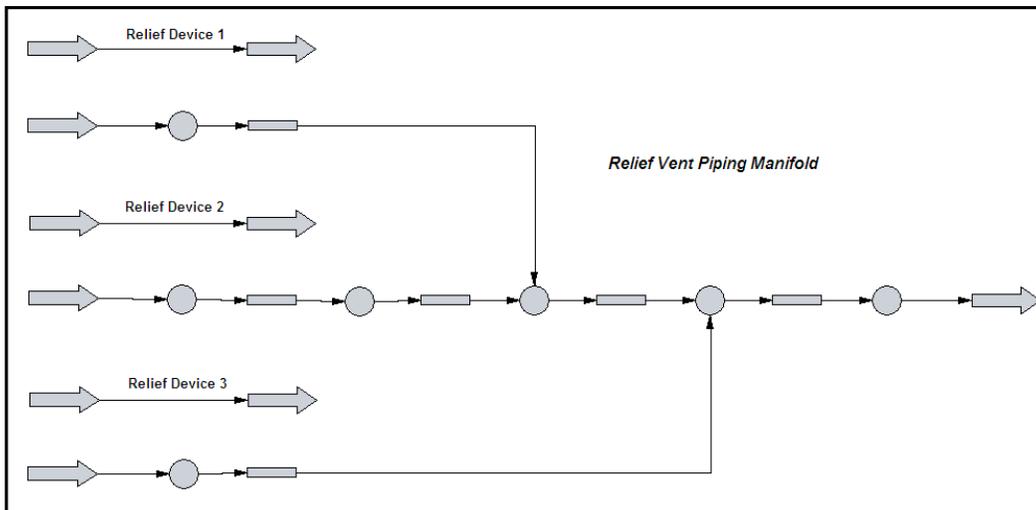


# PIPING WORKBOOK

Solving Piping  
and  
Header Networks  
Using CHEMCAD  
Version 6.2



- Piping Simulation Methods
- Sizing Piping Components
- Pump Performance Curves
- Piping Networks
- Relief Header Networks



Adapted from Chemstations CHEMCAD Piping Systems 5.3 User's Guide and Tutorial dated February 19, 2003.

Prepared by J.E.Edwards of P & I Design Ltd, Teesside, UK.

Dated 19 February 2010. MNL076 73 pages

**WORKBOOK PIPING CONTENTS**

Chapter 1	Introduction to Piping Networks	p 3
Chapter 2	Unit Operations in CHEMCAD	p 5
Chapter 3	Control Valve Sizing	p 11
Chapter 4	Simple Flow Example	p 16
Chapter 5	Branched Flow Example	p 20
Chapter 6	Relief Flow Piping Example	p 25
Chapter 7	Piping Calculation and Design Methods	p 29
	Energy Equation and Momentum	p 29
	Friction Factor	p 32
	Isothermal Flow	p 34
	Darcy Weisbach	p 34
	Hazen-Williams	p 35
	Fritzsche	p 35
	Two Phase Flow	p 36
	Baker Two Phase Flow Method	p 37
	Beggs and Brill Two Phase Flow Method	p 43
	3K-Darby Method Resistance Method	p 54
	Heat Loss in Piping Systems	p 54
	Net Positive Suction Head	p 54
	Piping Design Considerations	p 57
	Instrumentation Considerations	p 58
Chapter 8	Description of Valves and Fittings	p 59
	References	p 73

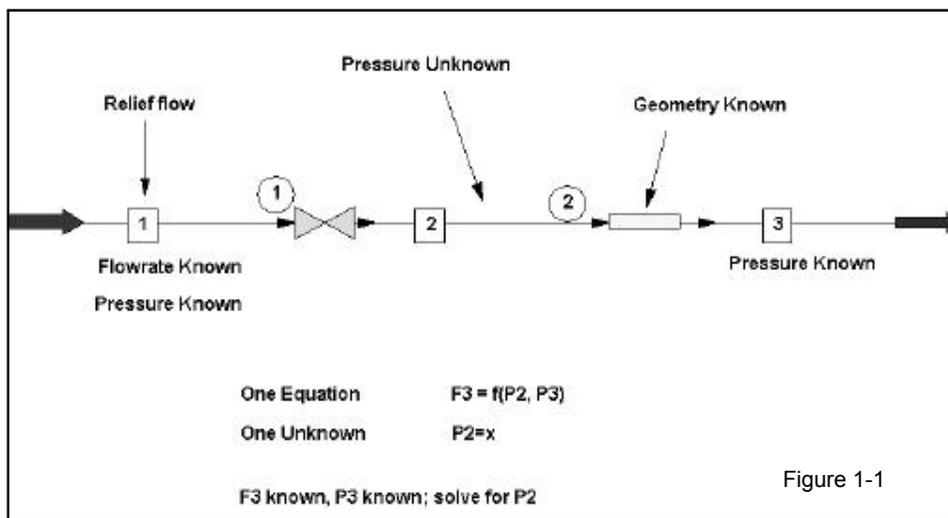
**CHAPTER 1 INTRODUCTION TO PIPING NETWORKS**

A piping network represents the flow of fluids through several items of equipment. If sufficient variables (flow rate and pressures) are specified on the network, the unknown variables may be calculated.

For fluid flow through equipment, flow rate may be calculated as a function of the inlet and outlet pressure. If the user can specify two of the three variables the third is dependent. Specification of pressure at various points on a piping network diagram allows the system to be described as a system of dependent equations.

The piping network models in CHEMCAD allow for the simultaneous solution of such a system. If sufficient constraints are specified, the model will simultaneously solve the flowsheet to converge on the unknown pressures/flow rates throughout the model.

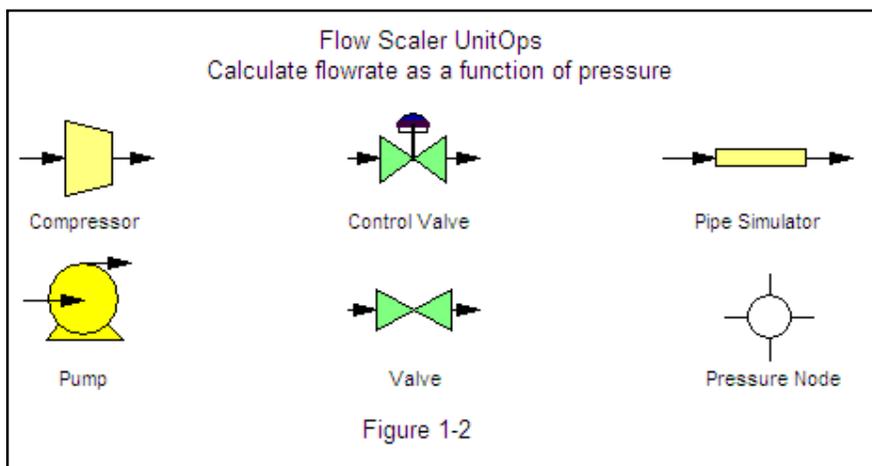
A simple relief flow system is shown in Figure 1-1.



To size the valve, the pressure out of the valve must be calculated. Known variables are geometry of the pipe, pressure out of pipe, and flow rate through pipe. A single equation can be used to solve for the pressure into the pipe as a function of the known variables.

**FLOW RATE PRESSURE DEPENDENCY FOR FLUID FLOW**

Fluid mechanics allows the calculation of fluid flow rate through a pipe or nozzle as a function of inlet and outlet pressures. The use of performance curves allows the calculation of fluid flow through a compressor or pump as a function of inlet and outlet pressures. Figure 1-2 shows UnitOps that may calculate flow rate as a function of pressures. These UnitOps are referred to as flow scaling UnitOps in this document because they scale the flow rate of the process stream.



The flow through a pipe section is determined from the general form of the Darcy Weisbach equation:

$$q = k \sqrt{\frac{\Delta P \rho d^5}{f L}}$$

The flow through a control valve is determined from an equation of the general form:

$$q = k C_v \sqrt{\frac{\Delta P}{\rho}}$$

The flow through an orifice plate is determined from an equation of the general form:

$$q = k d^2 C \sqrt{\frac{h}{\rho}}$$

$\Delta P$	pressure drop
$H$	orifice plate differential pressure
$q$	flow rate
$\rho$	fluid density
$d$	pipe inside diameter
$f$	pipe friction factor
$k$	constant
$C_v$	valve sizing coefficient
$C$	orifice discharge coefficient

## MODELING PIPE NETWORKS

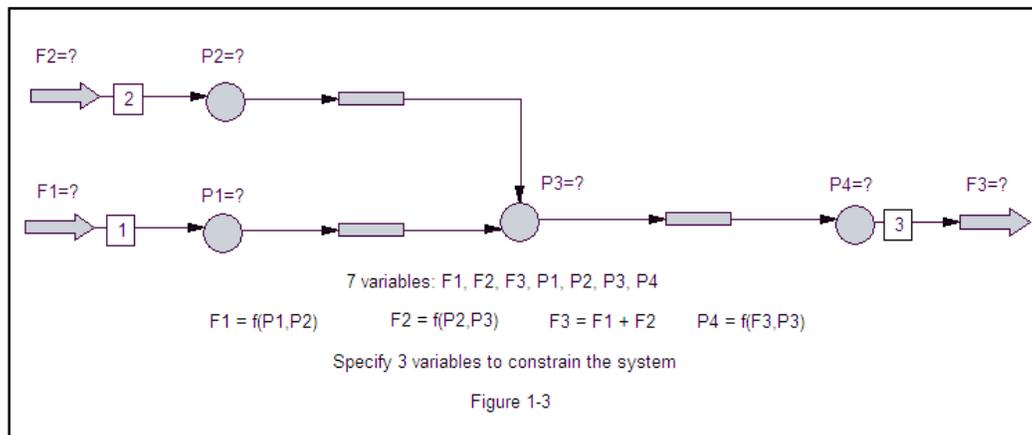
Piping network systems are used to solve for flow rates and/or pressures around a network of connected equipment. Typically the user has a flowsheet of equipment connections and various constraints (exit flow rates, pressure limitations on equipment, etc.) but does not have all the flow rate(s) and/or pressure(s) for the system.

You may solve a piping network system on a CHEMCAD flowsheet. New models in CHEMCAD allow you to specify the known variables and solve for the unknown variables on a flowsheet.

The NODE UnitOp allows you to specify the pressure on either side of a UnitOp and calculate the flow rate as a function of pressure. As an option, you may specify one pressure and the flow rate. Iterative calculations will solve for the unknown pressure based on the specified pressure and flow rate.

A series of UnitOps may be connected using several nodes. The flow rate through the chain may be specified at a single point, or calculated based on specified pressures around a UnitOp. It is not necessary to know the pressures around all UnitOps in the series.

Figure 1-3 shows a simple flare network. There are seven variables of pressure and flow rate. Three of the variables must be specified.



## CHAPTER 2

## UNIT OPERATIONS IN CHEMCAD

## PRESSURE NODES

The piping network calculations solve for pressure at nodes and then iteratively calculate the flow rates through the network as functions of pressure.

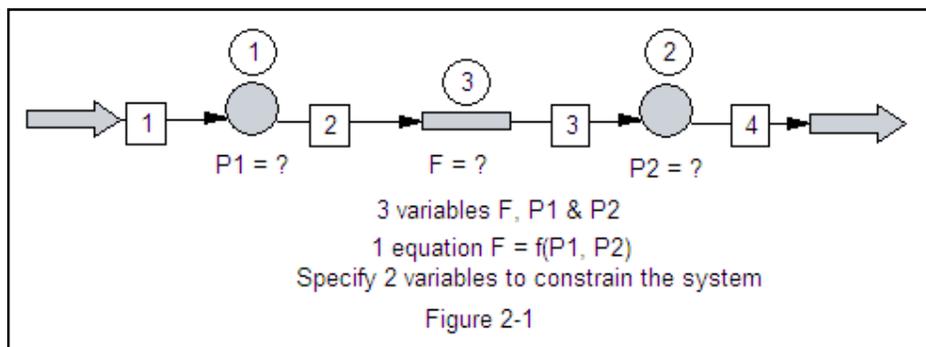
A NODE UnitOp represents a point in the piping network where a change in pressure occurs due to elevation change, flow through a pipe, or flow through equipment that changes pressure (pump, valve, etc). A CHEMCAD flowsheet for a piping network uses the pipe UnitOp for piping effects and UnitOps such as the pump, compressor, and control valve.

For design of a piping network it is necessary to determine pressure between all UnitOps that calculate pressure as a function of flow rate. The NODE UnitOp sets the pressure on one side of a UnitOp that calculates pressure as a function of flow rate.

The pressure at a node may be specified by the user or calculated by CHEMCAD. The flow rate(s) in and out of a node may be specified or calculated. The flow rates may be specified at the NODE UnitOp, or calculated as dependent on adjacent UnitOps.

The NODE UnitOp sets a fixed value on the flowsheet. For piping network calculations there are points on the flowsheet where either the pressure or flow rate is known. The NODE UnitOp allows specification of the known variable and calculation of the unknown variable.

To learn the concepts for specifying a node, look at a system of two nodes surrounding a UnitOp. This is shown in Figure 2-1.



The inlet pressure (P1), outlet pressure (P2), and flow rate (F) through the pipe are the three variables. A single equation constrains the system. Specification of any two of the variables allows CHEMCAD to solve for the third variable.

If pressure is specified at the first node and either node specifies flow rate, the pressure of the second node is variable. CHEMCAD will vary the pressure of the second node until flow rate as a function of pressure around the pipe equals the specified flow rate. The pressure may vary at either node.

The pressure of a feed or product stream of known flow rate may be adjusted by adjacent nodes. Specifying P1 as fixed pressure specifies the pressure of stream 1 as  $P=P1$ . If pressure at both nodes is specified, the flow rate through the UnitOp is a dependent variable.

The variable flow rate may be either the feed stream or product stream. In the NODE UnitOp, specify the location where flow rate is a variable. Use mode free outlet or free inlet to specify whether the inlet or outlet flow is calculated. The model will cascade this flow rate upstream and downstream of the UnitOp.

The pressure of streams attached to a NODE UnitOp will be set to the pressure of the node. The flow rates through a network will all be set to the calculated flow rate through a node. You may specify (N- 1) flow rates on a flowsheet, where N is the total of feed and product streams on the flowsheet. The calculated flow rate will be passed through nodes that use the dependent flow rate. You will receive an error message if you attempt to specify or calculate two conflicting flow rates through a system with two separate nodes.

**Flow Rate Options at Node**

The flow rate for an inlet or outlet stream may be manipulated by a node. The node acts by manipulating the flow rate of the adjacent UnitOp. The pressure settings for the nodes on either side of the adjacent UnitOp contribute to the flow rate manipulation.

**Fixed Flow Rates at Node**

Using a fixed inlet flow rate for a node specifies the flow rate through the upstream UnitOp. The pressure on one side (node) of the UnitOp must be variable. An exception is when a node is acting as a mixer or divider for N streams and the one stream is variable. In this situation the pressure can be fixed or variable for both nodes.

The Fixed outlet flow rate for a node specifies the flow rate through the downstream UnitOp. This setting is similar to fixed inlet.

The Current flow rate setting for an inlet stream is similar to fixed inlet. The current flow rate uses the flow rate currently stored for the inlet stream rather than a specified value in the node.

**Variable Flow Rates at Node**

Using free inlet for a node specifies that the feed stream flow rate is a calculated variable. The node will manipulate the upstream feed flow rate to solve the system. The free inlet specification works best on a node connected to a Feed stream but it may be placed elsewhere on the flowsheet.

If the outlet flow is specified, the free inlet specification allows the feed to be calculated to maintain mass balance. Only one free inlet specification is allowed per feed stream.

The free outlet stream for a node is similar to the free inlet setting. Using free outlet specifies that the product stream flow rate is a calculated variable. The node will manipulate the product flow rate to solve the system. The free outlet specification works best on a node connected to a product stream but it may be placed elsewhere on the flowsheet.

If the inlet flow to a system is specified, the free outlet specification allows the product to be calculated to maintain mass balance. Only one free outlet specification is allowed per product stream.

If you attempt to specify too many free outlet or free inlet streams, CHEMCAD will issue a warning message and reset the extra specifications to flow set by UnitOp.

The *flow set by UnitOp* setting indicates that the flow rate is controlled by the adjacent UnitOp. The UnitOp may be calculating flow rate as a function of pressure. The UnitOp may be using the flow rate calculated by another UnitOp.

**Mass Balance Limitations for Flow Rate Calculation**

Only one UnitOp on a branch of the network may calculate flow rate. If the nodes adjacent to a UnitOp both use *flow set by UnitOp* and *fixed pressure*, the calculated flow rate may be used as the flow rate at a *free inlet* or *free outlet* node. If the nodes adjacent to a UnitOp use *flow set by UnitOp* but do not both fix pressure, the flow rate through the UnitOp is calculated elsewhere on the flowsheet.

The behaviour of *Flow set by UnitOp* depends on the flow rate specifications of other nodes on the branch. To illustrate, we consider a system from Figure 2-1.

