolerance	Mandatory. Entering a value in this field adjusts the convergence tolerance. Considerable time can be saved by relaxing this tolerance and avoiding unnecessary iterations of the program when calculating flowrates especially if these are close to choked.	
Physical properties diagnostic level	Mandatory. The following levels can be selected:	
	Terminal errors only	Displays an output message only if a terminal error occurs (this is the default setting).
	All errors	Displays an output message if any terminal or non-terminal error occurs. This also provides a summary of violations of temperature and pressure ranges for physical properties.
	All errors, with details	Displays an output message if any terminal or non-terminal error occurs. This also provides a table of temperature and pressure ranges for physical properties together with details of all violations of the ranges.

Usage

Piper diagnostic level	Mandatory. The following levels can be selected: **None** – No diagnostic information displayed (this is the default setting). **Summary** – Provides a summary of the calculation through the system at each flowrate. **Detailed** – Provides a summary of the calculation through the system at each flowrate plus the inlet and outlet conditions for each section at each flowrate. **Complete diagnostics** – Provides a summary of the calculation through the system at each flowrate plus the inlet and outlet conditions for each section at each flowrate. Details of the course of the iterations in each section are shown as they proceed.

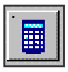
2.6.3.2. Show Calculation Progress

Selecting this option allows detailed output from the calculation to be viewed on-screen while it is still running. This slows down the running speed but also provides important information about the calculation.

Note. When selecting this option a window is displayed advising the user that this selection slows down the calculation time considerably.

2.6.4. Calculate Menu

The Calculate menu allows you to access the following option:



Menu Option	Definition
Calculate	To start a PIPER calculation either click on the calculate button (see left) or select ‘Calculate’ from the main PIPER menu bar.

2.6.4.1. Additional information on using the Calculate option

- When a PIPER calculation is started, the data entered is passed to the calculation module. This module runs separately from PIPER itself but the program informs the user that it is calculating. A 'Stop' button allows the calculation to be terminated as a complex and lengthy calculation (such as calculating the maximum non-choked flow) can take a significant amount of time.

- PIPER can be minimised during lengthy calculations such as a large piping system or when a high diagnostic output is specified. However, the PIPER calculation module is processor intensive and, even when it is minimised, other applications may run more slowly during the calculation.

2.6.5. Results Menu

The Results menu allows you to access the following options:

Menu Option	Definition
View Detailed Results	Displayed the detailed results on the screen (see section 3.2.7).
Print Detailed Results	Prints the detailed results on the default printer (see section 3.2.8).
Save Detailed Results	Allow the calculation results to be saved as an .OUT file (see section 3.2.10).

2.6.6. Help Menu

The Help menu allows access to the on-line help facility available with PIPER and the latest version information.

2.7. The PIPER Toolbar

The buttons on the toolbar allow the following options to be selected:

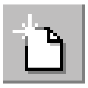
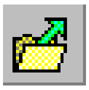
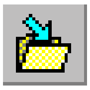
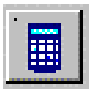
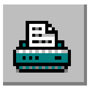
Button	Purpose
	**Create new PIPER file** Creates a new PIPER file. Equivalent File menu item – **New**
	**Open a PIPER file** Opens an existing (saved) PIPER file. Equivalent File menu item – **Open**
	**Save PIPER file** Saves the PIPER file currently open. Equivalent File menu item – **Save**
	**Calculate current problem** Solves the current network and displays the results in the Summary Results dialogue. Equivalent Calculate menu item – **Calculate**
	**Print summary of calculation results** Prints out the screen results including: • Results Report • Physical Properties Report • On-screen graphs. Equivalent menu item – N/A

Table 1 The PIPER Toolbar

3. PIPER Tutorial

3.1. General

This example details the procedure to calculate pressure drops and flowrates for a two-phase mixture (water and ethanol) in an unbranched pipe (see Figure 12).

