

Chapter 2

FLUID FLOW IN PIPES PUMPS AND COMPRESSORS

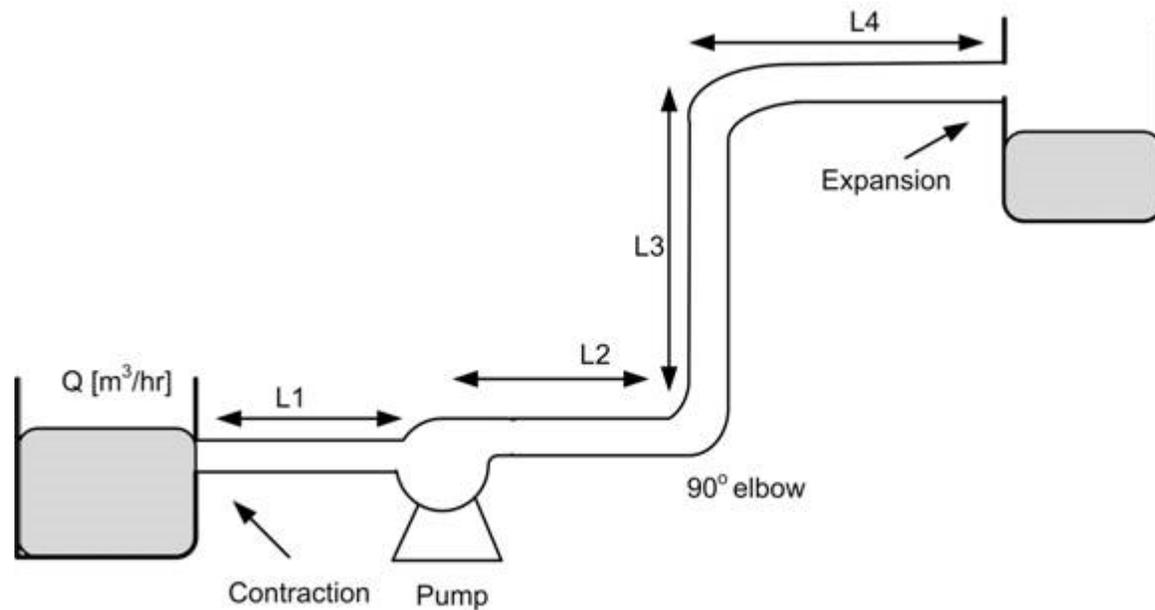
At the End of This Chapter You Should Be Able to

- Fully understand the type of flow regime in pipes, pumps and compressors,
- Perform hand calculations and verify their results with Hysys, Provision and Aspen simulation software package.
- Students will be able to determine pressure drop in pipeline, inlet pipe flow rate and pipe length.
- Determine the useful power input needed to overcome the friction losses in a pipeline. Calculate brake kW for pumps and compressors.

Flow in pipes

- In this section a brief summary of the most general form of Bernoulli equation for steady incompressible flows is introduced.
- Bernoulli equation is composed of kinetic energy, potential energy and internal energy.
- The energy equation is written between the inlet at point 1, and the exit of the pipeline at point 2, (Figure 2.1).
- The process flow diagram contains, sudden contraction (Exit of the first tank to inlet of pipe 1), two 90° elbows, sudden expansion (exit of pipe 4 and inlet of the second tank).

Figure 2.1 Process flow diagram of piping system



Energy equation

The energy equation for incompressible fluids,

$$\frac{P_1}{\rho} + gz_1 + \frac{V_1^2}{2} = \frac{P_2}{\rho} + gz_2 + \frac{V_2^2}{2} + W_s + \Sigma F \quad (2.1)$$

where P_1 is the pressure at point 1 and P_2 is the pressure at point 2, ρ is the average fluid density, z_1 is the height at point 1, z_2 the height at point 2, V_1 is the inlet velocity, V_2 is the exit velocity, W_s is shaft work, and the summation of friction losses is ΣF . The friction loss is due to pipe skin friction, expansion losses, contraction losses and fitting losses. The summation of friction loss can be calculated using equation (2.2),

$$\Sigma F = f \frac{L}{D} \frac{V_2^2}{2} + (K_{\text{exp}} + K_c + K_f) \frac{V_2^2}{2} \quad (2.2)$$

Where K_{exp} is the expansion loss, K_c is the contraction loss, and K_f is the fitting losses. Fitting losses includes losses due to elbows (K_e), tees (K_T), and Globe valves (K_G).