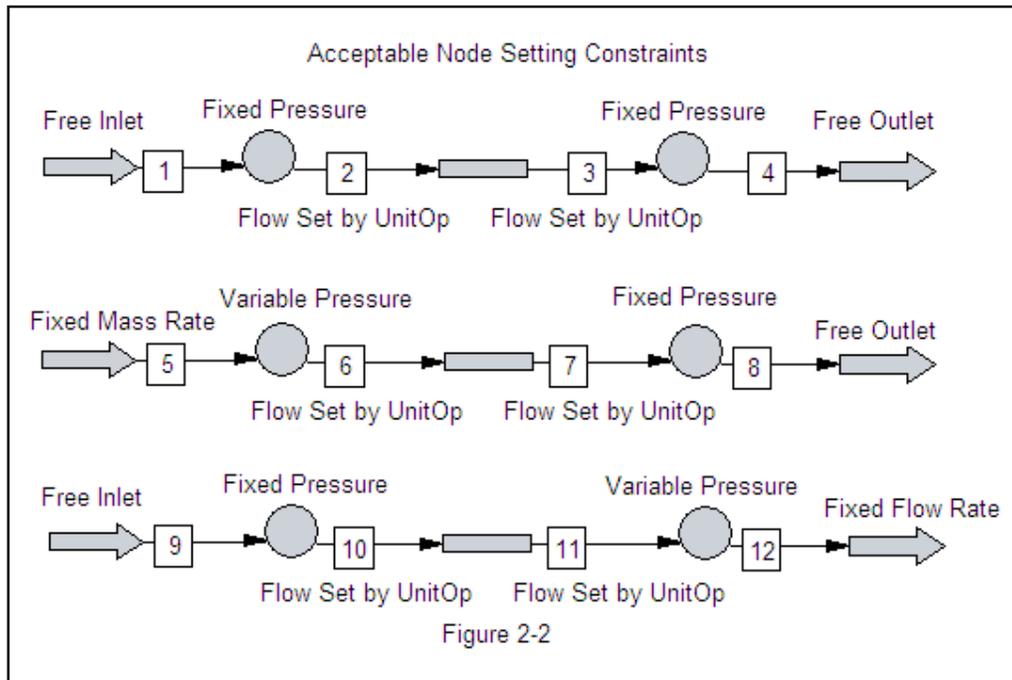
The inlet to SECOND NODE is *flow set by UnitOp*.

The node will use the flow rate from the pipe.

If the feed stream is fixed inlet, this is the flow rate for the pipe.

If the feed stream is free inlet and the product streams are fixed flow rate, the free inlet feed flow rate is set by mass balance. The free inlet is the flow through the pipe. If the feed stream is free inlet, one product stream is free outlet, and both nodes are fixed pressure, the free inlet and free outlet are set by the pipe flow rate. The pipe flow rate is set to the critical flow rate for the given pipe with the specified inlet and outlet pressures.

Figure 2-2 demonstrates various acceptable combinations of flow settings for nodes. shows the flowsheet for this job.



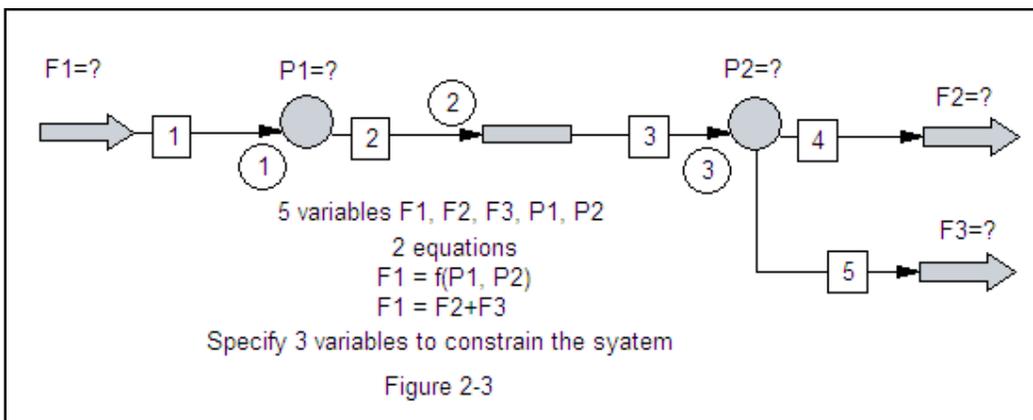
### NODE AS DIVIDER

A node may be used as a divider. Outlet streams from the node will be at the pressure of the node. Outlet streams will all have the same temperature and composition but flow rates may differ.

The flow rates may be specified as *set by pipe/valve* or fixed flow rates. Only one outlet stream flow rate may be *free outlet*.

A Node specified as a divider is shown in Figure 2-3. The second node acts as a divider (two product streams). For N inlet and outlet streams it is necessary to specify (N-1) values.

For the second node, specify the flow rate of two of the three connected streams. Allow the third stream to be *free* for mass balance requirements.



If both outlet flow rates are specified, the inlet stream must be calculated as *free inlet* at node 1 to maintain mass balance. If one outlet is calculated as *free outlet* by the node, the inlet stream may be *flow set by pipe* if both nodes are fixed pressure.

## PRESSURE NODE DIALOG SCREEN

### Mode

Select *Fixed pressure* to set the pressure at the node and allow flow rate to be variable. Select *Variable Pressure* to leave pressure variable at the node.

### Pressure at Node

Specify the pressure for *Fixed Pressure* mode. For *Variable Pressure* mode the calculated pressure is displayed. Optionally you may specify an estimate for *Variable Pressure* mode. The estimate will be replaced with calculated result.

#### Minimum Pressure

Specify upper bound for pressure at the node for *Variable Pressure* mode. Specifying minimum and maximum pressure will speed calculations.

#### Maximum Pressure

Specify a lower bound for pressure at the node for *Variable Pressure* mode. Specifying minimum and maximum pressure will speed calculations.

### Elevation

Specify the elevation at the node. The elevation will add a pressure contribution based on height. Specifications are absolute. The default elevation is zero. Positive and negative specifications are allowed.

### Flowrate options (Inlet)

#### Stream

The CHEMCAD stream number for the connected inlet stream is displayed.

#### Mode

Select the specification for the stream.

#### Fixed Mole Rate/Fixed Mass Rate/Fixed Volume Rate

The stream flow rate is a known variable. The stream flow rate is set to the specified value during calculations.

#### Flow set by Pipe/Valve/Pump

The stream flow rate is a dependent variable. Stream flow rate will be calculated by the adjacent (upstream) UnitOp to satisfy pressure requirements.

#### Free Inlet Stream

Specifies the inlet stream to the node is a dependent variable. The inlet stream flow rate will be calculated for mass balance of the node / flowsheet. Only one inlet to a node may be *Free inlet*. Specification of *Free inlet* is not allowed for a stream that is downstream of another node. Use *Free Inlet Stream* to specify a variable feed stream.

#### Use Current Stream Rate

The stream flow rate is a known variable. The stream flow rate is set to the current flow rate of the stream.

### Value

Specify a fixed flow rate for *Fixed mole rate*, *fixed mass rate*, or *fixed volume rate*. *Fixed Volume Rate* specifies the total Actual Volume Rate of the stream. Engineering Units for flow are displayed; the flowsheet units of mole rate, mass rate, and liquid flow rate are used.

### Flowrate Options (Outlet)

#### Stream

The CHEMCAD ID number for the connected outlet stream is displayed. A value of N/A indicates no stream is connected at this outlet.

#### Mode

Select the specification for the stream.

#### Fixed Mole Rate/Fixed Mass Rate/Fixed Volume Rate

The stream flow rate is a known variable. The stream flow rate is set to the specified value during calculations.

#### Flow set by Pipe/Valve /Pump

The stream flow rate is a dependent variable. Stream flow rate will be calculated by the adjacent (downstream) UnitOp to satisfy pressure requirements.

#### Free Outlet Stream

The stream flow rate from the node is a dependent variable. The Free Outlet stream flow rate will be calculated for mass balance of the node / flowsheet. Only one outlet from a node may be *Free outlet*. Specification of *Free outlet* is not allowed for a stream that is upstream of another node. Use *Free Outlet Stream* to specify a variable product stream.

### Value

Specify a fixed flow rate for *Fixed mole rate*, *fixed mass rate*, or *fixed volume rate*. *Fixed Volume Rate* specifies the total Actual Volume Rate of the stream. Engineering Units for flow are displayed; the flowsheet units of mole rate, mass rate, and liquid flow rate are used.

## PIPE SIMULATOR

### Description

The pipe simulator UnitOp in CHEMCAD is used to model pressure drop of a fluid through a pipe.

### Piping Network Modes of Pipe Simulator

Sizing Option 5 (*Given size, Pin and Pout, calc flowrate*) of the pipe UnitOp allows calculation of flow rate through the pipe as function of geometry, inlet and outlet pressure. The outlet pressure of a known pipe is a function of inlet pressure and flow rate. Any two of these three variables are independent variables.

A NODE UnitOp can use (or calculate) the flow rate from an adjacent pipe as the flow rate for a stream connected to the node. Use stream option *Flow set by pipe/valve* for the node.

The use of Sizing Option 5 for a pipe UnitOp connected to a node creates one variable on the Piping Network. The variable may be the flow rate through the pipe or the pressure at either end of the pipe.

If the node is *fixed pressure*, the pressure of the node will be used for the pressure of the attached pipe. If the nodes on either side of a pipe are *fixed pressure* with *Flow set by pipe* for the pipe UnitOp streams, the flow rate through the pipe is calculated based on the inlet and outlet pressures. If one of the nodes on either side of the pipe is *variable pressure*, the variable pressure is calculated based on the fixed pressure (from the other node) and flow rate. The flow rate may be a fixed value set by either node, or it may be specified elsewhere on the flowsheet.

## PUMP

### Description

The Pump UnitOp has a characteristic equation mode that calculates outlet pressure as a function of inlet pressure and flow rate.

### Piping Network Modes of Pump UnitOp

A NODE UnitOp can use (or calculate) the flow rate from an adjacent pump as the flow rate for a stream connected to the node. Use stream option *Flow set by pipe/valve* for the node.

The use of the characteristic equation mode specifies one unknown on the Piping Network. The unknown may be the flow rate through the pump or the pressure at either end of the pump. The node connected to the pump acts to constrain the unknown.

## VALVES

### Description

There are two UnitOps that may be used to represent valves in a Piping Network simulation. The Valve (VALV) UnitOp allows an arbitrary adiabatic pressure change of pressure between nodes.

### Piping Network Modes of Valve UnitOp

Use a valve UnitOp when a valve changes to or from a variable pressure node. The valve UnitOp is used to change the pressure of the stream to match the pressure calculated by the pressure node. The valve does not adjust flow rate unless turned off. Do not specify an outlet pressure for the valve.

## CONTROL VALVE

### Description

The PID control valve may be used in manual control model on a piping network. The valve flow coefficient (Cv) must be specified. The Control Valve (CVAL) UnitOp has three modes for manual control.

### Piping Network Modes of Control Valve

Mode *Fix valve position and adjust flow rate* is used to calculate flow rate as a function of Cv, valve position, inlet pressure, and outlet pressure. The downstream node is *fixed P* and *free inlet* stream, or *variable P* and *free outlet* stream *winlet flow set by UnitOp*.

## COMPRESSOR

### Piping Network Modes of Compressor

The Compressor UnitOp in mode 5 *Specify Performance Curves* calculates outlet pressure as a function of volumetric flow rate, efficiency, and head of gas. Volumetric flow rate and head to the compressor are functions of the mass flow rate and inlet pressure of the compressor.

### **NODE AS MIXER**

A node may be used as a mixer. The inlet streams to the node will all have the same pressure as the node. Only one inlet stream flow rate may be *free inlet*.

### **STEADY STATE UNITOPS**

Regular steady state UnitOps may be used on a piping network diagram. A constant pressure drop may be entered for a (non-scaler) steady state UnitOp. Adjacent nodes will recognize pressure drop specifications on the UnitOp.

Between two nodes there must be one flow rate scaler. A heat exchanger and a pipe can be between two nodes, as the heat exchanger does not calculate flow rate as a function of pressure. A constant pressure drop may be specified for the heat exchanger and it will affect the pressure drop between the two nodes. A heat exchanger cannot be the only UnitOp between two nodes, as the heat exchanger does not have an effect on pressure.

## CHAPTER 3

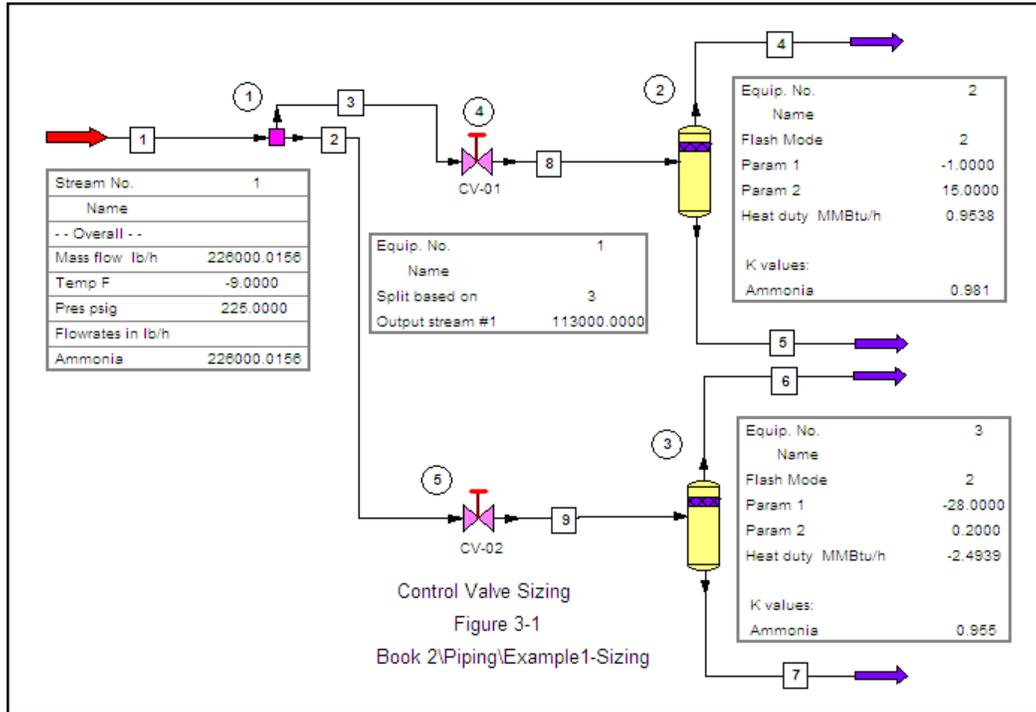
## CONTROL VALVE SIZING

Topics Covered: Control Valve Sizing / Rating, Control Valve UnitOp, Using Nodes

### Problem Statement

The example is to size control valves for handling a flow of 113,000 lb/hr of Liquid Ammonia in each line coming from Divider 1. We wish to select appropriate sized valves and then determine the percent open for each valve at the rated service. The Example is located in Book1\Piping\Example1-Sizing.

In the flowsheet Figure 3-1, inlet stream 1 is at the inlet conditions of -9 degrees F, 225 psig. The divider splits the 226,000 lb/hr flow into 2 equal flows of 113,000 lb/hr of ammonia.



### Control Valve Sizing

To do the initial sizing, run the Divider UnitOp (**Right Click>Run This UnitOp**) to calculate the flow information for streams 2 and 3. Both streams should be at -9 degrees F, 225 psig and 113,000 lb/hr of ammonia.

Next select stream 3 by left clicking on it. The stream is selected when it is shown bracketed by black squares. Go to the **Sizing** menu, and select **Control valve**. The following screen will appear: Enter 15 psig as the Downstream Pressure and press OK and the report in Excel or Word will appear.

- Control Valve Sizing -		CHEMCAD 6.2.0	
Stream ID:	3	Job name:	Example1-rating
Downstream pressure	15 psig	Date:	10/02/2010
Critical flow factor	0.98	Control Valve Sizing for Stream # 3	
Correction factor	1	Loadings and Properties	
Seat:		Vapor	Liquid
<input checked="" type="radio"/> Single-seat		Flowrate	0 lb/h
<input type="radio"/> Double-seat		Flowrate	113000.01 lb/h
Static head	ft	Flowrate	0 ft <sup>3</sup> /hr
		Density	2710.1208 ft <sup>3</sup> /hr
			41.6956 lb/ft <sup>3</sup>
		Total flow	113000.0078 lb/h
		Upstream pressure	225 psig
		Downstream pressure	15 psig
		Critical flow factor	0.98
		Corr. factor for reducers	1
		Static head	0 ft
		Seat type	Single-Seat
		Flow type	Critical flow
		Calc. coefficient Cvc	19.0972
		Capacity coefficient Cv	36
		Cvc / Cv ratio	0.5305
		Valve size	2 in

The sizing procedure is now repeated for stream 2.

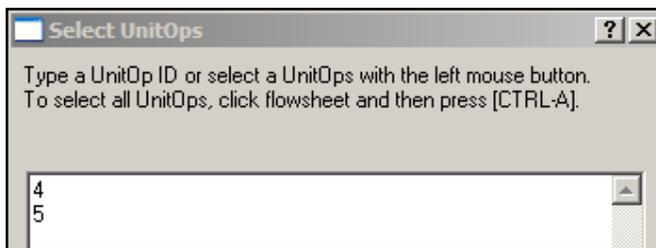
- Control Valve Sizing -		CHEMCAD	6.2.0	Page	1
Stream ID: 2		Job name:	Example1-rating	Date:	10/02/2010
Downstream pressure	0.2 psig	Control Valve Sizing for Stream # 2			
Critical flow factor	0.98	Loadings and Properties			
Correction factor	1	Vapor			
Seat:		Flowrate	0 lb/h	Liquid	113000.01 lb/h
<input checked="" type="radio"/> Single-seat		Flowrate	0 ft <sup>3</sup> /hr		2710.1208 ft <sup>3</sup> /hr
<input type="radio"/> Double-seat		Density	0 lb/ft <sup>3</sup>		41.6956 lb/ft <sup>3</sup>
Static head	ft	Total flow	113000.0078 lb/h		
		Upstream pressure	225 psig		
		Downstream pressure	0.2 psig		
		Critical flow factor	0.98		
		Corr. factor for reducers	1		
		Static head	0 ft		
		Seat type	Single-Seat		
		Flow type	Two phase flow		
		Calc. coefficient Cvc	25.7685		
		Capacity coefficient Cv	54		
		Cvc / Cv ratio	0.4772		
		Valve size	2.5 in		

Open control valve 4 by double clicking on it. The screen below will appear; note that the valve coefficient  $C_v = 36$  and the downstream pressure = 15 psig have been automatically entered from the sizing calculation. Set operating mode to Fix flow rate, adjust valve position. Repeat for valve 5.