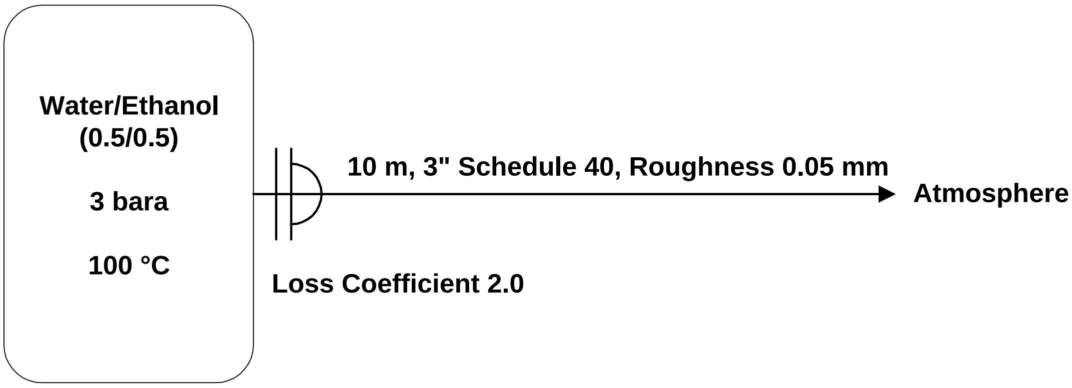


Figure 12 PIPER - Calculation example

3.2. Solving the Network

The procedure to solve the network involves the following steps:

- Defining the physical properties using the PhysPack program
- Accessing PIPER
- Entering Conditions data
- Entering the Physical Properties data defined in PhysPack
- Entering Pipe data
- Solving the Network.

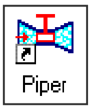
The results can be viewed, printed and saved after the network has been solved.

3.2.1. Defining a Physical Properties file

Use the PhysPack program to define a physical properties .PPB file for water and ethanol (see Appendix A for the procedure).

3.2.2. Accessing PIPER

Procedure



1. Access PIPER by selecting **Start | Programs |PEL |** then click on the **PIPER** icon (see left).

2. A splash screen showing the program name and version number appears briefly before the start up screen (see Figure 1) is displayed.

3.2.3. Entering Conditions data

Procedure

1. Select the ‘Inlet & exit pressure’ button on the ‘Conditions’ tab.

2. Click on the ‘Flowrate (estimate 1)’ box and enter the value **20**.

3. Click on the ‘Inlet Pressure’ box and enter the value **3 bara**. (PIPER automatically converts the units - see Appendix B for more information).

4. Click on the ‘System Back Pressure’ box and enter the value **1 atm**.

5. Click on the ‘Inlet Temperature’ box and enter the value **100 C**.

3.2.4. Entering Physical Properties data

Procedure

1. Select the ‘Physical Properties’ tab.

2. Click on the ‘Browse …’ button and select the .PPB physical properties file defined in PhysPack (see section 3.2.1 for more information and Appendix A for the procedure to produce the .PPB file).

3. Information from the .PPB file is loaded into PIPER and displayed in table form including a list of components (see Figure 13).

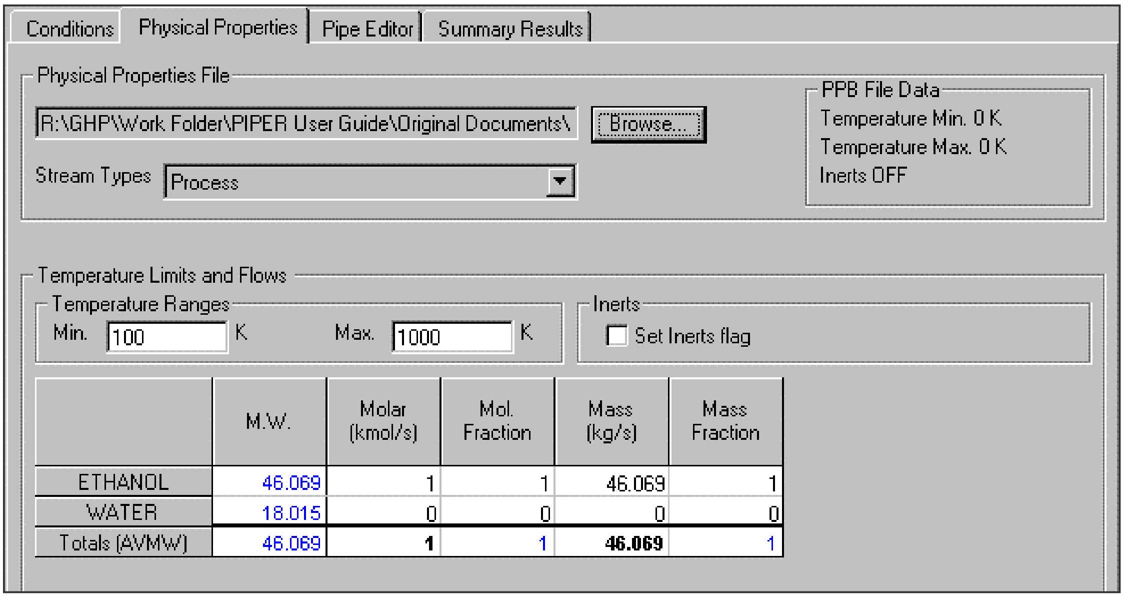


Figure 13 Physical Properties Editor showing .PPB file data

4. Click on the ‘Mass Fraction’ field and change the values for ethanol and water to **0.5**. The ‘Totals (AVMW)’ box below now displays 1.