Expansion loss (K_{exp}) is determined using equation (2.3),

$$K_{\text{exp}} = \left(1 - \frac{A_1}{A_2}\right)^2$$

Fittings

where A_1 and A_2 are the cross sectional area at inlet and exit, respectively. The contraction loss (K_c) is calculated using equation (2.4),

$$K_c = 0.55 \left(1 - \frac{A_2}{A_1} \right) \quad (2.4)$$

For turbulent flow, $K_e=0.75$ (for 90° elbow), $K_T= 1.0$ (Tee), $K_G=6.0$ (Globe valve), $K_C=2.0$ (Check valve). For horizontal pipe with same inlet and exit diameter and incompressible fluid, $V_1 = V_2$. To calculate the pressure drop between the inlet and exit of a horizontal pipe, first calculate the average velocity then, use Reynolds number to determine the flow regime (i.e. laminar, transient or turbulent). The average velocity can be expressed in terms of the flow rate as,

$$V = \frac{Q}{A_c} = \frac{Q}{\pi D^2/4} \quad (2.5)$$

Where, V is the average velocity, A_c is the pipe inner cross sectional area, Q is the inlet fluid volumetric flow rate and D is the pipe inner diameter. The Reynolds number Re from which the flow regime can be calculated by:

$$Re = \frac{\rho VD}{\mu} \quad (2.6)$$

2.1.1 Laminar flow

In fully developed laminar flow ($Re < 4000$) in a circular horizontal pipe, the pressure loss and the head loss are given by

$$\frac{\Delta P}{\rho} = \frac{P_1 - P_2}{\rho} = \Delta P_L = f \frac{L}{D} \frac{V^2}{2} \quad (2.7)$$

The friction factor,

$$f = \frac{64}{Re} \quad (2.8)$$

Under laminar flow conditions, the friction factor, f , is directly proportional to viscosity and inversely proportional to the velocity, pipe diameter and fluid density. The friction factor is independent of pipe roughness in laminar flow because the disturbances caused by surface roughness are quickly damped by viscosity. The pressure drop in laminar flow for circular horizontal pipe,

$$\Delta P_L = \frac{32\mu LV}{D^2} \quad (2.9)$$

When the flow rate and the average velocity are held constant, the head loss becomes proportional to viscosity. The head loss, h_L , is related to pressure loss by

$$h_L = \frac{\Delta P_L}{\rho g} = \frac{32\mu 2\mu}{\rho g D^2} \quad (2.10)$$

2.1.2 Turbulent flow

- When the flow is turbulent, the relationship becomes more complex and is best shown by the graph because the friction factor is a function of both Reynolds number and roughness.
- The degree of roughness was designated as the ratio of the sand grain diameter to the pipe diameter (ε / D).

Reynolds number

- The relationship between the friction factor and Reynolds number can be determined for every relative roughness. From these relationships, it is apparent that for rough pipes; the roughness is important in determining the magnitude of the friction factor.
- At high Reynolds number the friction factor depends entirely on roughness and the friction factor can be obtained from the rough pipe law.
- In fully developed turbulent flow ($Re > 4000$) in a circular pipe, the pressure drop for turbulent flow is,