- Control Valve -	
Valve specifications	
Controller specifications	
ID: 4	
Valve geometry	
Valve flow coefficient (Cv)	36
Rangeability	10
Critical flow factor	0.98
Valve type	
<input checked="" type="radio"/> Equal percentage valve	
<input type="radio"/> Linear valve	
<input type="radio"/> Specify valve curve	
Valve position %	72.5341
Minimum position %	
Maximum position %	
Calculated results	
Calc. flow rate	113000 lb/h
Controller output	
Steady state position	72.5341
Controller output SS	
Operating mode	
<input checked="" type="radio"/> Fix flowrate, adjust valve position	
<input type="radio"/> Fix valve position, adjust flowrate	
<input type="radio"/> Fix flow and position, calculate Pout	
Controller ID	
Static head	ft
Supply pressure	psig
Downstream pressure	15 psig
If downstream P not specified	
Destination ID	
Variable	<None>
<input type="checkbox"/> Force forward flow only	
Phase model	Selected by program

Run the simulation by going to the **Run** menu and selecting **Run All**.

To view your results, go to the **Results** menu, and select **UnitOp's**. You should see this dialog asking for what UnitOps to view:



If you don't see this, then you already have a unit selected in the flowsheet, and it is showing you a report for that unit. Close the report, deselect the UnitOps by holding down the shift key while clicking on the units, and go back to **Results, UnitOp's**. You can also deselect UnitOps by left-clicking on a blank section of the flowsheet.

Select units # 4 and 5 and press the OK button. You will see the following report:

CHEMCAD		6.2.0	
Job name:	Example1-rating	Date:	10/02/2010
<b>EQUIPMENT SUMMARIES</b>			
Control Valve			
Equip. No.		4	5
Name	CV-01	CV-02	
Valve flow coefficient		36	54
Rangeability		10	10
Critical flow factor		0.98	0.98
Downstream pressure (psig)		15	0.2
Calc. flow rate lb/h	113000.0156	113000.01	
Calculated Pout psig		15	0.2
Valve position %	72.5341	53.3338	
Valve Av	0.0625	0.0625	
Valve Bv	-0.25	-0.25	
Steady state position	72.5341	53.3338	
P crit. psig		7.3388	
DW mass rate lb/h	247.1508	238.3309	

This report shows that valve #4 is at 72.5% open, and valve #5 is at 53.3% open. By right-clicking on Control valve #5, and selecting **View stream properties**, we can see how much vaporization occurs across the valve:

CHEMCAD		6.2.0		Page
Job name:	Example1-rating	Date:		
<b>STREAM PROPERTIES</b>				
Stream No.		2	9	
Name				
-- Overall --				
Molar flow lbmol/h		6634.96	6634.96	
Mass flow lb/h	113000.0078	113000.01		
Temp F		-9	-26.367	
Pres psig		225	0.2	
Vapor mole fraction		0	0.03331	
Actual dens lb/ft3		41.6956	1.602	
Actual vol ft3/hr		2710.1208	70535.008	

This volume change is why CHEMCAD chose a larger valve for the second stream. With the vaporization occurring in the valve, a smaller 2-inch valve body would be approaching sonic velocity through the valve body.

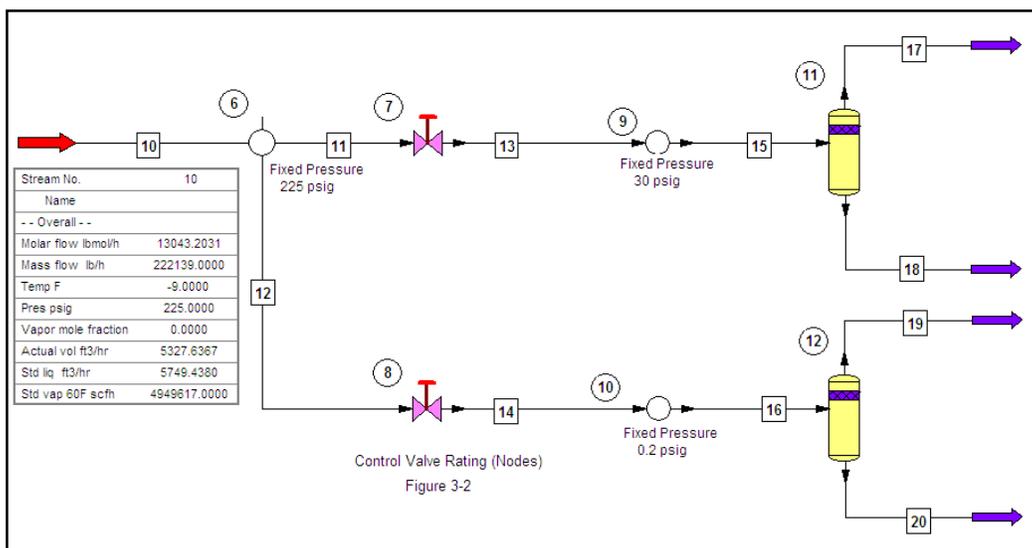
## Flow rate as a function of Pressure

In typical CHEMCAD simulations information flows in one direction: downstream. Upstream conditions determine the downstream conditions. In most simulations, you simply set the flow rates and pressures of feed streams. Pressure drops are either calculated based on flow or specified through UnitOps. The downstream pressures, flow rates, etc. are calculated when the simulation is run.

For piping simulations, flow rate and pressure are dependent on each other. The backpressure on valves, pipes, and other UnitOps affects the flow rate through the valve. Likewise, the flow rate through a valve, (or pipe, or pump) determines the downstream pressure. In flow models like the control valve sizing model, sometimes it is useful to let flow rate vary as a function of the pressure.

For example, assume a process upset caused the pressure in vessel 11 to rise from 15 psig to 30 psig. Assuming the valve positions don't change, what is the new inlet flow rate?

In order to answer this question, we need to introduce a special UnitOp called a node. A node is a point in the simulation that has a pressure, flow coming in, and flow going out. The node units create a network, solving for flow rate at each point based on the fixed pressures. Nodes are placed on the flowsheet before and after the control valves. The flowsheet is shown in Figure 3-2 below:



The function of the divider (to split the incoming flow) is now handled by NODE #6. The node will balance the flow rates such that all streams entering and exiting the node are at the same pressure. Nodes are also placed between the flash vessels and the control valves. At the nodes we can fix the pressures, and let the flow rate vary as a function of valve position and pressure difference. Open NODE 6 by double-clicking on it:

Node - ID: 6

Mode: Fixed pressure

Pressure at node: 225 psig

Elevation: [ ] ft

Flowrate Options

Inlet Streams			Outlet Streams		
Stream	Mode	Value	Stream	Mode	Value
10	Free inlet stream		11	Flow set by UnitOp	
N/A	Fixed Mole Rate		12	Flow set by UnitOp	

The pressure at this node is fixed at 225 psig. The inlet flow is set to *Free inlet stream* and the two outlet streams are set to *Flow set by UnitOp*. Flow into each control valve will be determined by the control valve Cv valve opening position, and pressure difference across the valve. The pressure in is set to 30 psig for NODE 9, 0.2 psig for NODE 10. Flow into the node is controlled by the control valve (*Flow set by UnitOp*), flow out is a *Free Outlet Stream*.

The control valves need to be changed to fix the valve position, and calculate flow rate.

We need to set the **Operating mode** for each valve to *Fix valve position, adjust flowrate* in order for the flow rate to change.

Now we can run the simulation. Go to the **Run** menu, select **Run All**. We can view the streams around NODE 6 by *right-clicking* on the node and selecting **View stream compositions** from the menu. The following report will appear:

FLOW SUMMARIES:			
Stream No.	10	11	12
Stream Name			
Temp F	-9	-9	-9
Pres psig	225	225	225
Enth MMBtu/h	-399.52	-196.29	-203.23
Vapor mole frac.	0	0	0
Total lbmol/h	13043.203	6408.2427	6634.9609
Total lb/h	222139	109138.78	113000.02
Total std L ft <sup>3</sup> /hr	5749.4379	2824.7504	2924.6877
Total std V scfh	4949617	2431791	2517825.8
Flowrates in lb/h			
Ammonia	222139	109138.78	113000.02

The flow rate to Vessel 11 dropped from 113,000 lb/hr to 109138 lb/hr. So we can see the effect of back pressure on the flow rates through the valves.

**CHAPTER 4 SIMPLE FLOW EXAMPLE**

Topics Covered: Pipe UnitOp, NPSH, Math Controllers, Pipe-Control Valve-Orifice Plate Sizing

**Problem Statement**

The piping system shown must be designed to transport 120 gpm of glacial acetic acid at 70-140F. The pressure at the inlet is known at 20 psia, the outlet must be no less than 20psia. The piping system and its individual elements must be sized for design conditions and then rated at operating conditions. Our goal is to determine the NPSH<sub>a</sub> and head requirements for future pump selection.

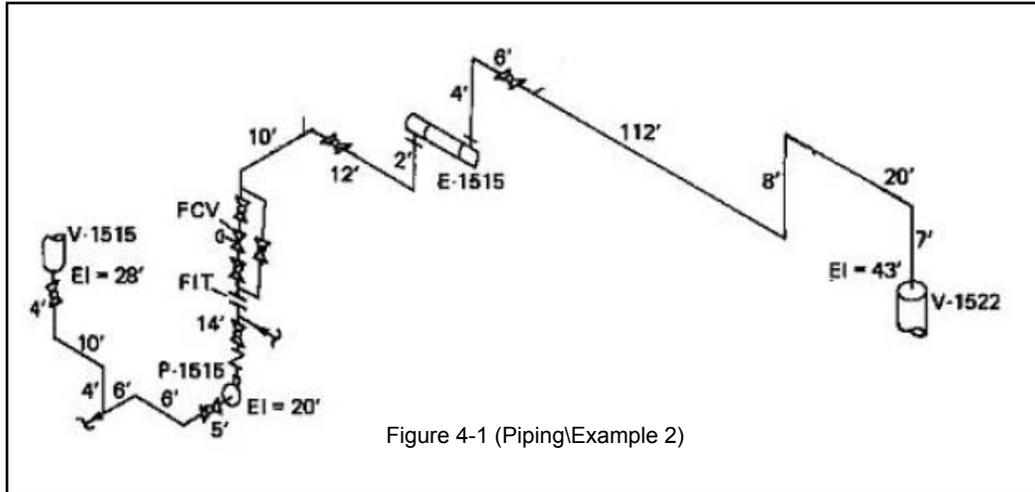
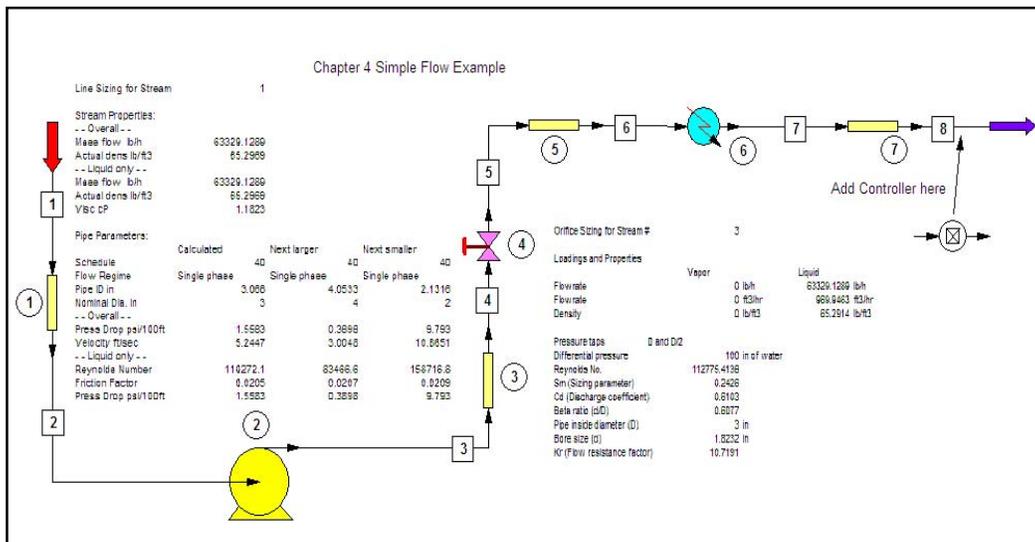


Figure 4-1 (Piping\Example 2)

**CREATING THE SIMULATION**

1) Convert the piping isometric to a CHEMCAD flowsheet. Pipe UnitOps are used to represent entire sections of piping, including fittings:



2) Using CHEMCAD's pipe sizing utility (**Sizing Menu>Pipes**) size the pipes in the network for the design flow rate of 120 gpm, at 70 °F. Use standard schedule 40 pipe. Since the fluid in this system is subcooled liquid and all flows are constant, this requires us to do only one calculation for the discharge side. As a rule of thumb, use 1 size larger pipe on the suction of the pump.

The pump discharge line size has been determined using a design velocity of 3 m/s. A discharge line size of 3 in was selected and for the suction pipe 4 in, a nominal size larger.

The Pipe UnitOps are configured identically namely Method 2 - Single Phase Flow, Sizing Option 5 – Given D, P<sub>IN</sub>, P<sub>OUT</sub>, Calculate Flow, Friction Factor - Churchill and Roughness Factor – Commercial Steel.

Stream Properties:			
-- Overall --			
Mass flow lb/h	92575.1563		
Actual dens lb/ft <sup>3</sup>	65.2869		
-- Liquid only --			
Mass flow lb/h	92575.1563		
Actual dens lb/ft <sup>3</sup>	65.2869		
Visc cP	1.1811		
Pipe Parameters:			
	Calculated	Next larger	Next smaller
Schedule	40	40	40
Flow Regime	Single phase	Single phase	Single phase
Pipe ID in	3.068	4.0533	2.1316
Nominal Dia. in	3	4	2
-- Overall --			
Press Drop psi/100ft	3.2011	0.7918	20.3878
Velocity ft/sec	7.6679	4.3931	15.8851
-- Liquid only --			
Reynolds Number	161360.8	122136.4	232249.8
Friction Factor	0.0197	0.0196	0.0203
Press Drop psi/100ft	3.2011	0.7918	20.3878

**PIPE Sizing and Rating (PIPE)**

Specifications | Properties | Calculated Results | Valves | Fittings | Heat Transfer

Method: 2 Single Phase flow ID: 4

Sizing option: 5 Given size, Pin and Pout, calc flowrate

Number of segments:

Pipe diameter: 3 in

Pipe Schedule:

Pipe Length: 14 ft

Elevation: 14 ft

Friction factor model: 0 Churchill

Enter one of the following:

Roughness factor: 0.00015 ft

Pipe Material: Commercial Steel

Buttons: Help, Cancel, OK

3) The pump discharge stainless steel orifice plate is calculated using the CHEMCAD Sizing > Orifice tool. It is specified with D and D/2 taps, a design differential of 100 in wc and the thermal expansion 9.6 E-06 in/in °F. Using the calculated bore hole, the flow resistance factor of the orifice is calculated from the following, where β= d/D, gives a value 10.719.

$$C = Cd / \sqrt{1 - \beta^4}$$

$$K_r - K = (1 - \beta^4) / (C^2 \beta^4)$$

The unrecovered pressure loss is accounted for by adding the calculated K<sub>r</sub> in the downstream Pipe UnitOp User specified window.

Orifice Sizing for Stream # 3			User specified fittings/valves				
Loadings and Properties			Type	L/D, K <sub>r</sub> , K <sub>s</sub>	K <sub>i</sub>	K <sub>d</sub>	Count
Vapor			K <sub>r</sub>	10.719			
Liquid			L/D				
Flowrate 0 lb/h			L/D				
Flowrate 63329.13 lb/h							
Flowrate 0 ft <sup>3</sup> /hr							
Flowrate 969.95 ft <sup>3</sup> /hr							
Density 0 lb/ft <sup>3</sup>							
Density 65.29 lb/ft <sup>3</sup>							
Pressure taps D and D/2							
Differential pressure 100 in of water							
Reynolds No. 112775.41							
Sm (Sizing parameter) 0.2426							
Cd (Discharge coefficient) 0.6103							
Beta ratio (d/D) 0.6077							
Pipe inside diameter (D) 3 in							
Bore size (d) 1.8232 in							
K <sub>r</sub> (Flow resistance factor) 10.7191							

4) Size the control valve (**Sizing > Control valve**) using a design pressure drop of 20 psi and a single seat valve. The **Valve position** is set to 50% open and the **Valve mode** option is set to *Fix flow, valve position, calculate Pout*.

To size the control valve from an initial build, copy the pipe network inlet stream to the control valve inlet stream using **Specifications > Copy Stream Data** and change the pressure to 45 psig. Specify valve with a 20 psi pressure drop and a correction factor of 0.95.

Loadings and Properties			
Vapor		Liquid	
Flowrate	0 lb/h	92575.1719 lb/h	
Flowrate	0 ft <sup>3</sup> /hr	176.7864 gpm	
Density	0 lb/ft <sup>3</sup>	65.2869 lb/ft <sup>3</sup>	
Total flow	92575.1719 lb/h		
Upstream pressure	39.3687 psia		
Downstream pressure	25 psia		
Critical flow factor	0.98		
Corr. factor for reducers	0.95		
Static head	0 ft		
Seat type	Single-Seat		
Flow type	Subcritical flow		
Calc. coefficient C <sub>vc</sub>	50.1999		
Capacity coefficient C <sub>v</sub>	75		
C <sub>vc</sub> / C <sub>v</sub> ratio	0.6693		
Valve size	3 in		

5) Assume a 2 psi pressure drop across the heat exchanger. Set the outlet temperature to 140 °F.