Note. PIPER normalises the value to 1 if they do not add up.

5. Go to the ‘Temperature Limits and Flows’ panel and click on the Temperature Ranges ‘Min.**’** box. Enter the value **300**.

6. Click on the ‘Max.’ box and enter the value **500**.

3.2.5. Defining the Pipe fittings using the Pipe Editor

The vessel outlet is represented as a contraction from a nominally large diameter such as 1 m to the pipe diameter.

Procedure

1. Select the ‘Pipe Editor’ tab.

2. Click on the ▼ arrow next to the ‘Type’ box and select **Contraction** from the list.

3. Click on the ‘Upstream pipe diameter’ box and enter the value **1**.

4. Click on the ‘Downstream pipe diameter’ box. Go to the ‘Tools’ menu and select the ‘**Pipe Inner Diameter Calculator’**.

5. Select a ‘3″ Schedule 40’ pipe and click on **OK**. PIPER pastes the internal diameter into the ‘Downstream pipe diameter’ box.

6. Click on the ‘Upstream pipe roughness’ box. Go to the ‘Tools’ menu and select the **Pipe Roughness Calculator**.

7. Select ‘Mild Steel (slightly corroded)’ and click on **OK**. PIPER pastes the value into the ‘Upstream pipe roughness’ box.

8. Repeat steps 6 and 7 for the ‘Downstream pipe roughness’ box.

9. Click on the **Add** button. The text ‘Contraction’ is added to the right-hand box.

10. The Pipe Editor now looks like this:

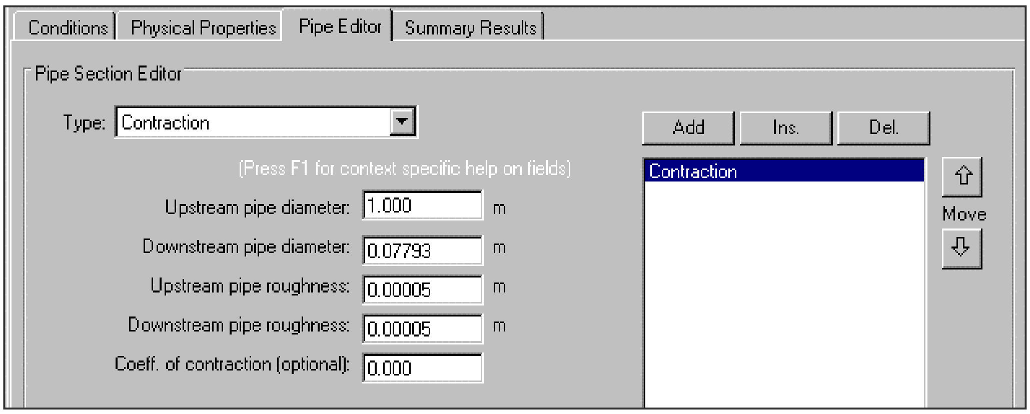


Figure 14 Pipe Editor showing Contraction values

Adding the Bursting Disc

The Bursting Disc is modelled using an orifice with an approximate loss coefficient to represent the fitting.

11. Click on the ▼ arrow next to the ‘Type’ box and select **Orifice** from the list. Notice that PIPER has inserted the current pipe diameter for the upstream pipe and orifice diameters.