$$\Delta P = \Delta P_L = f \frac{L}{D} \frac{\rho V^2}{2}$$

Friction factor

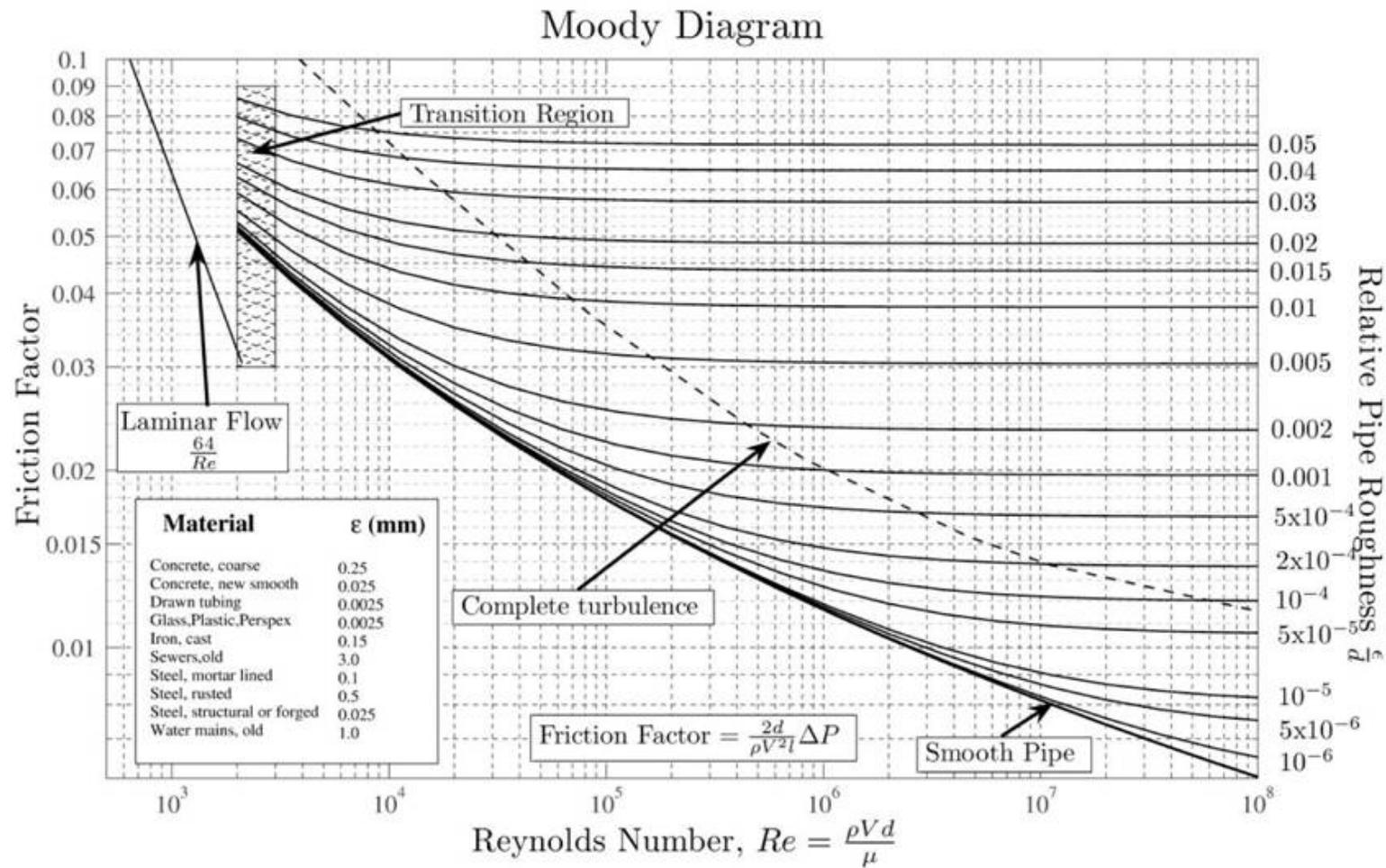
- The friction factor, f , can be found from the Moody diagram (Figure 2.2) which is based on the Colebrook equation in the turbulent regime (equation 2.12).

$$\frac{1}{\sqrt{f}} = -2 \log \left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re} \sqrt{f}} \right)$$

Alternatively, the explicit equation for the friction factor derived by Swamee and Jain (equation 2.13) can be solved for the absolute roughness.

$$f = \frac{0.25}{\left[\log \left(\frac{\varepsilon}{3.7D} + \frac{5.74}{\text{Re}^{0.9}} \right) \right]^2}$$

Figure 2.2 Moody Diagram (with permission from: S. Beck and R. Collins, University of Sheffield)



Head loss

- Head loss for turbulent flow, h_L

$$h_L = \frac{\Delta P_L}{\rho g}$$

- The relative roughness of the pipe is ε / D , where ε is pipe roughness and D is the inner diameter of the pipe. The friction factor can be determined from the Moody diagram (Figure 2.2).
- The useful power input is the amount needed to overcome the frictional losses in the pipe.

$$W_{\text{pump}} = Q\Delta P$$

Example 2.1 Pressure drop in a horizontal pipe

Water is flowing in a 10 m horizontal smooth pipe at 4 m/s and 25 °C. The density of water is 1000 kg/m³ and the viscosity of water is 0.001 kg/(m s). The pipe is Schedule 40, 1 in nominal diameter (2.66 cm ID). Water inlet pressure is 2 atm. Calculate pressure drop in the pipe with hand calculations and compare results with those obtained with Hysys, PROII and Aspen softwares.

Solution(Hand Calculations)

- Reynolds number is calculated to determine the flow regime

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(1000 \text{ kg/m}^3)(4 \text{ m/s})(0.0266 \text{ m})}{0.001 \text{ kg/m} \cdot \text{s}} = 1.064 \times 10^5$$

- Since Reynolds number is greater than 4000, the flow is turbulent. The relative roughness of the smooth pipe is:

$$\varepsilon / D = \frac{0}{0.04 \text{ m}} = 0$$

- The friction factor, f , can be determined from the Moody chart (Figure 2.2) or Swamee and Jain alternative equation,

$$f = \frac{0.25}{\left[\log \left(\frac{0}{3.7D} + \frac{5.74}{(1.064 \times 10^5)^{0.9}} \right) \right]^2} = 0.0176$$

Calculation of friction factor

- The calculated friction factor, $f = 0.0176$. Then the pressure drop,

$$\Delta P = P_1 - P_2 = \Delta P_L = f \frac{L}{D} \rho \frac{V^2}{2}$$

$$\Delta P = (0.0176) \frac{10 \text{ m}}{0.0266 \text{ m}} \frac{(1000 \text{ kg/m}^3)(4 \text{ m/s})^2}{2} \left(\frac{1 \text{ kN}}{1000 \text{ kg} \cdot \text{m/s}} \right) \left(\frac{1 \text{ kPa}}{1 \text{ kN/m}^2} \right) = 52.93 \text{ kPa}$$