6) Tabulate the valves, fittings, pipe lengths, and elevation changes of each section of pipe. All of our pipes will use the *Single phase method*, and the **mode** should be set to *rating*. We will be using flanged fittings throughout.

Pipe #1
1 entrance, well rounded
2 ball valve
4 std elbow 90 degree
1 tee, flow through branch
35 feet piping
-8 foot elevation change

Pipe #2 (before control valve)
1 swing check valve, clearway
2 ball valves
2 tee, flow through run
1 orifice plate (as determined above)
14 feet piping
14 foot elevation change

Pipe #3 (after control valve)
2 ball valve
2 tee, flow through run
3 Std elbow 90 degree
1 exit from pipe
24 feet piping
2 foot elevation change

Pipe #4 (after E-1515)
1 Ball valve
4 std elbow 90 degree
1 tee 100% flow through run
1 well rounded entrance
1 exit from pipe
157 feet piping
5-foot net elevation change

In the example shown no allowance has been made for reducers at the pump suction and discharge which are normally required; as discussed earlier these can have a significant effect and would require additional Pipe UnitOps at the pump inlet and discharge.

7) Specify the pump outlet pressure at some arbitrary value (*45 psia*) and make a trial run. Check the calculated outlet pressure. We can iterate to find the required pump head.

### USING CONTROLLERS TO SIMPLIFY THE PROBLEM

While manually changing the pump outlet pressure will get us where we need to be, it is easier to let the program do the work. We're going to use a UnitOp called a feedback controller (**CONT**) to adjust this pressure for us. A Feedback controller in CHEMCAD has nothing to do with process control valves or PID settings. In a CHEMCAD steady-state model, when we use the term "feedback control" we are talking about a mathematical controller. It's a math tool used to adjust a variable on a flowsheet until a target value reaches our specified value.

Change the flowsheet to include a feedback controller in location shown above.

Specify the **Controller mode** as a *feedback controller*. Adjust pump outlet pressure until the pressure of stream 8 is equal to a constant target of 20 psia. When you are finished, the controller screen should be as follows:

When you run the simulation, the controller will automatically change the pump outlet pressure until the pressure leaving the last pipe unit is equal to 20. We now know the head requirements for our pump.

**Calculate *NPSHa*.** The net positive suction head is defined as pressure available at pump suction minus the fluid vapor pressure, expressed in feet of fluid. To calculate this in CHEMCAD is an easy task. Open the Pump dialog, and put a checkmark where it says Check here to Calculate *NPSHa*. Rerun the simulation, and the calculated *NPSHa* will appear. It is important to the *NPSHa* calculation that the inlet piping to the pump be correctly specified. If the piping is not correct, then the pressure at the inlet may not be correct, and the *NPSHa* may not be correct.

## CHAPTER 5

## BRANCHED FLOW EXAMPLE

**Topics: Node UnitOp, Pipe Networks, Pump Selection, Pump UnitOp, Performance Curves**

**Problem Statement**

The previous piping system has been changed; due to the branched flow to the two heat exchangers, the problem is no longer a simple one. The piping layout, valves and fittings to be used are shown in the isometric. An orifice plate and control valve is to be installed downstream of the pump to measure and control flow manually. It is required to determine the branched flow split flow and pressure drops in the pipe network

Further more the layout is to be tested to ensure an adequate Net Positive Suction Head (NPSH) is available at the pump suction. The NPSHa is defined as the total pressure available at the pump suction minus the vapour pressure of fluid at pump suction conditions. If the NPSHa is less than that required by the pump then cavitation will result.

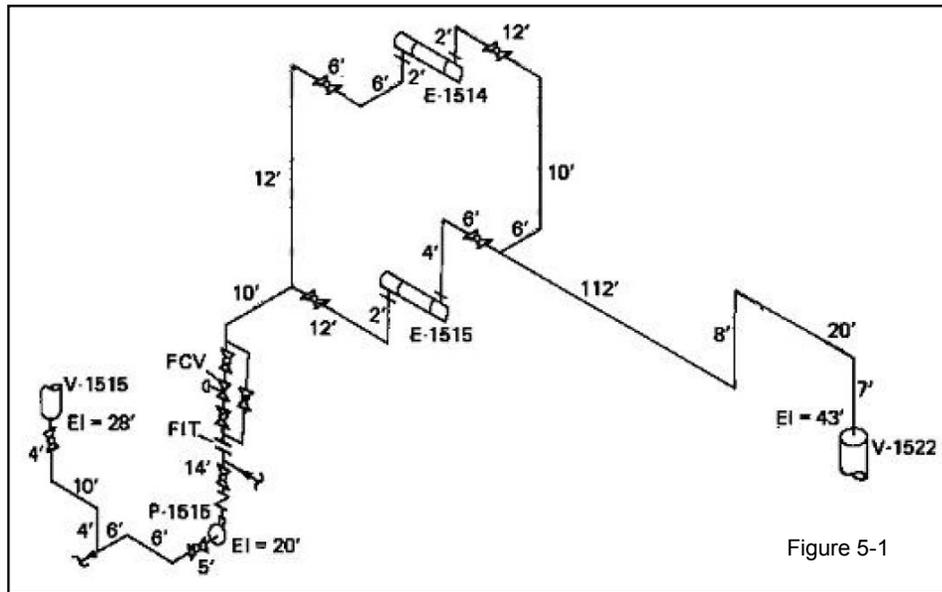


Figure 5-1

This example is located in the *PipingExample3*

The branched flow is a difficult problem to solve using our controller approach. Both exchangers have different piping and therefore different flow rates. What we need is an approach where we split and recombine flows, and have the simulation calculate the pressure and flow rates in an iterative manner. The "node" UnitOp gives us this flexibility.

**CREATING THE SIMULATION**

For practice you can build the model or use the model called "Piping Example 3 Build" in the electronic media supplied. It is strongly recommended that you work with a copy of this job. This problem is solved in CHEMCAD using the Pressure Node UnitOp.

- 1) Convert the piping isometric to a CHEMCAD flowsheet. Figure 5-2 below shows the equivalent flowsheet.

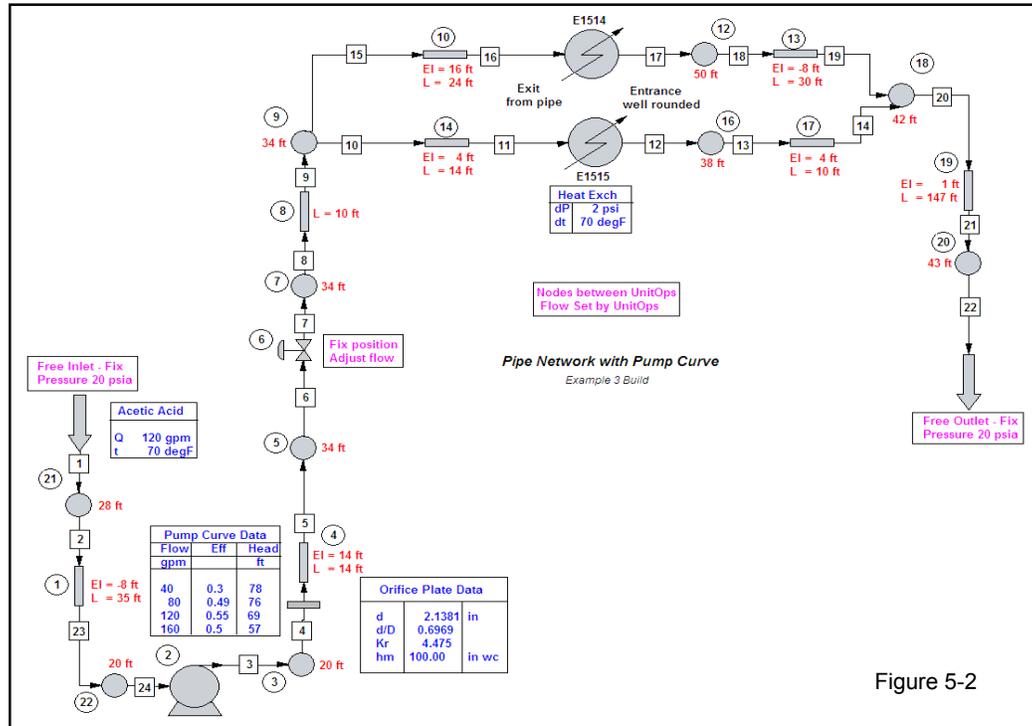


Figure 5-2

The Pressure Node UnitOp can be considered a calculator that adjusts the network pressure at the node based on the flow rate. In the network the Node sets the pressure between UnitOps that calculates flow as a function of pressure.

Pipe UnitOps calculate flows based on the  $P_{in}$  and  $P_{out}$ , the Pump and Control Valve UnitOps calculate flows based on the downstream pressure in the Node; it follows that Node UnitOps located between UnitOps that calculate the flow are set in the "Flow Set by Upstream and Downstream UnitOps".

The pressure at the Inlet and Outlet Nodes of this network are fixed at 20 psi and the stream is defined as Free ie not effected by a UnitOp. The inlet flow could also have been fixed by the Inlet Node.

At a UnitOp there are three variables- $P_{IN}$ ,  $P_{OUT}$  and  $F$ ; a single equation constrains the system so specification of any two variables sets the remaining variable.

A node is a point where pressure is uniform. There may be multiple inlets and outlets. The flow rates for each stream will be balanced by CHEMCAD to reach a single pressure. Pressure may be specified or allowed to vary.

The Inlet and Outlet Nodes configuration is Fixed Pressure and all other Nodes are Variable Pressure as shown below:

<b>Mode</b> Fixed pressure				<b>Mode</b> Variable pressure			
Pressure at node: 20 psia		Minimum pressure: 10 psia		Pressure at node: 33.3687 psia		Minimum pressure: 30 psia	
Elevation: 28 ft		Maximum pressure: 30 psia		Elevation: 34 ft		Maximum pressure: 80 psia	
<b>Flowrate Options</b> Inlet Streams: Stream 1, Mode: Free inlet stream, Value:				Inlet Streams: Stream 5, Mode: Flow set by UnitOp, Value:			
Outlet Streams: Stream 2, Mode: Flow set by UnitOp, Value:				Outlet Streams: Stream 6, Mode: Flow set by UnitOp, Value:			

- 2) Pipe UnitOps are used to represent entire sections of piping, including fittings. NODE UnitOps are placed where pressure or flow rate are unknown.
- 3) Assume a 2 psi pressure drop across each heat exchanger.
- 4) Tabulate the valves, fittings, pipe lengths, and elevation changes of each section of pipe. We will be using flanged fittings throughout.

Pipe #1
1 entrance, well rounded
2 ball valve
4 std elbow 90 degree
1 tee, flow through branch
35 feet piping
-8 foot elevation change

Pipe #2 (before control valve)
1 swing check valve, clearway
2 ball valves
2 tee, flow through run
1 orifice plate (as determined above)
14 feet piping
14 foot elevation change

Pipe #3 (after control valve)
1 ball valve
1 tee, flow through run
2 Std elbow 90 degree
10 feet piping
No elevation change

Pipe #4 (to E-1514)
1 Ball valve
3 std elbow 90 degree
1 tee flow through branch
1 exit from pipe
24 feet piping
16 foot net elevation change

Pipe #5 (from E-1514)
1 Ball valve
3 std elbow 90 degree
1 tee flow through branch
1 well rounded entrance
30 feet piping
-8 foot net elevation change

Pipe #6 (to E-1515)
1 Ball valve
1 std elbow 90 degree
1 tee flow through branch
1 exit from pipe
14 feet piping
4 foot net elevation change

Pipe #7 (from E-1515)
1 Ball valve
1 std elbow 90 degree
1 tee flow through run
1 well rounded entrance
10 feet piping
4 foot net elevation change

Pipe #8 (to V-1522)
3 std elbow 90 degree
1 exit from pipe
147 feet piping
1 foot net elevation change

- 5) Pump- At this time we don't know the pump specifications, so we will set the pump to "Specify Outlet Pressure" and leave the pressure specification blank. The NODE UnitOps will solve for the pressure increase, and set the pump outlet pressure accordingly.

- 6) Setup nodes with appropriate information:

Node 3	Variable pressure, use current stream rate for inlet, flow set by UnitOp for outlet. Elevation = 20 feet
Node 5	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 34 feet
Node 7	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 34 feet
Node 9	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 34 feet
Node 12	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 50 feet.
Node 16	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 38 feet.
Node 18	Variable pressure, flow set by UnitOp for both inlet and outlet. Elevation = 42 feet.
Node 20 (last node)	Fixed pressure, 20 psia, flow set by UnitOp for inlet, free outlet stream for outlet. Elevation = 43 feet.

### RUNNING THE SIMULATION

To run the simulation, click the **Run All** button, or select **Run > Run > Run All**.

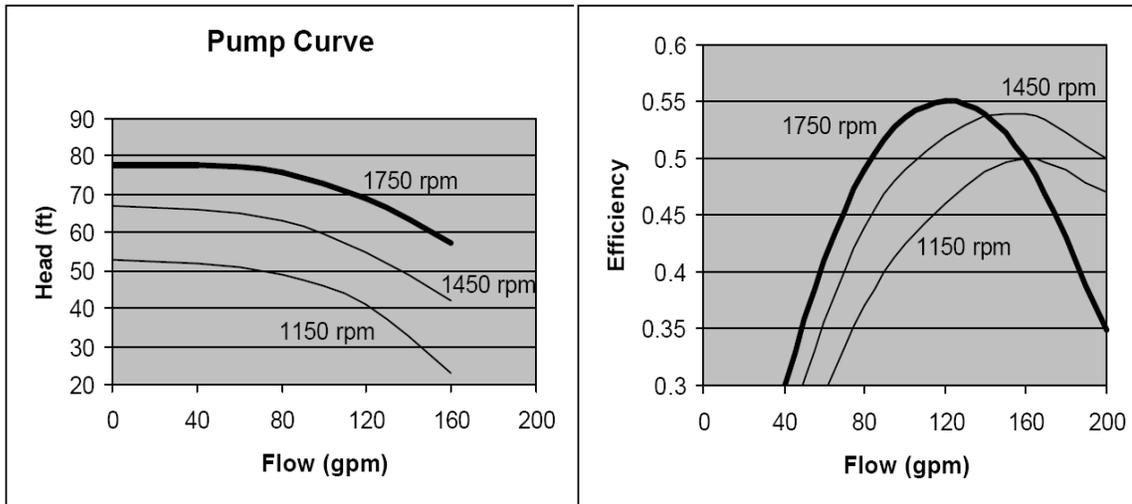
If the simulation doesn't converge, check all the input settings and run it again. Simulations can be sensitive to initial estimates and min/max settings in nodes. Convergence is also an iterative process, so you might need to increase the maximum number of iterations for a given flowsheet.

Once the simulation has run, check the pump UnitOp's pressure increase. This tells us the head requirements of our pump. Using this information and our flow rate (120 gpm) we can consult a pump handbook to determine the correct pump size.

### SELECTING A PUMP

Below is an appropriate pump for this application.

The centrifugal pump to be used has the following performance characteristics:



To enter this curve into our pump, select *Specify Performance Curve* for the pump **Mode**. Once you do you have the opportunity to enter multiple speed lines and an operating speed. For our purposes, we'll assume we want the pump to be at 1750 RPM, so we enter a single speed line. Press the **OK** button, and you will see the following entry screen:

Performance Curve			
	Flow (gpm)	Efficiency	Head (ft)
1	39.99542	0.3	78
2	79.99084	0.49	76
3	119.9863	0.55	69
4	159.9816	0.5	57
5	0	0	0

In this screen we can enter as many points as we wish to define our curve. CHEMCAD will fit a quadratic equation to the points, creating a smooth curve fit to our data.

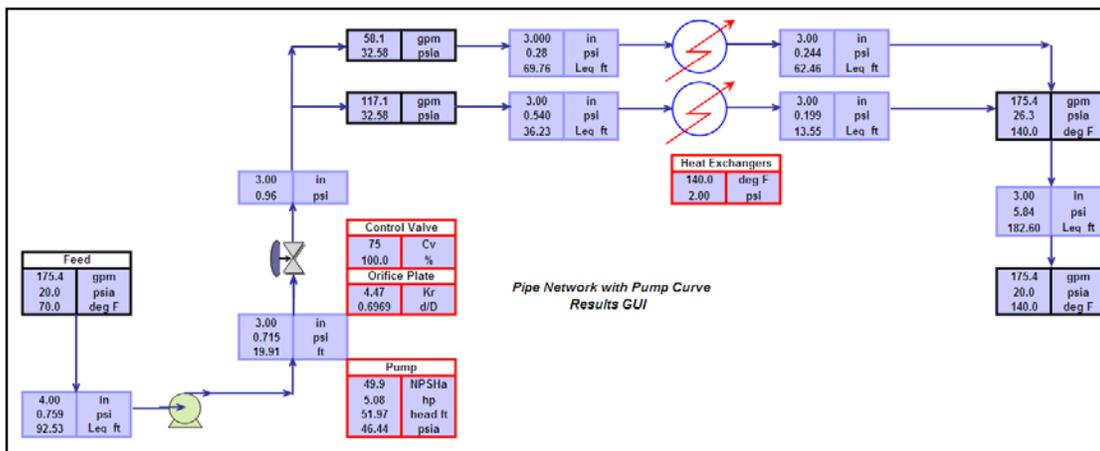
Now that we have entered the pump performance curve we must add node UnitOps to the suction side of the pump to allow the flow rate to vary, as shown on flowsheet above. In this way we can calculate the max flow rate of our system.