12. Click on the ‘Orifice diameter’ box and enter the value **0.07** (this represents 90% of the orifice diameter).

13. Click on the ‘K-value for orifice’ box and enter the value **2.0**.

14. Click on the **Add** button. The text ‘Orifice’ is added below ‘Contraction’ in the right-hand box.

15. The Pipe Editor now looks like this:

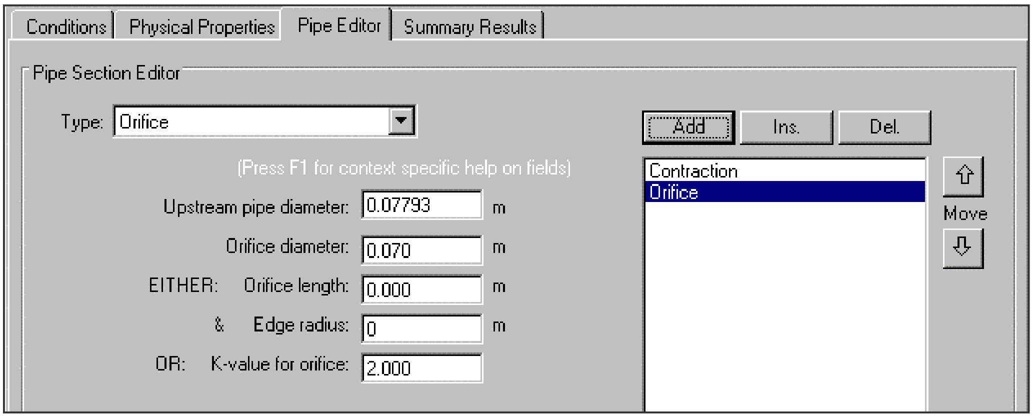


Figure 15 Pipe Editor showing Orifice values

Adding the Straight

The 10 meter straight pipe is now added to the model.

16. Click on the ▼ arrow next to the ‘Type’ box and select **Straight** from the list. Notice that PIPER has again inserted the current pipe diameter and roughness as default values.

17. Click on the ‘Pipe length’ box and enter **10**.

18. Click on the **Add** button. The text ‘Straight’ is added to the list below ‘Orifice’ in the right-hand box.

Adding the Enlargement

An enlargement is now added to model the pipe exit to the atmosphere. This is modelled by an expansion to a nominally large diameter such as 1 m.

19. Click on the ▼ arrow next to the ‘Type’ box and select **Enlargement** from the list. Notice that PIPER has inserted suitable default values to the ‘Upstream/Downstream pipe diameter’ and ‘Upstream/Downstream pipe roughness’ boxes.

20. Click on the **Add** button. The text ‘Enlargement’ is added to the list in the right-hand box.

21. The Pipe Editor now looks like this:

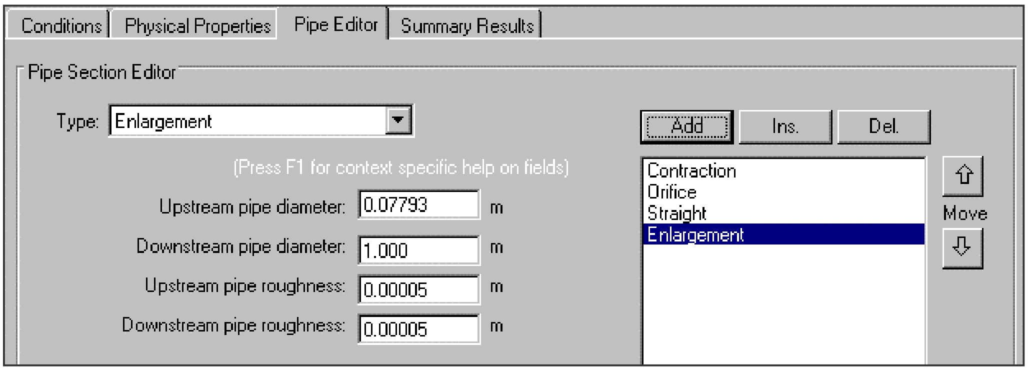
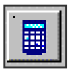


Figure 16 Pipe Editor showing final values

3.2.6. Calculating the Network Solution

Procedure



1. Click on the **Calculate** button (see left) on the PIPER toolbar or select the **Calculate** menu option.

2. A message ‘Calculating – Please wait’ is displayed.

3. After a few seconds the ‘Summary Results’ tab is activated and displays the results of the calculation as shown in Figure 17:

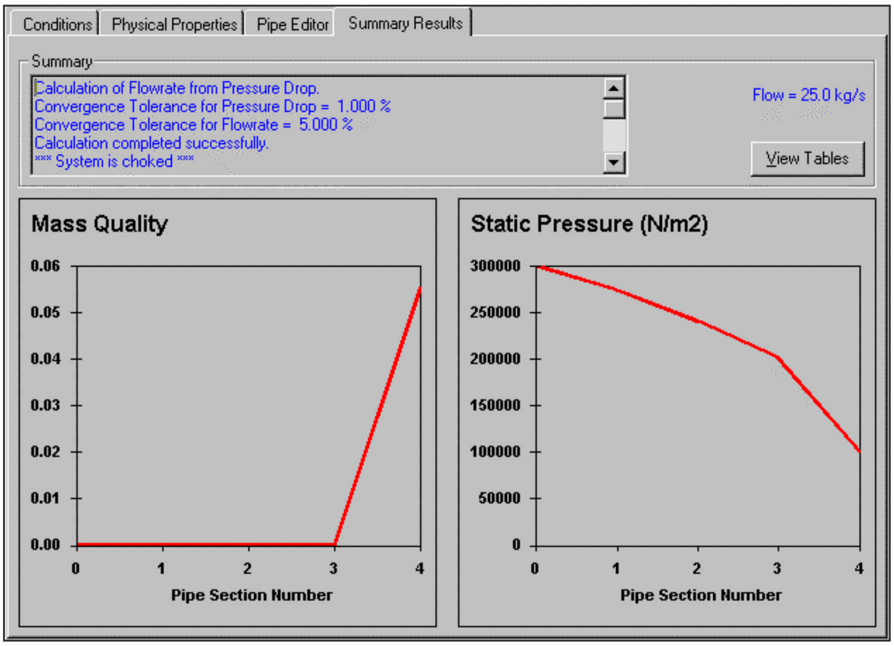


Figure 17 Summary Results dialogue

3.2.7. Viewing Detailed Results

Selecting the ‘View Detailed Results’ menu option from the ‘Results’ menu displays the results on a NotePad screen.

3.2.8. Printing Results

Results can be printed using the following methods:

3.2.8.1. Print Detailed Results menu option

Selecting the ‘Print Detailed Results’ menu option from the ‘Results’ menu prints the contents of the NotePad screen viewed in section 3.2.7 showing the results of the problem that has just been run.

The report contains detailed information under the following headings:

- Details of Conduit System
- Details of Input Stream
- Summary of Calculation
- Details of Output Stream
- Conditions of the end of each conduit
- Physical Properties at the end of each conduit

- Violations of temperature and pressure ranges for physical properties.

Note. The printer should be set to 'Landscape' page mode when printing this report.

3.2.8.2. Print Summary of Calculation Results

This button (see left) prints the results data in the form of:

- Results Report
- Physical Properties Report
- Graphs.

The following show examples of these:

Results

Pipe Section	Section Type	K value or Length m	Static Pressure N/m2	Pressure Drop N/m2	Temperature K	Mass Quality	Homogenous Velocity m/s	Pressure Discontinuity
0	Feed	-	300000	0	373.15	0	0.038914	-
1	Contraction	.58613 CC	274015	25984.8	373.144	0	6.40762	-
2	Orifice	2.00000 KV	240431	33584.4	373.144	0	6.40762	-
3	Straight	10	201679	38751.4	373.144	0	6.40762	-
4	Expansion	0	101325	100354	355.44	0.055058	1.5288	Yes

Figure 18 Results Report

Physical Properties

Pipe Section	Section Type	Straight Pipe Void Fraction	Liquid Mol Wt	Vapour Mol Wt	Liquid Density kg/m3	Vapour Density kg/m3	Liquid Viscosity N.s/m2	Vapour Viscosity N.s/m2	Liquid Enthalpy J/kg	Vapour Enthalpy J/kg	Fail Flag
0	Feed	0	25.9014	1	817.977	0	0.000294	0	-1404947	0	
1	Contraction	0	25.9014	1	817.983	0	0.000294	0	-1404968	0	
2	Orifice	0	25.9014	1	817.983	0	0.000294	0	-1404968	0	
3	Straight	0	25.9014	1	817.983	0	0.000294	0	-1404968	0	
4	Expansion	0.58971	25.5563	33.7157	837.961	1.17392	0.000365	1.1E-005	-14091718	84243.8	

Figure 19 Physical Properties Report

3.2.9. On-screen Graphs

On-screen graphs are produced in the following format:

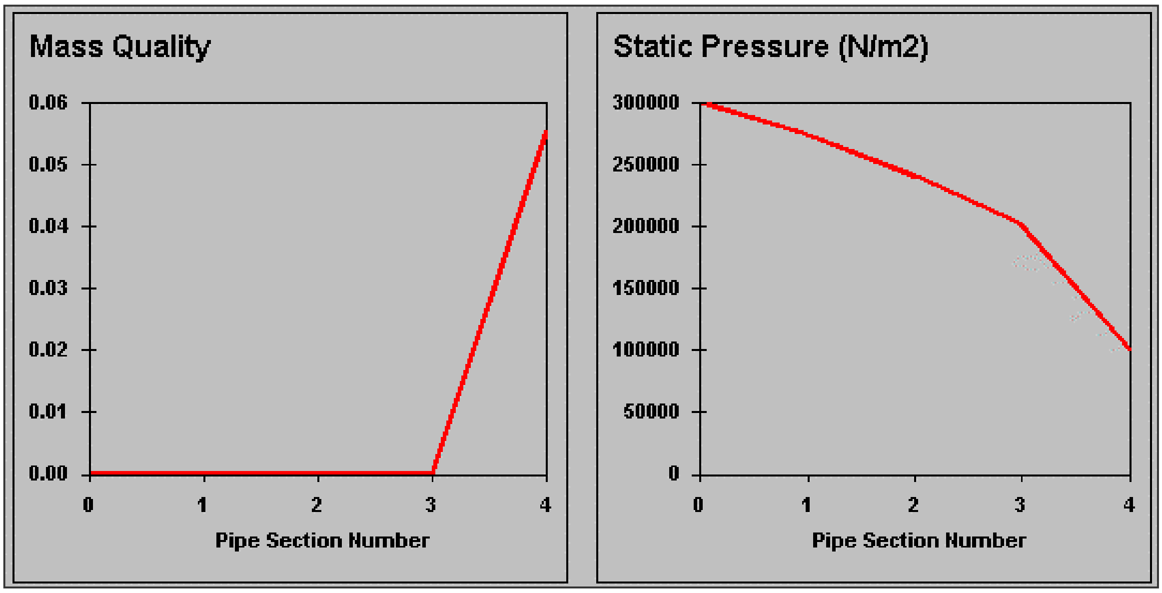


Figure 20 Results Graphs

3.2.10. Saving Detailed Results

Procedure

1. Click on the ‘Save Detailed Results’ menu option from the ‘Results’ menu. The ‘Save Results File As’ dialogue is displayed.

2. This saves the results as a *.out text file. This can then be edited using any text editor or Lotus 1.2.3.

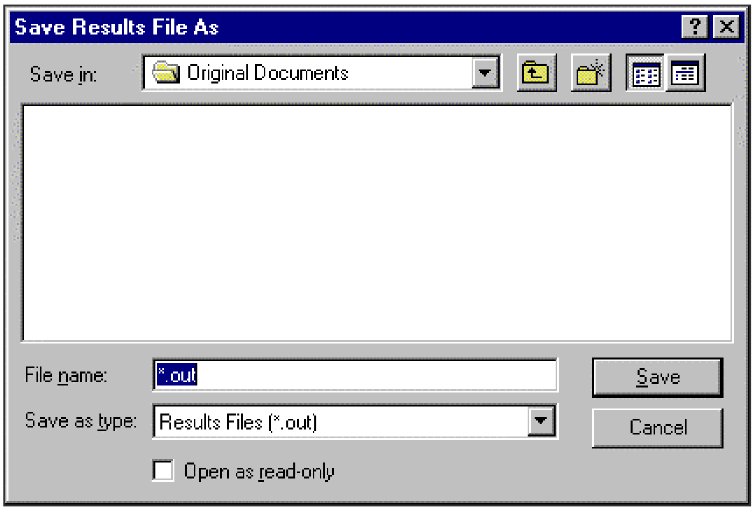


Figure 21 Save Results dialogue

Appendices

Appendix A – Creating a .PPB file

The following text is adapted from the 60 Second Guide to Physical Property Files.

Procedure

1. Click on **Start | Programs | PEL | PhysPack**. The main PhysPack window appears.

2. There are two tabs – one for **Data Selection** and one for **Calculation**. The one for Data Selection is selected by default. The **Data Selection** tab has two panels. The one on the left is a tree view that resembles Microsoft Explorer. Under **Process,** there are two branches – **Components** and **Properties**.

   The first thing we need to do is add the components.

3. Click on **Components** in the tree view. The right hand panel changes to the Components tab. In the **Search for Name** box, type **ETHA** for ethanol. All the components on the databank beginning with ETHA are then listed below. Double-click on Ethanol to add it to the tree. Next, type **W** for water and double click.

   Next, we need to add the properties that PIPER requires.

4. Click on the box near the top of the screen containing the text **select a program to select the required properties**.

5. Select **PIPER – Low Pressure** from the pull-down list. PhysPack will automatically add the properties that PIPER requires to the **Properties** in the tree view.

   PhysPack will have chosen a default method to calculate each property. Let's see which ones it has selected.

6. Click on the property **Vapour Liquid Equilibrium** in the tree view. The right hand panel changes to the Methods panel. You can see that the IDEAL method has been selected.

   But is this sensible for a mixture of ethanol and water? Let's ask the PhysPack Advisor.