- The head loss, h_L

$$h_L = \frac{\Delta P_L}{\rho g} = f \frac{L}{D} \frac{V^2}{2g} = 0.0176 \frac{10 \text{ m}}{0.0266 \text{ m}} \frac{(4 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} = 5.13 \text{ m}$$

- The volumetric flow rate, Q

$$Q = VA = (4 \text{ m/s}) \left(\frac{\pi(0.0266 \text{ m})^2}{4} \right) = 2.22 \times 10^{-3} \text{ m}^3 / \text{s}$$

Input power

- The power input needed to overcome the frictional losses in the pipe,

$$\dot{W}_{\text{pump}} = \Delta P = (2.22 \times 10^{-3} \text{ m}^3 / \text{s})(50.4 \text{ kPa}) \left(\frac{1 \text{ kW}}{1 \text{ kPa} \cdot \text{m}^3 / \text{s}} \right) = 0.12 \text{ kW}$$

- Therefore, useful power input in the amount of 0.12 kW is needed to overcome the frictional losses in the pipe.

Hysys Simulation

- In Hysys, the pipe segment in the object palette offers three calculation modes, pressure drop, flow rate and pipe length. The appropriate mode will automatically be selected depending on the information supplied. The Hysys simulation of fluid flowing in a pipe is simulated as follows:
- Start a new case in Hysys.
- Use SI units, from Tools menu, *Preferences* then *Variables*.
- Choose water as the component flowing in the pipe, and ASME STEAM as Property packages.
- Click *Enter the simulation Environment*.
- Select a material stream by double clicking on the blue arrow from the top of the object palette.

Volumetric flow rate

- Fill in the stream Name: Feed.
- Specify the volumetric feed rate, Q , based on the velocity of 4 m/s, and inner pipe diameter of 0.0266 m,

$$Q = A_c \times V = \frac{\pi(0.0266\text{ m})^2}{4} \times \frac{4\text{ m}}{\text{s}} \left(\frac{3600\text{ s}}{1\text{ hr}} \right) = \frac{8.03\text{ m}^3}{\text{hr}}$$

- Enter values for feed pressure, temperature and volumetric flow rate (Figure 2.3).
- In the composition menu; type in the mole fraction as 1 for water.

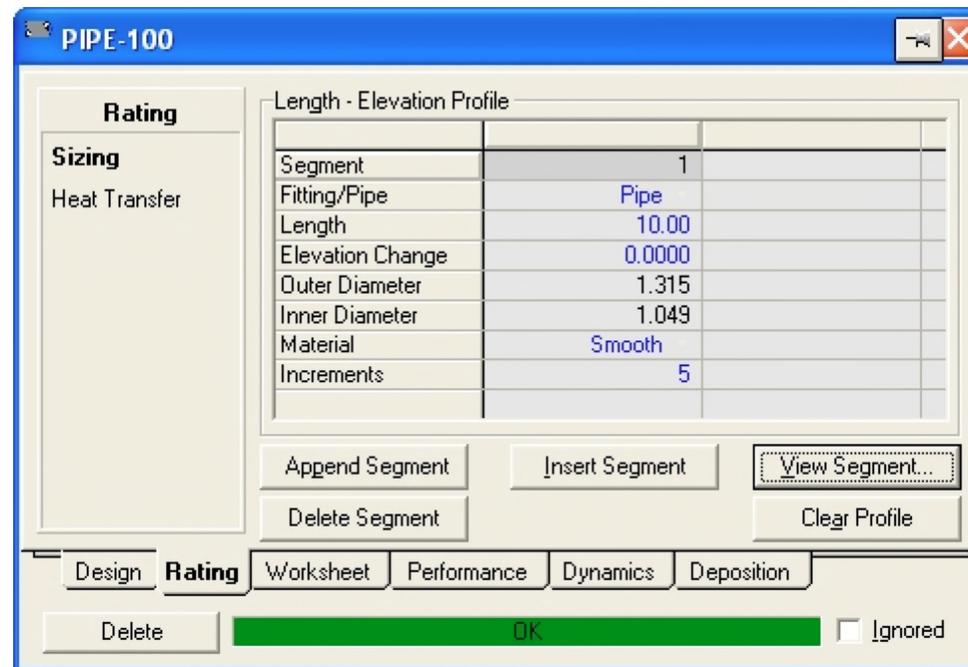
Figure 2.3 Inlet stream conditions for example 2.1