*Note: when computing the max flow for the system, be sure to open the control valve to 100%.*

## RESULTS

The results have been presented in a Graphical User Interface (GUI) format to give a clearer representation. It has been generated using the CHEMCAD DataMap facility. The graphics and reporting have been done using Excel.

The results are shown for the control valve fully open. It can be seen there is adequate NPSH and the discharge pressure criteria have been met. The flow split through the heat exchangers, as a result of the piping layout and resistances, is predicted to be 56.1 gpm and 117.1 gpm.



Alternatively a report can be generated using the standard CHEMCAD reporting facilities.

## CHAPTER 6

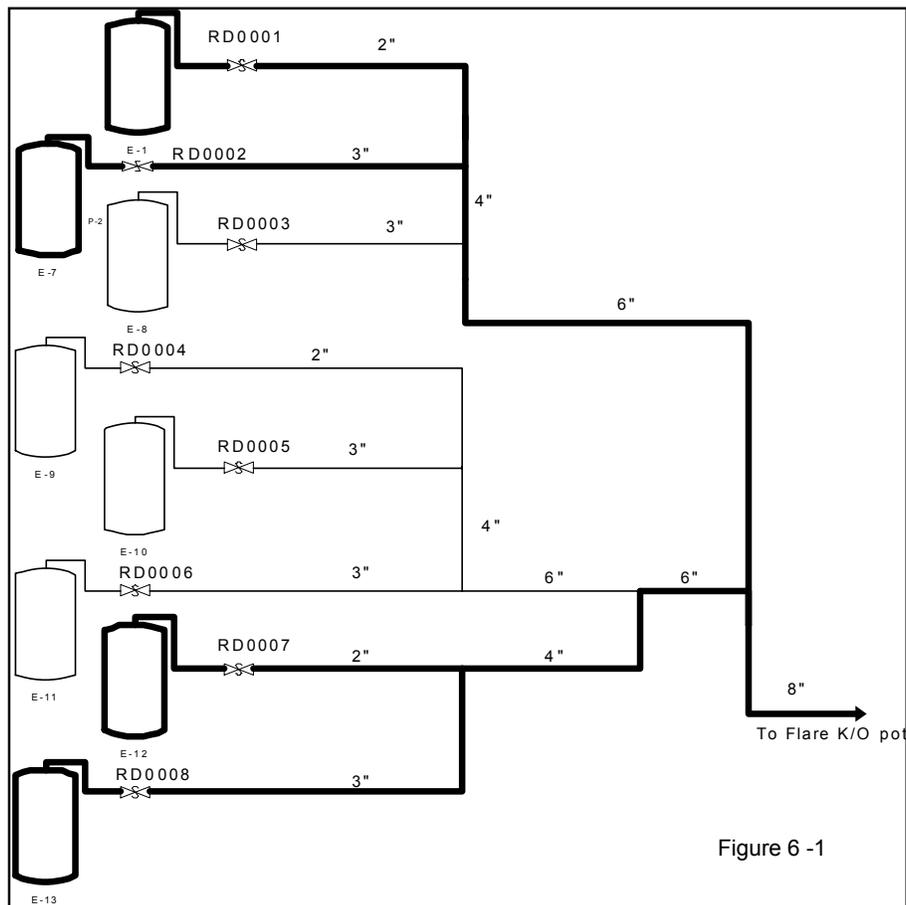
## RELIEF FLOW PIPING EXAMPLE

**Topics: Branched Network, Compressible flow, Degrees of Freedom, Flare header systems, Node UnitOp, Pipe UnitOp, Valve UnitOp**

### FLARE HEADER DESIGN

Flare headers are specialized piping networks designed to convey relief device flows to a flare where products are consumed before release to the atmosphere. Using CHEMCAD, we can design and evaluate flare header networks for many relieving scenarios.

Figure 6-1 shows a simplified flowsheet for a relief header network. The flows and pressures from relief devices RD0001-RD0008 are previously calculated. The piping network has been designed but not certified or built. The pipes on this flow diagram represent continuous sections of pipe with various fittings (elbows, etc).



### Problem Statement

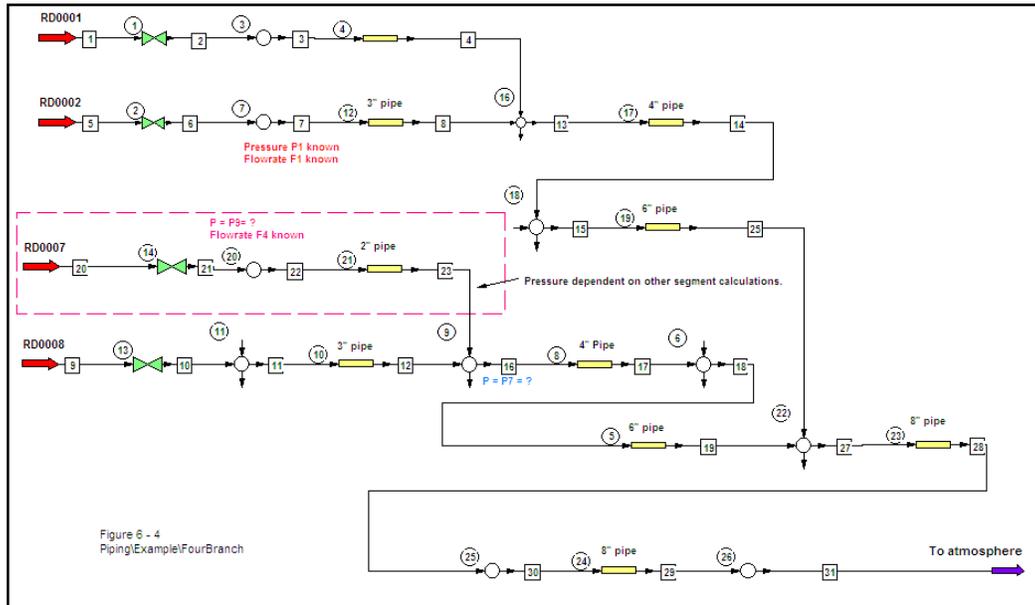
- Calculate pressures throughout the network when RD0001, RD0002, RD0007, and RD0008 are in a relief scenario
- Determine if an additional section of 8 inch pipe can be added to the discharge
- Determine effects of a two phase flow scenario through RD0007
- Calculate pressures throughout the network if only three segments of the network have a relief event

These tasks may be solved using the NODE UnitOp and Piping Network. The problem will be solved one section of piping at a time. It is easier to tackle the problem if we divide it into smaller problems.



## FOUR BRANCH

A fourth branch is added to the network, shown in Figure 6-4 below. As our flowsheet becomes more complex, the upper/lower bounds for pressure in the nodes become more important. A more complex network may require more careful settings of upper and lower bounds.



The flowsheet is run and converged after adding the third and fourth sections. All four branches are on the network. The calculated discharge pressure is above atmospheric; the network is adequate for this relieving scenario.

## SPECIFYING THE OUTLET PRESSURE

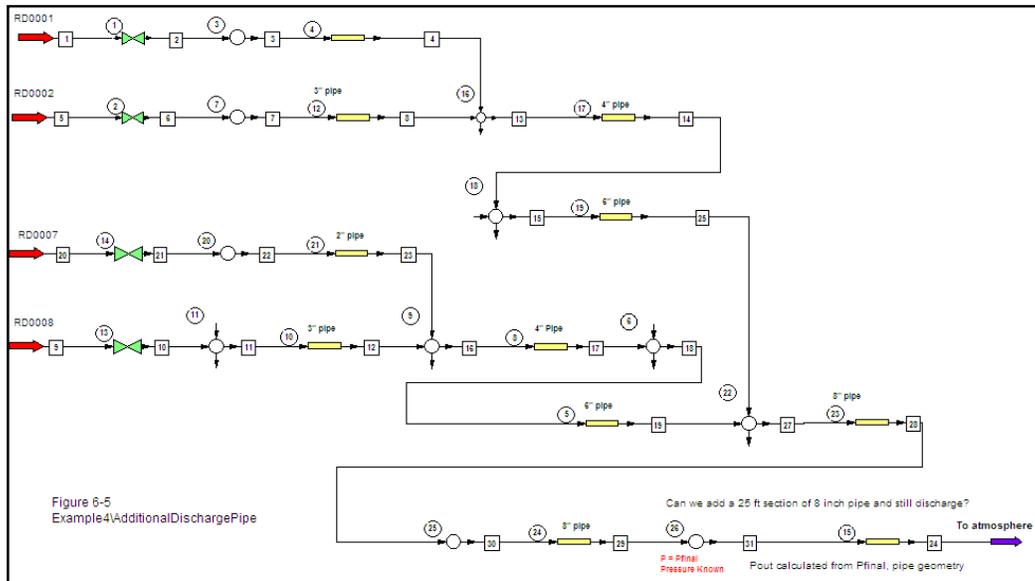
We can now perform a rating case on this network, to determine the minimum pressure at RD0002 for relief under this scenario.

The pressure on the first section node is changed to variable pressure. The outlet pressure for the network is changed to fixed pressure. The boundaries on variable pressures are tightened and the flowsheet is converged to the same results, but with the fixed pressure node changed from inlet to outlet.

To drop the pressure to 2 psig, one should relax the lower pressure boundaries on variable nodes. By making several minor adjustments from the current pressure to 2 psig, fewer calculations will be required.

## ADDITION OF PIPE TO DISCHARGE

A pipe is added to the discharge of the network. The pipe is set to rating mode in CHEMCAD; there is no need to use nodes around this final pipe. The flowsheet is run; the final pipe calculates outlet pressure. The design modes of the pipe can be used to determine a required diameter for the pipe if the required length of pipe is known.



## EFFECT OF CLOSED VALVE

The simple valve (VALV) UnitOp in CHEMCAD has a closed setting. This can be used to shutoff the flow to a section of pipe. In this example we set the valve on RD001 to closed and run the process.

## DIFFERENT APPROACHES TO RELIEF PROBLEMS

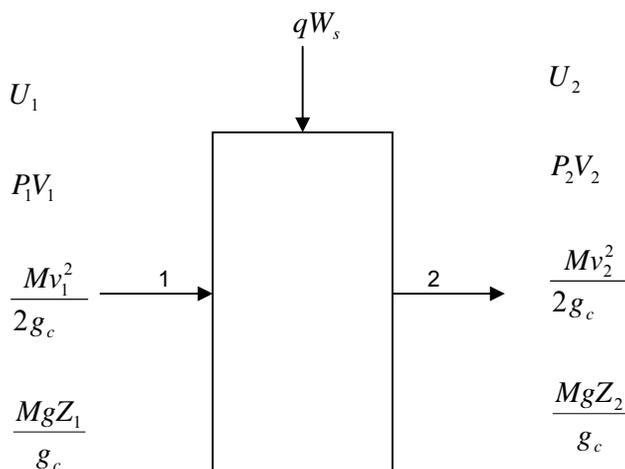
This has not been an exhaustive study of the use of the piping network calculations in CHEMCAD. This example process does not represent the only way to study a relief header system; the approach used was selected for instructional purposes. CHEMCAD's methods are flexible; you can use a different approach (fixed pressure inlets, fixed flow on one section, etc) to suit your purposes.

## CHAPTER 7

## PIPING CALCULATION AND DESIGN METHODS

## THE GENERAL ENERGY EQUATION

An expression for the balance or conservation of energy between two points in a system can be derived as follows:



An energy balance results in:

$$U_1 + P_1V_1 + \frac{Mv_1^2}{2g_c} + \frac{MgZ_1}{g_c} + q + W_s = U_2 + P_2V_2 + \frac{Mv_2^2}{2g_c} + \frac{MgZ_2}{g_c} \quad \dots(1)$$

Divide by M and writing the equation in the differential form, on a unit mass basis:

$$dU + d\left(\frac{P}{\rho}\right) + \frac{v dv}{g_c} + \frac{g}{g_c} dZ + dq + dW_s = 0 \quad \dots(2)$$

Equation 2 is difficult to apply because U is hard to evaluate. Therefore, equation 2 is converted into the mechanical energy form.

Since:

$$\begin{aligned} dU &= dh - d\left(\frac{P}{\rho}\right) \\ dh &= TdS + \frac{dP}{\rho} \\ dU &= TdS + \frac{dP}{\rho} - d\left(\frac{P}{\rho}\right) \end{aligned} \quad \dots(3)$$

Replacing equation 3 into equation 2 and rearranging the equation, we obtain:

$$TdS + \frac{dP}{\rho} + \frac{v dv}{g_c} + \frac{gdZ}{g_c} + dq + dW_s = 0 \quad \dots(4)$$

For an irreversible process,

$$dS \geq \frac{-dq}{T} \quad \text{or} \quad TdS = -dq + P_f$$

and assuming no work is done on the system, equation 4 becomes:

$$\frac{dP}{\rho} + \frac{v dv}{g_c} + \frac{gdZ}{g_c} + dP_f = 0 \quad \dots(5)$$

if  $Z = dL \sin \phi$  and multiplying by  $\left(\frac{\rho}{dL}\right)$

Equation 5 becomes:

$$\frac{dP}{dL} + \frac{v\rho}{g_c} \frac{dv}{dL} + \frac{g\rho}{g_c} \sin \phi + \frac{\rho dP_f}{dL} = 0 \quad \dots(6)$$

ENERGY EQUATION NOMENCLATURE	
$U_i$	Internal energy
$P_i V_i$	Energy of expansion or contraction
$P$	Pressure
$V$	Volume
$\frac{Mv_i^2}{2gc}$	Kinetic energy
$M$	Mass
$v$	Velocity
$g_c$	Gravity constant
$\frac{MgZ_i}{gc}$	Potential energy
$Z$	Elevation above datum
$q$	Heat additions (losses)
$W_s$	Work done on the fluid
$h$	Enthalpy
$\rho$	Density
$T$	Temperature
$S$	Entropy
$P_f$	Losses due to irreversible processes like friction.
$L$	Length
$\sin \phi$	Some angle of elevation

**CONSERVATION OF MOMENTUM**

Conservation of Linear Momentum Equation:

$$\frac{d}{dL}(\rho v) + \frac{d}{dL}(\rho v^2) = -\frac{dP}{dL} - \tau \frac{\pi D}{A} - \rho g \sin \theta \quad \dots(7)$$

At steady state:

$$\frac{d(\rho v^2)}{dL} = \frac{dP}{dL} - \tau \frac{\pi D}{A} - \rho g \sin \theta \quad \text{and} \quad \frac{d(\rho v^2)}{dL} = v \frac{d(\rho v)}{dL} + \rho v \frac{dv}{dL}$$

then,

$$v \frac{d(\rho v)}{dL} + \rho v \frac{dv}{dL} = \frac{dP}{dL} - \tau \frac{\pi D}{A} - \rho g \sin \theta$$

At steady state  $\frac{d(\rho v)}{dL} = 0$ , replacing this value and rearranging the above equation, we obtain:

$$\frac{dP}{dL} = \rho v \frac{dv}{dL} + \tau \frac{\pi D}{A} + \rho g \sin \theta \quad \dots(8)$$

CONSERVATION OF MOMENTUM NOMENCLATURE			
L	Length	$\tau$	Shear stress coefficient
$\rho$	Density	$\pi$	Pi
v	velocity	D	Diameter
$\frac{dP}{dL}$	Pressure loss	A	Cross sectional area
P	Pressure	$\rho g \sin \theta$	Gravitational loss
$\tau \frac{\pi D}{A}$	Shear stress loss (friction)	$\rho v \frac{dv}{dL}$	Acceleration losses. Normally negligible and significant only if compressible flow exits at low pressure.

An evaluation of wall shear stress (or friction losses) can be accomplished by defining a dimensionless friction factor f, the Darcy-Weisbach(or Darcy or Moody Or Blasius) friction factor:

$$f = \frac{2\tau_w}{\rho v^2} \quad \text{manipulation produces the Darcy-Weisbach equation:}$$

$$\left(\frac{dP}{dL}\right)_f = \left(\frac{f\rho v^2}{8}\right) \left(\frac{\pi D}{\pi D^2/4}\right) = \frac{f\rho v^2}{2D} \quad \dots(9)$$

WALL SHEAR STRESS NOMENCLATURE			
$\tau_w$	Wall shear stress	$\left(\frac{dP}{dL}\right)_f$	Pressure loss due to friction
$\frac{\rho v^2}{2}$	Fluid kinetic energy	$\rho$	Density
P	Pressure	v	velocity
L	Length	D	Diameter

**FRICTION FACTOR DETERMINATION**

For Laminar Flow an analytical expression can be derived for the Darcy friction factor:

$$f = \frac{64}{Re} \quad \text{or} \quad f_F = \frac{16}{Re}$$

For Turbulent flow, the friction factor is developed from Colebrook:

$$f = \left\{ 1.74 - 2 \log \left( \frac{2.27}{D} + \frac{18.7}{Re \sqrt{f}} \right) \right\}^{-2}$$

In terms of a Darcy-Weisbach or Moody friction factor:

$$f = 4f_F \quad \text{and} \quad \left( \frac{dP}{dL} \right)_f = \frac{f \rho v^2}{2g_c D}$$

**Acceleration Component**

- 7) The acceleration component will be equal to zero for constant area, incompressible flow.
- 8) For any flow condition in which a velocity change occurs, such as compressible flow, a pressure drop will occur in the direction of velocity increase.

The Jain equation is used to solve directly for the Darcy Weisbach friction factor  $f$  for a full-flowing circular pipe. It is an approximation of the implicit Colebrook-White equation.