7. Click on the **Method Advice Wizard** button on the toolbar. The wizard will then appear. When you've read the introduction, click **Next** . The two key components are set to ethanol and water. Click **Next** again and set the system pressure to **Low**. Click **Next** again and you will see that all the binary interaction parameters are present to model the liquid phase by either the WILSON E and UNIQUAC methods. Follow the advice and select WILSON E. When you click **Next**, you will see that the Advisor suggests IDEAL for the vapour phase and WILSON E for the liquid. Follow his advice and let him make the changes by clicking **Finish** and then **Yes** to Add Methods?

   That's it for Vapour Liquid Equilibrium. Now let's check the methods for the other properties.

8. Click on each of the other properties in turn. As you do so, look in the Method panel on the right and check that **All Data** is present in the Data Status column for the selected method. Note that for the properties VAPOUR DENSITY and RATIO OF SPECIFIC HEATS **Miss Binary** is reported. Binary interaction parameters are desirable for these properties but RKS will give a reasonable approximation without them so it's okay if they are missing.

   That's all there is to preparing the PPB file but before we save it it's a good idea to do some calculations to check everything is okay.

9. Click on the **Calculation** tab near the top left hand side of the screen. Two new panels appear – the one on the left is a vertical menu and the one on the right is the input screen for the selected menu item.

   The first thing to do is to select a Mixture (Standard) type of calculation

10. Click on **Conditions** on the left, then in the right hand panel click on the down arrow in the **Calculation Type** box and select **Mixture (Standard)**.

    Next we need to set the feeds for each component – 50% ethanol, 50% water by mass fraction

11. Click on **Feed** on the left, then in the right hand panel click on the spreadsheet twice – once to give it focus and again to select the mass fraction column. Type **0.5 for ethanol, 0.5 for water**.

    Now we're ready to do the calculation and see the results so …

12. Click on **Calculate** on the left. After a few seconds PhysPack presents the results in a spreadsheet on the right.

    Note that the values in the Stream Vapour Fraction columns are in blue. This is because the temperatures and pressures are below the accuracy limits for the WILSON E binary interaction parameters. This is not important for this example.

    So now we are ready to export the PPB file.

13. Click on the **File** menu and select **Export**. The *Export a PPB File* window appears. Select the **C** drive and save it as **PIPER.PPB**

Appendix B – In-cell Units Conversion

Units conversion is available for any field where it is appropriate. The conversion is always from the units entered to the units shown on the dialogueue box.

In order to perform a units conversion you should type the following information into a field:

Value_Unit

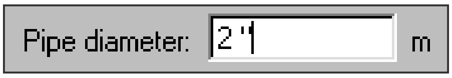
where:

Value is the figure to be converted
Unit is the unit to be converted
'_' is a space.

The units conversion (if it is possible) is performed automatically as soon the cursor is moved from the current field, as shown below:

Example

1. Enter the value and unit **2 ″** (inches).



2. Move the cursor off the field or press **Enter**. The field now displays the following:

3. If the conversion is not possible a message box is displayed and the value in the field is taken as being in the units specified on the dialogue box.

Appendix C – Choosing a Fluid Flow Program

Eutech PEL contains four programs which are capable of fluid flow calculations. These are:

- PEW
- ADRIAN
- FLONET
- PIPER

The capabilities of each of these programs are summarised below and in Figure 22:

PEW

This program can handle:

- A single, unbranched pipe of a fixed diameter.
- Incompressible and compressible flow calculations. Compressible flow calculations are valid up to approximately 0.3 Ma and include isothermal and adiabatic modes.
- Single phase gas or liquid.
- Design calculation as well as rating calculations, that is, for a fixed flow and required pressure drop it will calculate the pipe diameter.

ADRIAN

This program can handle:

- Rating calculations in branched networks of high velocity gas flows.
- Single phase gas flows up to 1.0 Ma including modelling of choked systems.
- Isothermal and adiabatic calculations.
- Directed piping network (that is, the fluid flow from the beginning of the pipe to the end of the pipe) allowing for changes in diameter and full consideration of the order in which fittings occur.
- Loop-like as well as tree-like networks with care.

Note. ADRIAN is only valid for fully turbulent flow regimes.

FLONET

This program can handle:

- Rating calculations in complex networks of single phase gas or liquids.
- Isothermal calculations only and gas flow below 0.2 Ma.
- Complex networks can be modelled including loop-like structures with unknown direction.
- Turbulent, transitional and laminar flow regimes.
- Pumps and None-Return Valves can be included in the network.