Worksheet	Stream Name	Inlet
Conditions	Vapour / Phase Fraction	0.0000
Properties	Temperature [C]	25.00
Composition	Pressure [kPa]	202.6
K Value	Molar Flow [kgmole/h]	443.3
User Variables	Mass Flow [kg/h]	7987
Notes	Std Ideal Liq Vol Flow [m3/h]	8.003
Cost Parameters	Molar Enthalpy [kJ/kgmole]	-2.850e+005
	Molar Entropy [kJ/kgmole-C]	6.613
	Heat Flow [kJ/h]	-1.264e+008
	Liq Vol Flow @Std Cond [m3/h]	7.995
	Fluid Package	Basis-1

Hysys simulation

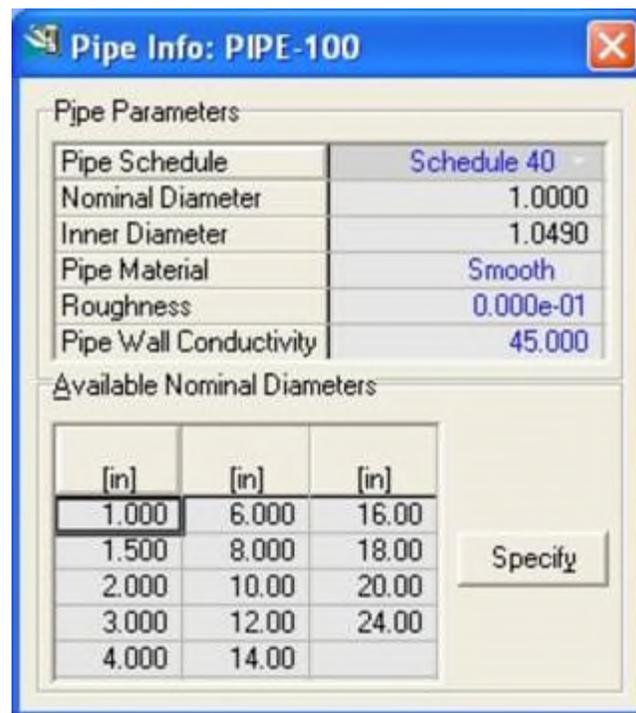
- Add the pipe segment by double clicking on the pipe segment in the object palette.
- Click on the *Rating* tab and click *Add Segment* button. The pipe length is 10 m, specify the Pipe Material as “smooth” by choosing this value from the drop down list see Figure 2.4

Figure 2.4 Length and elevation profile of the pipe in example 2.1.



Selection of pipe nominal diameter

- Click on the *View Segment* button and select Schedule 40. Then click on the nominal diameter entry and select the 1 inch diameter. To choose one of the options – click on 25.4 mm (1in) and press the *Specify* button as you see in this figure:



Workbook

- ❖ Double click on the product stream and enter 25°C for the temperature of the product stream (isothermal operation).
- ❖ To display the steam summary table below the process flow diagram in the PFD area, click the *Workbook* icon in the toolbar.
- ❖ Once the *Workbook* appears, from the workbook menu, click the *Setup* command, and then click the *Add* button in the workbook tabs group and select the variable needs to appear in the table.
- ❖ Right click in the PFD area below the process flow diagram and click on *Add Workbook Table*,

Table 2.1. Roughness Factors used by HYSYS

Pipe Material Type	Absolute Roughness (ϵ), m
Smooth	0.0
Drawn Tube	1.52×10^{-6}
Mild Steel	4.57×10^{-5}
Asphalted Iron	1.22×10^{-4}
Galvanized Iron	1.52×10^{-4}
Cast Iron	2.59×10^{-4}
Smooth riveted steel	9.14×10^{-4}
Rough riveted steel	9.14×10^{-3}