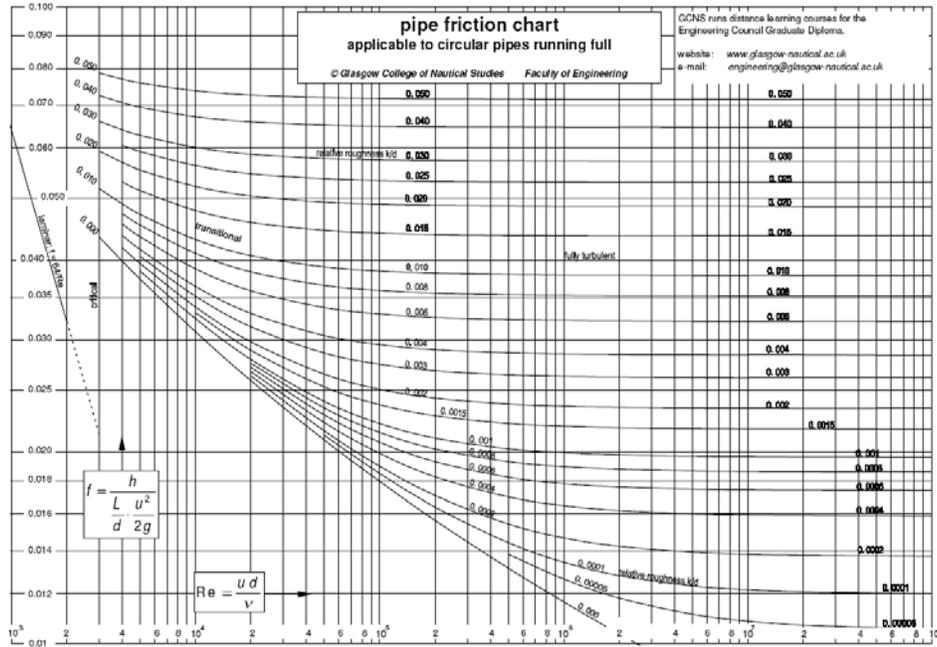
$$f = \frac{0.25}{\left[ \log_{10} \left( \frac{\epsilon}{3.7D} + \frac{5.74}{Re^{0.9}} \right) \right]^2}$$

The equation was found to match the Colebrook-White equation within 1.0% for  $10^{-6} < \epsilon/D < 10^{-2}$  and  $5000 < Re < 10^8$ . However the Churchill method is applicable for all values of  $\epsilon/D$  and  $Re$ .

FRICTION FACTOR NOMENCLATURE	
$f$	Darcy Friction factor (CHEMCAD uses this form throughout)
$Re$	Reynolds number
$\rho$	Density
$D$	Diameter
$v$	velocity
$f_F$	Fanning friction factor
$\left( \frac{dP}{dL} \right)_f$	Pressure loss due to friction
$g_c$	Gravitational constant
$\epsilon$	Pipe roughness

For an independent check of the friction factor the Moody diagram is used. Knowing the pipe flow  $Re$  and the pipe roughness coefficient  $\epsilon$  (units of m or ft), giving the relative roughness  $\epsilon/D$  (consistent units), the friction factor can be determined. The laminar flow line formula will allow verification of the diagram friction factor being used. Check friction factor at  $Re=1000$ ; if Darcy  $f=0.064$  and if Fanning  $f_F=0.016$ .

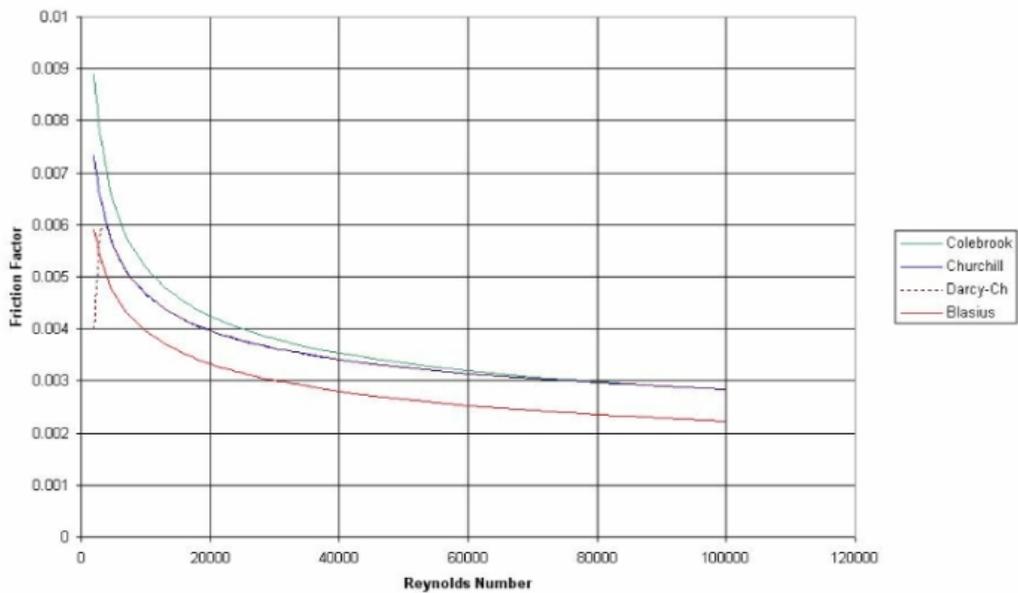
**Moody Diagram – Darcy Friction Factor**



Example Friction factor for cast iron pipe D = 500mm, ε = 0.5 mm (ε/D = 0.001) with Re of 300000 is 0.026

The diagram below shows Colebrook, Churchill, Darcy-Churchill and Blasius friction factors for smooth pipes. The Blasius Equation being the most accurate for smooth pipes. estimating turbulent pressure drops. Smooth pipe conditions are very well defined.

**Comparison of Smooth-pipe Friction Factor Correlations.**



**ISOTHERMAL FLOW EQUATION**

The isothermal flow equation is solved as follows:

$$w = 1337.024 \left\{ \frac{\rho D^4}{f \left( \frac{12L}{D} \right) + 2 \ln \left( \frac{P_1}{P_2} \right)} \left[ \frac{P_1^2 - P_2^2}{P_1} \right] \right\}^{\frac{1}{2}}$$

$$P_2 = \left\{ P_1^2 - \left[ f \left( \frac{12L}{D} \right) + 2 \ln \left( \frac{P_1}{P_2} \right) \right] \left[ \frac{P_1^2 - P_2^2}{P_1} \right] \right\}^{\frac{1}{2}}$$

$$\rho = \frac{(MW)P}{RT}$$

ISOTHERMAL EQUATION NOMENCLATURE	
w	Gas flow, lb/h
$\rho$	Gas density, lb/ft <sup>3</sup>
D	Terminal gas pressure, psia
L	Pipe length, ft.
P <sub>1</sub>	Initial gas pressure, psia
P <sub>2</sub>	Terminal gas pressure, psia
f	Friction factor
MW	Molecular weight of gas
P	Gas pressure, psia
R	Gas constant, 10.73
T	Absolute, temperature °R

**THE DARCY-WEISBACH EQUATION**

$$\Delta P = \frac{\rho f L v^2}{144 D (2g_c)}$$

DARCY-WEISBACH NOMENCLATURE	
$\Delta P$	Pressure drop
$\rho$	Density
f	Darcy Friction factor
L	Pipe length
v	Velocity
D	Diameter
g <sub>c</sub>	Gravitational constant

**HAZEN-WILLIAMS EQUATION**

The Hazen-Williams equation is used for the design of water sprinkler systems for fire protection. The National Fire Protection Association mandates the design of such systems.

The Hazen and Williams formula for flow of water is:

$$Q = 0.442D^{2.63}C\left(\frac{P_1 - P_2}{L}\right)^{0.54}$$

HAZEN-WILLIAMS EQUATION NOMENCLATURE	
Q	Volumetric flow of water, gal/min
D	Inside diameter, ft
C	140 for new steel pipe 130 for new cast iron pipe 100 is often used for design purposes to account for pipe fouling
P	Pressure, psia
L	Length of pipe segment, ft

**FRITZSCHE EQUATION**

The Fritzsche equation is used to calculate the pressure drops in steam systems:

$$\Delta P_{100} = \frac{210.82w^{1.85}}{\rho(10)^7 D^{4.97}}$$

FRITZSCHE EQUATION NOMENCLATURE	
$\Delta P_{100}$	Pressure drop per 100 feet, psi
w	Mass flow rate, lb/hr.
$\rho$	Density, lb/ft <sup>3</sup>
D	Inside diameter of pipe, in.

## TWO PHASE FLOW DEFINITIONS FOR TWO PHASE FLOW

### Liquid Holdup

$$H_L = \frac{\text{volume of liquid in a pipe segment}}{\text{volume of pipe segment}}$$

### Gas Holdup or Gas Void Fraction

$$H_g = 1 - H_L$$

### No-Slip Liquid Holdup

$$\lambda_L = \frac{Q_L}{Q_L + Q_V}$$

### No-Slip Gas Holdup

$$\lambda_G = 1 - \lambda_L = \frac{Q_g}{Q_L + Q_g}$$

### Two-Phase Density

Calculation of the two-phase density requires knowledge of the liquid holdup, three equations are used:

$$\rho_s = \rho_L H_L + \rho_g H_g$$

$$\rho_n = \rho_L \lambda_L + \rho_g \lambda_g$$

$$\rho_k = \frac{\rho_L \lambda_L^2}{H_L} + \frac{\rho_g \lambda_g^2}{H_g}$$

TWO PHASE FLOW NOMENCLATURE	
$H_L$	Liquid holdup
$H_g$	Gas Holdup
$\lambda_L$	No-Slip liquid holdup
$\lambda_g$	No-Slip gas holdup
$Q$	volumetric flow rate in-situ conditions
$Q_L$	volume of liquid in a pipe segment
$Q_L + Q_V$	volume of pipe segment if the gas and liquid traveled at the same velocity (no-slippage)
$\rho$	Density
$\rho_s$	
$\rho_n$	
$\rho_k$	

### Superficial Velocity

The velocity the phase would exhibit if it flowed through the total cross section of the pipe.

$$\text{Superficial gas velocity} = v_{sg} = \frac{Q_g}{A}$$

$$\text{Superficial liquid velocity} = v_{sl} = \frac{Q_L}{A}$$

$$\text{Superficial two phase velocity} = v_M = v_{sl} + v_{sg}$$

### Slip Velocity

The slip velocity is the difference between the actual gas and liquid velocities,  $V_s$ :

$$v_s = v_g - v_L = \frac{v_{sg}}{H_g} - \frac{v_{sl}}{H_l}$$

Therefore:

$$\lambda_L = \frac{v_{sl}}{v_M}$$

## MODIFICATION OF THE PRESSURE GRADIENT EQUATION FOR TWO PHASE FLOW

Assume that the gas-liquid mixture can be considered homogeneous over a finite volume of pipe,

$$\text{The Elevation Component} = \left( \frac{dP}{dL} \right)_l = \frac{g}{g_c} \rho_s \sin \theta \quad \text{where } \rho_s = \rho_g H_g + \rho_l H_l$$

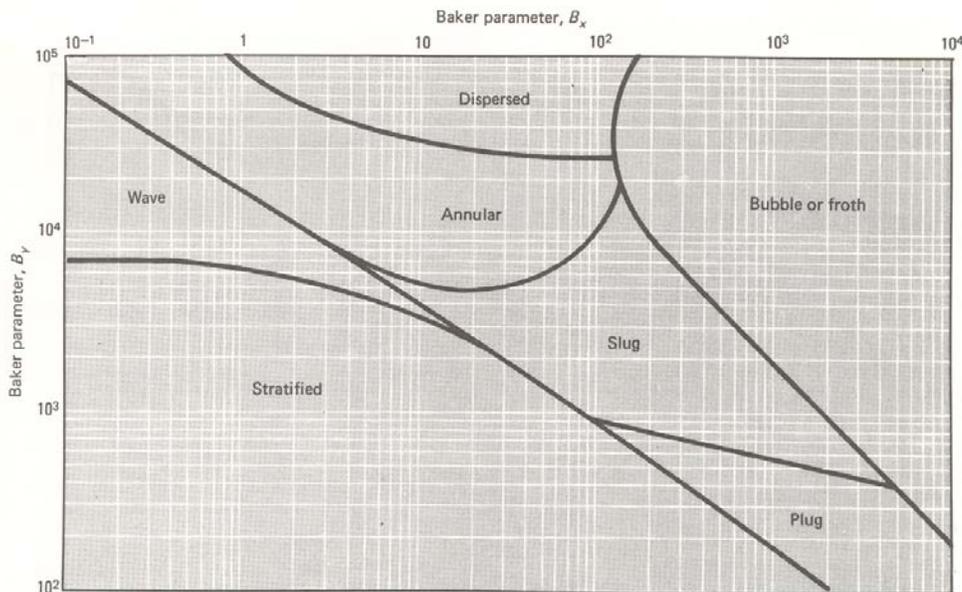
$$\text{The Friction Loss Component} = \left( \frac{dP}{dL} \right)_f = \frac{f_{tp} \rho_f v_M^2}{2 g_c D}$$

$$\text{The Acceleration Component} = \left( \frac{dP}{dL} \right)_{acc} = \text{frequently ignored.}$$

## THE BAKER METHOD FOR TWO PHASE FLOW CALCULATIONS

### Establishing the Two-Phase Flow Pattern

CHEMCAD establishes the two-phase flow pattern by using the following Baker Chart, this chart is accurate for horizontal flow:



Source: Deutsch, David J. and Chemical Engineering

where

$$B_x = 531 \left( \frac{W_l}{W_v} \right) \left( \frac{\rho_l \rho_v}{\rho_l^{2/3}} \right)^{0.5} \left( \frac{\mu_l^{1/3}}{\text{st}} \right)$$

$$B_y = 2.16 \left( \frac{W_v}{A(\rho_l \rho_v)^{0.5}} \right)$$

### Pressure Losses for Two-Phase Flow

The calculation of pressure losses for two-phase flow is based on the method of Lockhart-Martinelli. The general calculation procedure is outlined below:

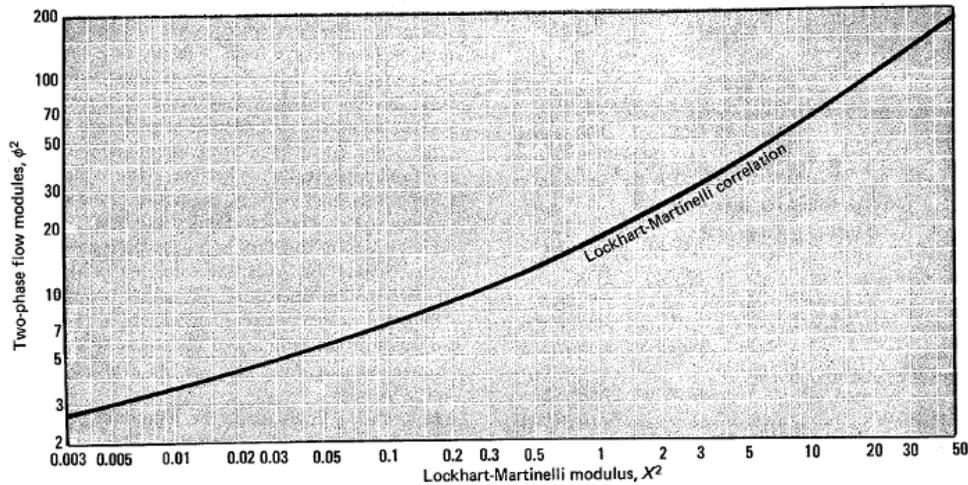
The pressure drop of the vapor is calculated by using only vapor flows in the pipe.

$$\Delta P_{100} = \Phi_v^2 \cdot (\Delta P_{100})_v \quad \dots (1)$$

The pressure drop is then corrected with the applicable correlations.

### Dispersed Flow

For dispersed flow  $\Phi^2$  is read off the Lockhart-Martinelli modules correlation like so:



*Source: Deutsch, David J. and Chemical Engineering*

where:

$$X^2 = \left( \frac{W_l}{W_v} \right)^{1.8} \left( \frac{\rho_v}{\rho_l} \right) \left( \frac{\mu_l}{\mu_v} \right)^{0.2} \quad \dots (2)$$

### Bubble Flow

$$\Phi = 14.2 \cdot \frac{X^{0.75}}{(W_l/A)^{0.1}} \quad \dots (3)$$

X is the square root of  $X^2$  as calculated in Equation 2 above.