PIPER

This program can handle:

- Single phase gas or liquid and two phase flow.
- Single unbranched pipe of variable diameter.
- Very detailed physical properties representation allowing modelling of phase change throughout the piping system and modelling of non ideal gases.
- Allows for heat transfer through the pipe wall.
- Fully models all pressure discontinuities and calculate maximum non-choked flow in a system.

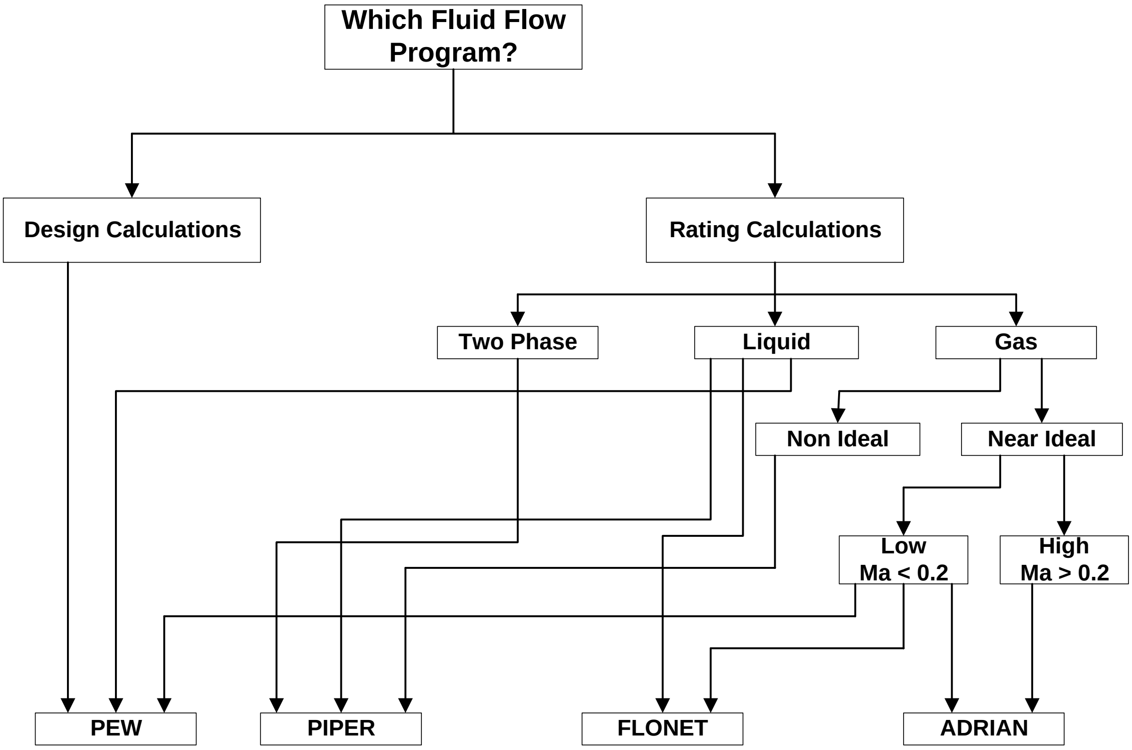


Figure 22 Choosing a Fluid Flow Program

Appendix D – Troubleshooting

Refer to the following points if you are experiencing problems when running PIPER.

- PIPER does not contain a large list of different fitting types. This can be overcome by using an orifice model with an appropriate loss co-efficient to represent missing fittings.
- Inlets from a vessel can be modelled by a contraction from an arbitrarily large inlet diameter (suggested size of 1 m) down to the inlet pipe diameter.
- Similarly, outlets to atmosphere can be modelled by an expansion from the outlet pipe diameter up to an arbitrarily large diameter (suggested size of 1 m).
- Incorrect answers are most commonly caused by poor physical properties representation or by incorrectly specified pipe systems. Always check the .PPB file in PhysPack (or similar program) over the range of conditions present within the piping system to ensure that it gives an accurate representation of the fluid in the system.
- Remember that PIPER requires input in strict SI units. Use the in-cell units conversion feature (see Appendix B for more information) to enter inputs in normal engineering units.
- Be aware of 'non-fatal' errors where the program appears to produce results without a problem but these results not match your system. Causes could be:

On the Input side:	These include cases where the piping system has been inadequately described, that is, failure to specify a pipe length for a straight section or a correct bend radius for a bend.
On the Output side:	These include cases where the solution involves an exit pressure less than atmospheric when the system actually does exhaust into the atmosphere.

Note. The system will always choke if you try to specify a system that involves full vacuum.

Notes

Use these pages to record any notes.

Notes