Figure 2.6 Pipe process flow sheet

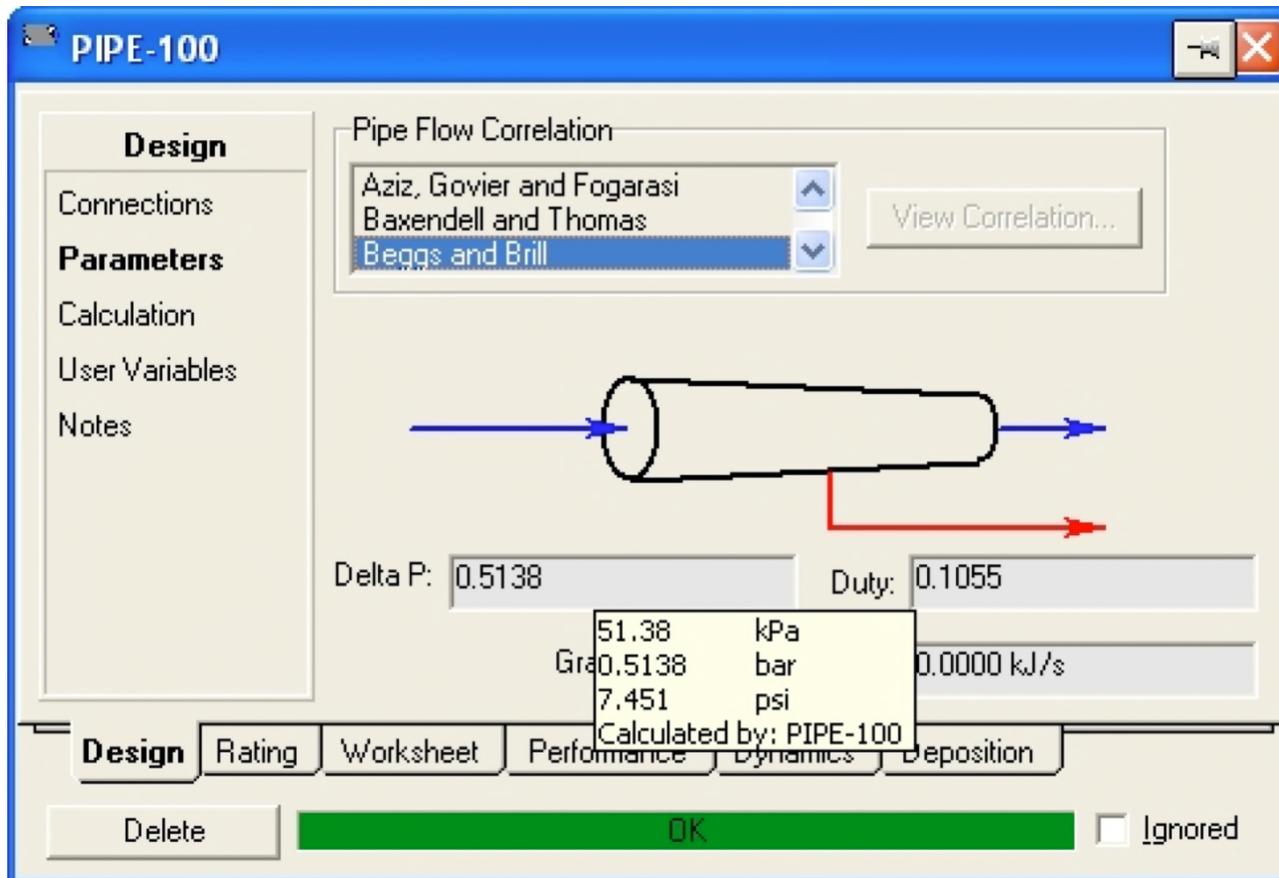
- The result looks like the figure 2.6



Streams			
		Inlet	Outlet
Temperature	C	25.00	25.00
Pressure	kPa	202.6	151.3
Mass Flow	kg/h	7987	7987
Comp Mass Frac (H2O)		1.0000	1.0000

Figure 2.7 Pipe Delta P calculated

- While in Design page, click on the parameters, the Hysys calculated pressure drop is 51.69 kPa (see Figure 2.7



PRO/II Simulation

- Open new case in Pro/II, click on the component selection icon (the benzene ring in tool bar), select water then select the steam fluid package.
- Click on the pipe segment in the object palette and then click anywhere in PFD area to place the pipe.
- Click on Stream icon in the object pallet then generate inlet (S1) and exit stream (S2).
- Double click on stream S1 and specify inlet temperature and pressure feed stream (Figure 2.8).

Figure 2.8 Inlet stream conditions

The screenshot shows the 'PRO/II - Stream Data' dialog box. At the top, there are menu options: 'UOM', 'Range', 'Help', 'Tag', 'Overview', 'Status', and 'Notes'. The 'Stream' field is set to 'S1' and the 'Description' field is empty. Below this, 'To Unit' is set to 'P1'. The 'Stream Type' section has a list box with 'Composition Defined' selected, and three buttons: 'Flowrate and Composition...', 'Stream Solids Data...', and 'Stream Polymer Data...'. The 'Thermal Condition' section has 'First Specification' set to 'Temperature' with a value of '25.00 C' and 'Second Specification' set to 'Pressure' with a value of '202.65 kPa'. The 'Thermodynamic' section is set to 'Determined From Connectivity'. At the bottom, there are 'OK' and 'Cancel' buttons, and a footer note: 'Exit the window after saving all data.'

PRO/II - Stream Data

UOM Range Help Tag Overview Status Notes

Stream: S1 Description:

To Unit P1

Stream Type

- Composition Defined
- Petroleum Assay
- Referenced to Stream
- Solids Only Stream

Flowrate and Composition...

Stream Solids Data...

Stream Polymer Data...

Thermal Condition

First Specification:

Temperature 25.00 C

Second Specification:

Pressure 202.65 kPa

Thermodynamic: Determined From Connectivity

OK Cancel

Exit the window after saving all data.