### Slug Flow

$$\Phi = 1190 \cdot \frac{X^{0.815}}{(W_l/A)} \quad \dots (4)$$

X is calculated as in Equation 2

### Stratified Flow

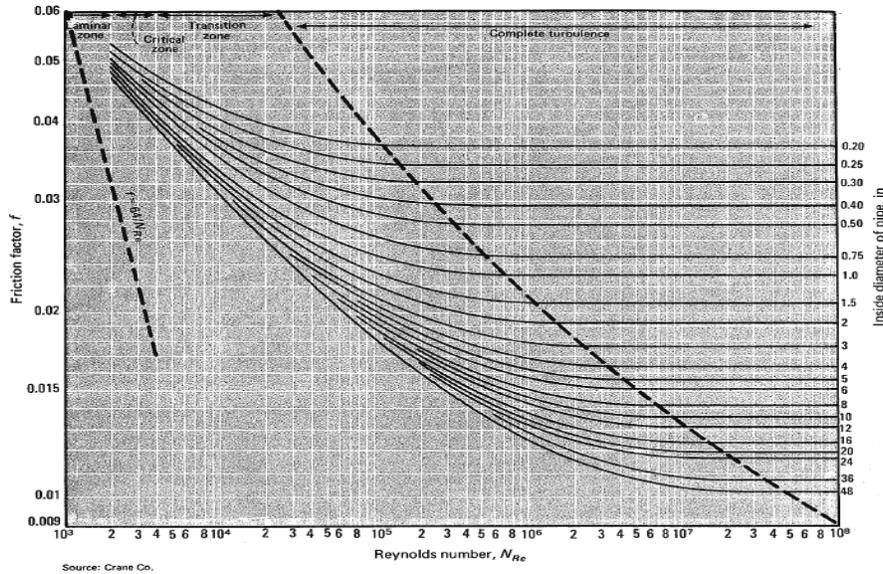
$$\Phi = 15400 \cdot \frac{X}{(W_l/A)^{0.8}} \quad \dots (5)$$

**Wave Flow**

First calculated  $H_x$ :

$$H_x = \left( \frac{W_l}{W_v} \right) \left( \frac{\mu_l}{\mu_v} \right) \dots (6)$$

then find  $f_H$ , using the following chart:



Source: Deutsch, David J. and Chemical Engineering

the pressure drop is then,

$$\Delta P_{100} = \frac{0.000336 f_H (W_v)^2}{D^5 \rho_v} \dots (7)$$

**Plug Flow**

$$\Phi = 27.315 * \frac{X^{0.855}}{(W_l/A)^{0.17}} \dots (8)$$

X is calculated in Equation 2.

**Annular Flow**

$$\Phi = aX^b \dots (9)$$

X is calculated in Equation 2.

TWO PHASE FLOW PRESSURE DROP NOMENCLATURE	
A	Internal cross section of pipe, ft <sup>2</sup>
D	Internal diameter of pipe, ft.
L	Equivalent length of pipe and fittings, ft.
W	Flow rate, lb/h
ΔP	Pressure loss, psi
μ	Viscosity, cp
ρ	Density, lb/ft <sup>3</sup>
ST	Surface tension, dyne/cm
l	Subscript refers to liquid
v	Subscript refers to vapor
100	Subscript refers to 100 ft. of pipe

**Accounting for elevation differences**

The elevation component of pressure drop is found using the Flanigan method. Within this method, the elevation component is calculated using the general equation below:

$$\Delta P_1 = \frac{\rho_L \cdot H_{Lf}}{144} \sum Z_e$$

ELEVATION DIFFERENCE NOMENCLATURE	
$\Delta P$	Pressure drop
$\rho_L$	Density of the liquid
$H_{Lf}$	$= \frac{1}{1 + 0.3264(v_{sg})^{1.006}}$
$Z_e$	Elevation changes
$v_{sg}$	Velocity of the gas

**BAKER METHOD PROCEDURE****1) Assumptions**

- a. Two phase flow is isothermal
- b. Turbulence exists in both phases
- c. Vapor and liquid move at the same speed
- d.  $\Delta P$  is not greater than 10% of  $P_{in}$ .

**2) Procedure**

- a. Determine flow regime using Baker Chart
- b. Calculate

$$\Delta P_{2\Phi} = \Delta P_v \cdot \Phi^2$$

**9) Two Phase Flow Regime**

TYPE	DESCRIPTION
Bubble (froth)	Liquid with dispersed bubbles of gas
Plug	Alternating plugs of gas and liquid in the upper section of pipe.
Stratified	Liquid on the lower and gas on the upper section of the pipe separated by a smooth interface.
Wave	Same as stratified, except separated by a wavy interface traveling in the same direction of flow
Slug	Similar to stratified, except the gas periodically picks up a wave and forms a bubbly plug. This flow can cause severe and dangerous vibrations because of the impact of the high-velocity slugs against the equipment.
Annular	Gas in the center and liquid on the outer portion of the pipe.
Spray (dispersed)	Liquid droplets in the gas.

$$a) \quad B_y = 2.16 \cdot \frac{W_v}{A \sqrt{\rho_l \rho_v}}$$

- $w_v$  Mass flow rate of the vapor  
 $A$  Pipe cross-sectional area  
 $\rho_l$  Density of liquid  
 $\rho_v$  Density of vapor

$$b) \quad B_x = 531 \left( \frac{W_l}{W_v} \right) \left( \frac{\sqrt{\rho_l \rho_v}}{\rho_l^{2/3}} \right) \left( \frac{\mu_l^{1/3}}{ST} \right)$$

- $w_l$  Mass flow rate of liquid  
 $\mu_l$  Liquid viscosity  
 $ST$  Liquid surface tension

- c. Using  $B_y$  and  $B_x$ , read flow regime from Baker Chart.

#### 4. Calculate $\Delta P_{v,100}$

$$Re_v = \frac{6.31 W_v}{D \mu_v}$$

$f_r$  is then read from the Moody friction factor chart.

$$\Delta P_{v,100} = \frac{0.000336 \cdot f_v \cdot w_v^2}{D^5 \cdot \rho_v}$$

#### 5. Calculate Lockhart-Martinelli Parameter, $X^2$

$$Re_l = \frac{6.31 W_l}{D \mu_l}$$

$f$  is read from the Moody friction factor chart.

$$X^2 = \left( \frac{W_l}{W_v} \right)^{1.8} \left( \frac{\rho_v}{\rho_l} \right) \left( \frac{f_l}{f_v} \right)^{0.2}$$

$$X = \sqrt{X^2}$$

**6. Calculate  $\Phi^2$** a. Dispersed flow: $\Phi^2$  read from Lockhart-Martinelli chart.b. Bubble flow:

$$\Phi = \frac{14.2 X^{0.75}}{(W_l/A)^{0.1}}$$

c. Slug flow:

$$\Phi = \frac{1190 X^{0.815}}{(W_l/A)}$$

d. Stratified flow: (Horizontal Pipe)

$$\Phi = \frac{15400 X}{(W_l/A)^{0.8}}$$

e. Plug flow: (Horizontal Pipe)

$$\Phi = \frac{27.315 X^{0.855}}{(W_l/A)^{0.17}}$$

f. Annular

$$\Phi = a \cdot X^b$$

$$a = 4.8 - 0.3125 D$$

$$b = 0.343 - 0.021 D$$

D = pipe ID, inches (for D &gt; 12; use D = 10)

g. Wave flow

$$H_x = \left( \frac{W_l}{W_v} \right) \left( \frac{\mu_l}{\mu_v} \right)$$

Read  $f_h$  from the Huntington Chart

$$\Delta P_{\text{wave},100} = \frac{0.000336 f_H (W_v)^2}{D^5 \rho_v}$$

**7. Calculate  $\Delta P_{2\Phi,100}$ :**

$$\Delta P_{2\Phi,100} = \Delta P_{v,100} \cdot \Phi^2$$

## BEGGS AND BRILL METHOD FOR TWO PHASE FLOW CALCULATIONS

The Beggs and Brill correlation was developed from experimental data obtained in a small-scale test facility. The facility consisted of 1-inch and 1.5-inch sections of acrylic pipe 90 feet long. The pipe could be inclined at any angle. The parameters studied and their range of variation were:

1. Gas flow rates of 0 to 300 Mscf/D
2. Liquid flow rates of 0 to 30 gpm
3. Average system pressure of 35 to 95 psia
4. Pipe diameter of 1 and 1.5 inches
5. Liquid hold-up of 0 to 0.87
6. Pressure gradients of 0 to 0.8 psi/ft.
7. Inclination angles of  $-90$  degrees to  $+90$  degrees

Fluids used were air and water. For each pipe size, liquid and gas rates were varied so that all flow patterns were observed. After a particular set of flow rates was set, the angle of the pipe was varied through the range of angles so that the effect of angle on holdup and pressure gradient could be observed. Liquid holdup and pressure gradient were measured at angles from the horizontal at 0, plus and minus 5, 10, 15, 20, 35, 55, 75, and 90 degrees. The correlations were developed from 584 measured tests.

### HORIZONTAL FLOW

Different correlations for liquid holdup are presented for each of the three horizontal flow regimes. The liquid holdup, which would exist if the pipe were horizontal, is first calculated and then corrected for the actual pipe inclination. Three of the horizontal flow patterns are illustrated on Figure 7-1. A fourth, the transition region, was added by Beggs and Brill to produce the map shown on Figure 7-2. The variation of liquid holdup with pipe inclination is shown on Figure 7-3 for three of the tests. The holdup was found to have a maximum at approximately  $+50$  degrees from the horizontal and a minimum at approximately  $-50$  degrees. A two-phase friction factor is calculated using equations, which are independent of flow regime, but dependent on holdup. A graph of a normalized friction factor as a function of liquid holdup and input liquid content is given on Figure 7-4.

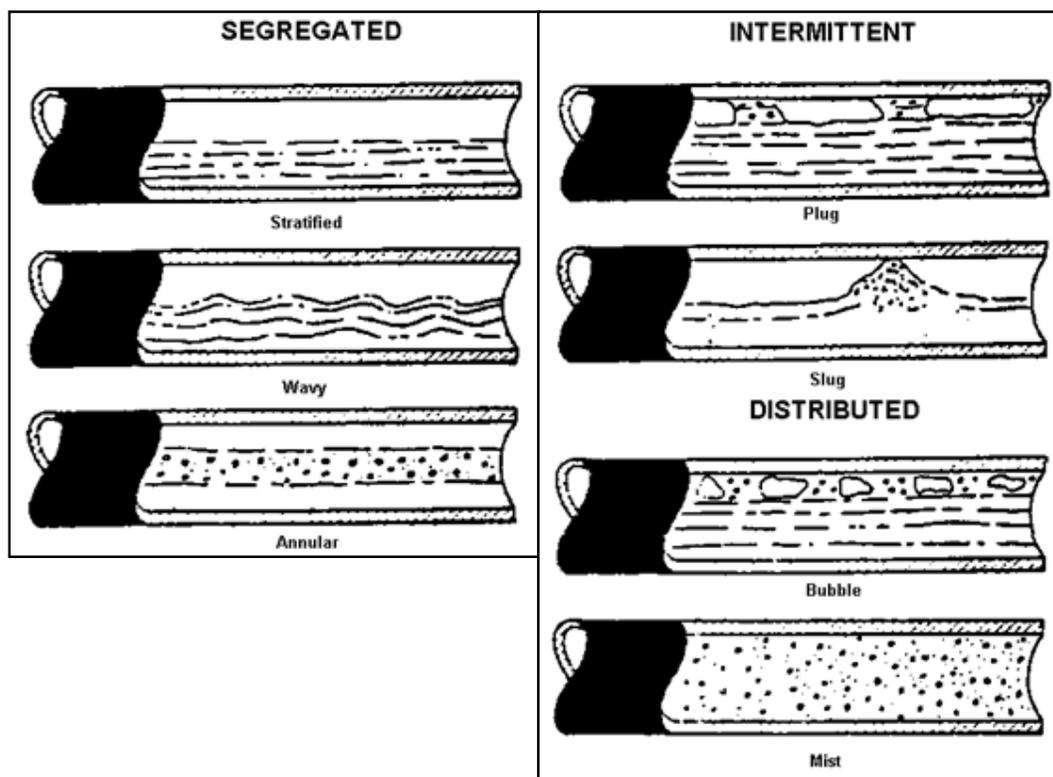


Figure 7-1: Horizontal Flow Patterns

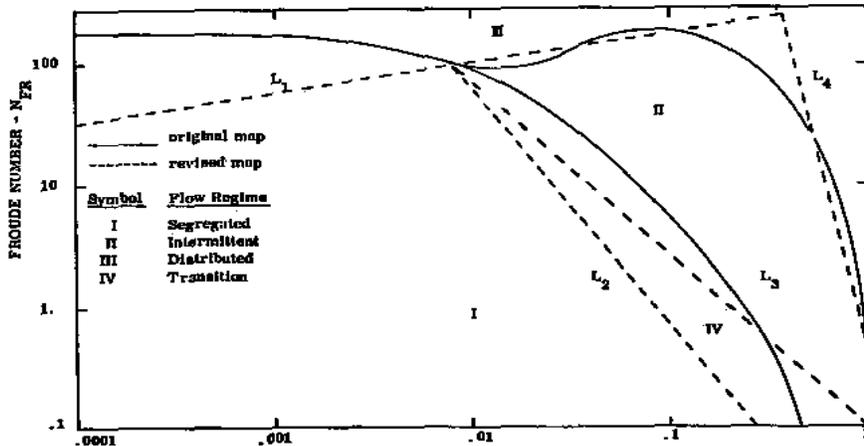


Figure 7-2 Input Liquid Content Horizontal Flow Pattern Map

Figure 7-2: Horizontal Flow Pattern Map

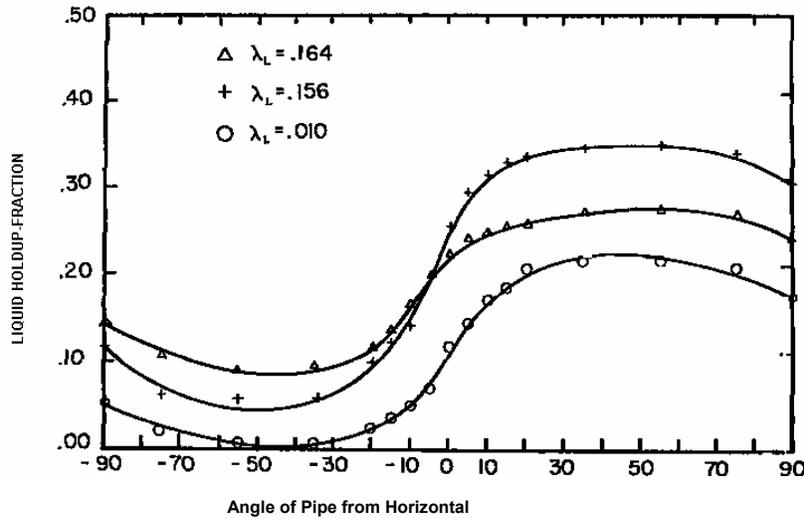


Figure 7-3: Liquid Holdup vs. Angle

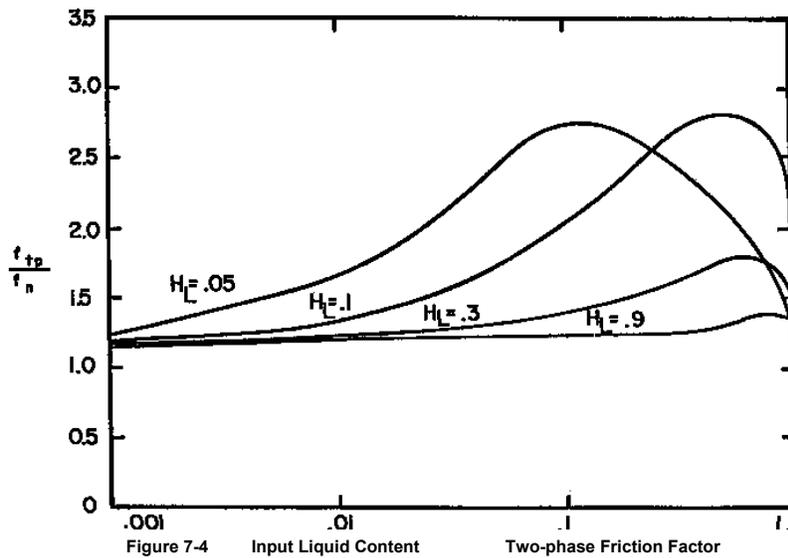


Figure 7-4: Two-phase Friction Factor

**FLOW REGIME DETERMINATION**

The following variables are used to determine which flow regime would exist if the pipe was in a horizontal position. This flow regime is a correlating parameter and gives no information about the actual flow regime unless the pipe is completely horizontal.

$$Fr = \frac{V_m^2}{gd} \quad (\text{Froude No.})$$

$$\lambda_L = \frac{V_{SL}}{V_m}$$

$$L_1 = 316 \lambda_L^{0.302}$$

$$L_2 = 0.0009252 \lambda_L^{-2.4684}$$

$$L_3 = 0.10 \lambda_L^{-1.4516}$$

$$L_4 = 0.5 \lambda_L^{-6.738}$$

The horizontal flow regime limits are:

<b>Segregated</b>	$\lambda_L < 0.01$ and $FR < L_1$
	or $\lambda_L > 0.01$ and $FR < L_2$
<b>Transition</b>	$\lambda_L > 0.01$ and $L_2 < Fr < L_3$
<b>Intermittent</b>	$0.01 < \lambda_L < 0.4$ and $L_3 < Fr < L_1$
	or $\lambda_L > 0.4$ and $L_3 < Fr < L_4$
<b>Distributed</b>	$\lambda_L < 0.4$ and $Fr > L_1$
	or $\lambda_L > 0.4$ and $Fr > L_4$

When the flow falls in the transition region, the liquid holdup must be calculated using both the segregated and intermittent equations, and interpolated using the following weighting factors:

$$HL(\text{transition}) = A \times HL(\text{segregated}) + B \times HL(\text{intermittent})$$

where:

$$A = \frac{L_3 - Fr}{L_3 - L_2}$$

$$B = 1 - A$$

**TWO-PHASE DENSITY**

$$\rho_s = \rho_L H_L + \rho_g H_g$$

$$H_{L(0)} = \phi H_{L(0)}$$

$H_{L(0)}$  = The holdup which would exist at the same conditions in a horizontal pipe

$$H_{L(0)} = \frac{a \lambda_L b}{F_R c}$$

For  $H_{L(0)} > \lambda_L$ :

Flow Regime	a	b	c
For segregated flow	0.98	0.4846	0.0868
For intermittent flow	0.845	0.5351	0.0173
For distributed flow	1.065	0.5824	0.0609
( $H_{L(0)}$ must be $\geq \lambda_L$ )			

The factor for correlating the holdup for the effect of pipe inclination is given by:

$$\Psi = 1 + C \left[ \sin(1.8\theta) - 0.333 \sin^3(1.8\theta) \right]$$

$\theta$  = the actual angle of the pipe from horizontal

$$C = (1 - \lambda_L) \ln(D \lambda_L^e N_{LV}^f Fr^g)$$

Flow Regime	d	e	f	g
For segregated uphill	0.011	-3.768	3.539	-1.614
For intermittent uphill	2.96	0.305	-0.4473	0.0978
For distributed uphill	No correction $c = 0$ , $\Psi = 1$ , $H_L \neq f(\theta)$			
For all downhill	4.7	-0.3692	0.1244	-0.5056

with the restriction that  $C > 0$ .