Figure 2.9 Pipe segment input menu completed

- Click “Flow rate and Composition” button and specify inlet flow rate and stream composition, then double click on the pipe icon in the PFD and specify nominal pipe diameter, pipe length, elevation change and K factor when available. For smooth pipe, the relative roughness is zero. See Figure 2.9

Pipe - Line/Fitting Data

UOM Define Range Help

Line/Fitting Diameter

Inside Diameter: mm

Nominal Pipe Size: mm Schedule:

Line Length: m

Elevation Change: m

Fitting K-Factor:

Roughness

Absolute: mm

Relative:

Divide line length into 1 segments for pressure drop calculation.

OK Cancel

Enter fitting K-factor

Figure 2.10 Process flowsheet of example 2.1

- Click run or the small arrow in the tool bar. After the run is successfully converged generate the results report. The converged process flow sheet is change to blue as shown in Figure 2.10.



Figure 2.11 Pressure drop and stream properties of example 2.1

- Select *Generate report* under Output in the Pro/II toolbar menu to display the results. The calculated total pressure drop is 52.1 kPa as shown in Figure 2.11.

CALC TOTAL PRESSURE DROP, KPA	52.05908	
CALC MAX LINE FLUID VELOCITY, M/SEC	3.99782	
MIXTURE FLOWING FLUID PROPERTIES	INLET	OUTLET
	-----	-----
TEMPERATURE, C	25.00000	25.00000
PRESSURE, KPA	202.64999	150.59091
MOLE FRACTION LIQUID	1.00000	1.00000
VELOCITY, M/SEC	3.99782	3.99782
SLIP DENSITY, KG/M3	994.93445	994.93445
SLIP LIQUID HOLDUP FRACTION, (VOL/VOL)	1.00000	1.00000
TAITEL-DUKLER-BARNEA FLOW REGIME	SINGLE PHASE	SINGLE PHASE

Aspen simulation

- ◆ Start the Aspen program, select *Aspen Plus User Interface*
- ◆ When the *Connect to Engine* window appears, use the default Server Type *Local PC*.
- ◆ Select *Pipe* under *Pressure Changes* tab from the *Equipment Model Library* and then click on the flowsheet window where you would like the piece of equipment to appear.
- ◆ To add material streams to the simulation select the material stream from the *Stream Library*. When material stream option is selected, a number of arrows will appear on each of the unit operations. Red arrows indicate a required stream and blue arrows indicate an optional stream.
- ◆ Streams can be added by clicking on the process flowsheet where one would like the stream to begin and clicking again where you would like the stream to end. In a similar fashion to the equipment, each click will add a new stream to the process flowsheet until you click on the *Select Mode Button* (the arrow at the left button corner).

FIGURE 2.12

Process flow sheet of example 2.1

- ◆ For this example, add one streams into the pipe, and one product stream leaving the pipe.
- ◆ At this point the process flowsheet should be complete and it should somewhat resemble the one shown in Figure 2.12.



Aspen Plus

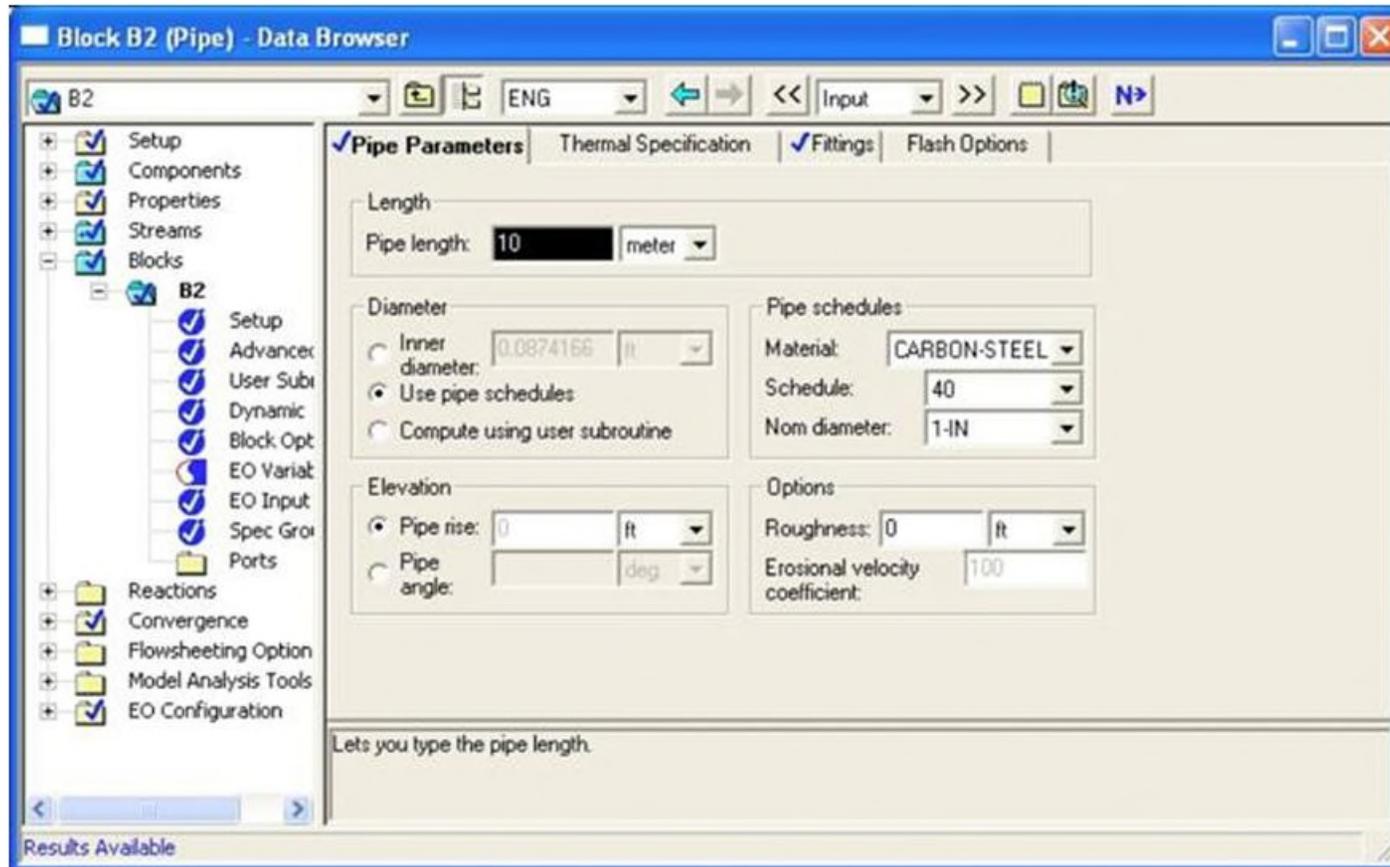
- Under the *Components tab* select *water*.
- The user input under the *Property* tab is probably the most critical input required to run a successful simulation. This key input is the *Base Method* found under the *Specifications* option. The *Base Method* is the thermodynamic basis for all of the simulation calculations. Since the fluid is water select steam fluid package.

Using N button

- Aspen has a tool in the toolbar that will automatically take the user through the required data input in a stepwise fashion.
- The button that does this is the blue *N* with the arrow next to it in the tool bar, an alternative method is to double click on the material stream and specify the feed stream conditions, and then double click on the pipe segment and specify pipe conditions as shown in Figure 2.13.

FIGURE 2.13

Pipe length and pipe schedule



Aspen Plus

- After the feed stream is specified and pipe segment is defined the Simulation Status changes to “*Required Input Complete*”.
- There are a few ways to run the simulation. The user could select either the *Next* button in the toolbar which will tell whether the required inputs are completed and ask if you would like to run the simulation. The user can also run the simulation by selecting the *Run* button in the toolbar.
- After the simulation is run and converged, the *Results Summary* Tab on the *Data Browser Window* has a blue check mark. Clicking on that tab will open up the *Run Status*. If simulation has converged it should state “*Calculations were completed normally*”.

Aspen Simulation .. continued

- Adding stream tables to the process flowsheet is a simple process, but we will first go over some options for formatting and modifying the stream tables.
- On the current screen you will see two of the options for varying the stream table: Display and Format.
- Under the Display drop down menu there are two options, all streams or streams. The streams option allows the user to choose the streams they would like presented, one by one. .

continued

- Under the Format drop down menu there are a number of types of stream tables. Each of the options presents the data in a slightly different fashion, depending on the intended application. The CHEM_E option gives the results.
- To add a stream table to process flowsheet, click on the *Stream Table* button and a stream table will be added to your process flowsheet. Process flow sheet and stream results are shown in Figure 2.14

FIGURE 2.14

Process flow sheet and stream table conditions



Example 2.1			
Stream ID		1	2
Temperature	C	25.0	25.0
Pressure	kPa	202.65	151.14
Vapor Frac		0.000	0.000
Mole Flow	kmol/sec	0.123	0.123
Mass Flow	kg/sec	2.214	2.214
Enthalpy	MMBtu/hr	-119.830	-119.831
Mass Flow	kg/sec		
WATER		2.214	2.214
Mass Frac			
WATER		1.000	1.000

Conclusion

The comparison between pressure drop values calculated by hand calculation (50.4 kPa), Hysys (51.69 kPa), PRO/II (52.1 kPa) and Aspen (51.51 kPa) reveals that there is a slight deviation between hand calculations and software simulations. The discrepancy in the hand calculation value is due to the assumption made by taking the inlet conditions in calculating Reynolds number, while the average of inlet and exit streams should be considered to have better results