**FRICITION FACTOR**

$$\left( \frac{dP}{dZ} \right)_f = \frac{f_{tp} \cdot \rho_n \cdot V_m^2}{2g_c \cdot D}$$

$$\rho_n = \rho_L \cdot \lambda_L + \rho_g \cdot \lambda_g = \rho_L \cdot \lambda + \rho_g \cdot (1 - \lambda_L)$$

$$\mu_n = \mu_L \cdot \lambda_L + m_g \cdot \lambda_g = \mu_L \cdot \lambda_L + m_g \cdot (1 - \lambda_L)$$

$$N_{Ren} = \frac{\rho_n \cdot V_m \cdot d}{\mu_n}$$

$$f_{tp} = f_n \cdot X \quad \text{where} \quad X = e^s$$

$$s = \frac{\ln y}{-0.0523 + 3.182 \cdot \ln y - 0.8725 \cdot (\ln y)^2 + 0.01853 \cdot (\ln y)^4}$$

$$y = \frac{\lambda_L}{H_{L(\phi)}^2}$$

The value of  $y$  becomes unbounded at a point in the interval  $1 < y < 1.2$ ; and for  $y$  in this interval, the function  $S$  is calculated from:

$$S = \ln(2.2y - 1.2)$$

**ACCELERATION TERM**

$$\left(\frac{dP}{dZ}\right)_{acc} = \frac{\rho_s \cdot V_m \cdot V_{sg}}{g_c \cdot P} \cdot \frac{dP}{dZ} = E_k \cdot \frac{dP}{dZ}$$

$$\text{Since } \left(\frac{dP}{dZ}\right) = \left(\frac{dP}{dZ}\right)_{el} + \left(\frac{dP}{dZ}\right)_f + \left(\frac{dP}{dZ}\right)_{acc}$$

$$\left(\frac{dP}{dZ}\right) = \left(\frac{dP}{dZ}\right)_{el} + \left(\frac{dP}{dZ}\right)_f + E_k \cdot \left(\frac{dP}{dZ}\right)$$

$$\frac{dP}{dZ} - E_k \cdot \frac{dP}{dZ} = \left(\frac{dP}{dZ}\right)_{el} + \left(\frac{dP}{dZ}\right)_f$$

$$\left(\frac{dP}{dZ}\right) = \frac{\left(\frac{dP}{dZ}\right)_{el} + \left(\frac{dP}{dZ}\right)_f}{1 - E_k}$$

**ELEVATION TERM**

$$\left(\frac{dP}{dZ}\right)_{el} = \frac{\rho_s \cdot g}{g_c}$$

$g_c$       Gravitational constant

$g$         Acceleration due to gravity

## VERTICAL FLOW

For vertical flow,  $\theta = 1$ , and  $dL = dZ$ , so;

$$\frac{dP}{dL} = \frac{dP}{dZ} = \left( \frac{dP}{dZ} \right)_{el} + \left( \frac{dP}{dZ} \right)_f + \left( \frac{dP}{dZ} \right)_{acc}$$

The pressure drop caused by elevation change depends on the density of the two-phase mixture and is usually calculated using a liquid holdup value. Except for high velocity situations, most of the pressure drop in vertical flow is caused by elevation change. The frictional pressure loss requires evaluation of the two-phase friction factor. The acceleration loss is usually ignored except for high velocity cases.

## VERTICAL FLOW REGIMES

### Bubble flow

In bubble flow the pipe is almost completely filled with liquid and the free gas phase is present in small bubbles. The bubbles move at different velocities and except for their density, have little effect on the pressure gradient. The wall of the pipe is always contacted by the liquid phase.

### Slug flow

In slug flow the gas phase is more pronounced. Although the liquid phase is still continuous, the gas bubbles coalesce and form plugs or slugs, which almost fill the pipe cross-section. The gas bubble velocity is greater than that of the liquid. The liquid in the film around the bubble may move downward at low velocities. Both the gas and liquid have significant effects on the pressure gradient.

### Transition flow

The change from a continuous liquid phase to a continuous gas phase is called transition flow. The gas bubbles may join and liquid may be entrained in the bubbles. Although the liquid effects are significant, the gas phase effects are predominant.

### Mist flow

In mist flow, the gas phase is continuous and the bulk of the liquid is entrained as droplets in the gas phase. The pipe wall is coated with a liquid film, but the gas phase predominantly controls the pressure gradient.

Illustrations of bubble, slug, transition, and mist flow are shown below.

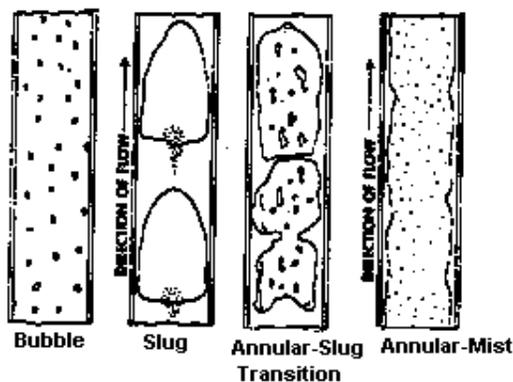


Figure 7-5: Vertical Flow Patterns

A typical two-phase flow regime map is shown on Figure 7-6.

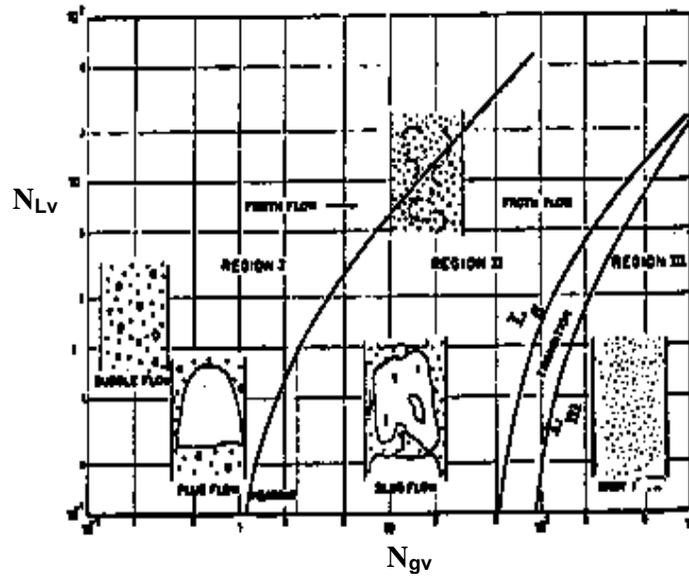


Figure 7-6: Flow Regime Map

Where the following are the procedure equations for calculating vertical flow with Beggs and Bill method.

$$N_{LV} = V_{SL} \left( \frac{\rho_L}{gST} \right)^{\frac{1}{4}}$$

$$N_{GV} = V_{SV} \left( \frac{\rho_L}{gST} \right)^{\frac{1}{4}}$$

$$N_D = d \left( \frac{\rho_L g}{ST} \right)^{\frac{1}{2}}$$

$$N_L = \mu_L \left( \frac{g}{\rho_L ST^3} \right)^{\frac{1}{4}}$$

$$L_S = 50 + 36 N_{LV}$$

$$L_M = 75 + 84 N_{LV}^{0.75}$$

**SET UP CALCULATIONS**

1) Given

- a. Inlet  $m_v, m_L, \rho_v, \rho_L, \mu_v, \mu_L, ST$
- b. Pipe inside diameter, D
- c. Acceleration of gravity,  $g, \frac{ft}{s^2}$
- d. Inlet pressure, P

2) Calculate

$$3) A = \frac{\pi D^2}{4}$$

4) Calculate Volumetric Flow rates

$$5) Q_g = \frac{m_v}{\rho_v} \quad \& \quad Q_L = \frac{m_L}{\rho_L}$$

6) Calculate Superficial Velocities  $V_{sg}, V_{sl}, V_m$ 

$$V_{sg} = \frac{Q_g}{A} \quad V_{sl} = \frac{Q_L}{A} \quad V_m = V_{sg} + V_{sl}$$

**DETERMINE FLOW REGIME**

5. Calculate Froude No.

$$7) FR = \frac{v_m^2}{gD}$$

6. Calculate No-slip liquid holdup

$$8) \lambda_L = \frac{V_{sl}}{v_m}$$

7. Calculate dimension less "L" parameters

$$L_1 = 316 \lambda_L^{0.302}$$

$$L_2 = 0.0009252 \lambda_L^{-2.4684}$$

$$L_3 = 0.10 \lambda_L^{-1.4516}$$

$$L_4 = 0.5 \lambda_L^{-6.738}$$

8. Apply flow regime rules

a. Is  $\lambda_L < 0.01$  and  $Fr < L_1$ ?

No                      Yes → Segregated flow

↓

b. Is  $\lambda_L \geq 0.01$  and  $Fr < L_2$ ?

No                      Yes → Segregated flow

↓

c. Is  $\lambda_L \geq 0.01$  and  $L_2 \leq Fr \leq L_3$ ?

No                      Yes → Transition flow

↓

d. Is  $0.01 \leq \lambda_L < 0.4$  and  $L_3 < Fr \leq L_1$ ?

No                      Yes → Intermittent flow

↓

e. Is  $\lambda_L \geq 0.4$  and  $L_3 < Fr \leq L_4$ ?

No                      Yes → Intermittent flow

↓

f. Is  $\lambda_L < 0.4$  and  $Fr \geq L_1$ ?  
 No Yes → Distributed flow  
 ↓

g. Is  $\lambda_L \geq 0.4$  and  $Fr > L_4$ ?  
 No Yes → Distributed flow  
 ↓  
 Outside the range of the Beggs - Brill method

**DETERMINE LIQUID HOLDUP & TWO-PHASE DENSITY**

9. Calculate  $H_{L(0)}$  (the holdup which would exist at the same conditions in a horizontal pipe.)

$$H_{2(0)} = \frac{a \lambda_L^b}{Fr^c}$$

Flow Regime	a	b	c
For segregated flow	0.98	0.4846	0.0868
For intermittent flow	0.845	0.5351	0.0173
For distributed flow	1.065	0.5824	0.0609
( $H_{L(0)}$ must be $\geq \lambda_L$ )			

For transition flow calculate  $H_{L(\text{segregated})}$  and  $H_{L(\text{intermittent})}$ .

10. Calculate  $N_{LV}$  (the liquid velocity number)

$$N_{LV} = V_{sl} \left( \frac{\rho_L}{gST} \right)^{\frac{1}{4}}$$

11. Calculate c

$$c = (1 - \lambda_L) \cdot \ln \left( D^1 \cdot \lambda_L^e \cdot N_{LV}^f \cdot Fr^g \right)$$

Flow Regime	D <sup>1</sup>	e	f	g
For segregated uphill	0.011	-3.768	3.539	-1.614
For intermittent uphill	2.96	0.305	-0.4473	0.0978
For distributed uphill	No correction c = 0, $\Psi = 1$ , $H_L \neq f(\theta)$			
For all downhill	4.7	-0.3692	0.1244	-0.5056

For transition uphill flow, calculate c<sub>(segregated)</sub> and c<sub>(intermittent)</sub>. (C must be  $\geq 0$ )

12. Calculate  $\Psi$

$$\Psi = 1 + c [\sin(1.8\theta) - 0.333 \sin^3(1.8\theta)]$$

where  $\theta$  is the actual angle of the pipe from horizontal. For vertical flow  $\theta = 90$ .

13. Calculate the liquid holdup,  $H_{L(\theta)}$

For segregated, intermittent, and distributed flow:

$$H_{L(\theta)} = H_{L(0)} \cdot \Psi$$

For transition flow:

$$H_{L(\text{transition})} = A \cdot H_{L(\text{segregated})} + B \cdot H_{L(\text{intermittent})}$$

$$A = \frac{L_3 - Fr}{L_3 - L_2}$$

$$B = 1 - A$$

14. Calculate  $H_g = 1 - H_{L(\theta)}$

15. Calculate two phase density  $\rho_s = \rho_L \cdot H_{L(\theta)} + \rho_g \cdot H_g$

**DETERMINE ELEVATION TERM**

$$\left(\frac{dP}{dZ}\right)_{el} \left[ \frac{=}{=} \right] \text{psf/ft.}$$

$$16. \left(\frac{dP}{dZ}\right)_{el} = \frac{\rho_s \cdot g}{gc}$$

**DETERMINE FRICTION TERM**

$$\left(\frac{dP}{dZ}\right)_f$$

17. Calculate no-slip two-phase density

$$\rho_n = \rho_L \cdot \lambda_L + \rho_g \cdot \lambda_g = \rho_L \cdot \lambda + \rho_g \cdot (1 - \lambda_L)$$

18. Calculate no-slip viscosity,  $\mu_n$

$$\mu_n = \mu_L \cdot \lambda_L + \mu_g \cdot \lambda_g = \mu_L \cdot \lambda_L + \mu_g \cdot (1 - \lambda_L)$$

19. Calculate no-slip Reynolds No.

$$Re = \frac{\rho_n \cdot V_m \cdot d}{\mu_n}$$

20. Calculate no-slip friction factor

$$f_n = \frac{1}{\left[ 2 \cdot \log \left( \frac{Re}{4.5223 \cdot \log Re - 3.8215} \right) \right]^2}$$

21. Calculate

$$9) \quad y = \frac{\lambda_L}{\left( H_{L(\phi)} \right)^2}$$

22. Calculate

$$s = \frac{\ln(y)}{-0.0523 + 3.182 \cdot \ln y - 0.8725 \cdot (\ln(y))^2 + 0.01853 \cdot (\ln y)^4}$$

23. Calculate the two phase friction factor

$$f_{tp} = f_n \cdot e^s$$

24. Calculate the friction loss term,  $\left(\frac{dP}{dZ}\right)_f$

$$\left(\frac{dP}{dZ}\right)_f = \frac{f_{tp} \cdot \rho_n \cdot v_m^2}{2gc \cdot d}$$

**CALCULATE THE ACCELERATION TERM**

$$25. E_k = \frac{\rho_s \cdot v_m \cdot v_{sg}}{gc \cdot P}$$

**CALCULATE THE TOTAL PRESSURE GRADIENT**

$$\left(\frac{dP}{dZ}\right)$$

$$26. \frac{dP}{dZ} = \frac{\left(\frac{dP}{dZ}\right)_f + \left(\frac{dP}{dZ}\right)_{el}}{1 - E_k}$$

BEGGS AND BRILL METHOD NOMENCLATURE	
A	Flow area
D	Diameter
$E_k$	Beggs and Brill acceleration term
$F_r$	Froude number
f	Friction factor
g	Acceleration due to gravity
$g_c$	Gravitational constant
H	Holdup
h	enthalpy
L	Length
M	Molecular weight
P	pressure
$P_{LW}$	Losses due to irreversible processes
q	Heat loss (input)
Q	Volumetric flow rate
Re	Reynolds number
S	Entropy
SG	Specific gravity
ST	Surface tension
U	Internal energy
V	Volume
v	Velocity
W	Mass flow rate
X	Dimensionless flow pattern region coefficient
Y	Flow pattern region coefficient
Z	Elevation
$\lambda$	Slip
$\mu$	Viscosity
$\Phi$	Lockhart–Martinelli correlating parameter
$\rho$	Density
$\tau$	Shear force
$2\Phi$	Two phase

### 3-K (DARBY) METHOD OF SPECIFYING FLOW RESISTANCES

In the past, the flow resistance of valves and fittings has been represented by one or two numbers;  $L/D$ ,  $K$ ,  $K_f$ , etc. This method is accurate over a wide range of Reynolds numbers, but does not accurately reflect data over a wide range of sizes. Specifically, these methods tend to under predict the friction loss for pipes of larger diameters. Darby (2001) evaluated data from the literature for various valves, tees, and elbows and found that they can be represented more accurately by the following "3-K" equation:

$$K_f = \frac{K_1}{Re} + K_i \left( 1 + \frac{K_d}{D_n^{0.3}} \right)$$

$K_f$	Loss coefficient, dimensionless
$K_1, K_i, K_d$	Constants, can be found in the literature
Re	Reynolds number
$D_n$	Nominal diameter

### HEAT LOSS IN PIPING SYSTEMS

The PIPE module will calculate the heat lost (or gained) to the surroundings if the user enters the overall heat transfer coefficient,  $U$ , and the ambient temperature,  $T_a$ . The heat loss is calculated as follows:

$$Q = UA(T_a - T_b)$$

Q	The heat transferred to the surroundings (negative indicator heat lost from the piping system).
U	The overall heat transfer coefficient (user specified).
$T_a$	The ambient temperature (user specified).
$T_b$	The temperature of the bulk material in the pipe.

### NET POSITIVE SUCTION HEAD

Centrifugal pumps require vapor free liquid flow in the suction line at the entrance to the impeller vanes to operate satisfactorily. Reciprocating pumps require such flow right after the inlet valve.

A centrifugal pump cannot pump vapor-liquid mixtures. A positive-displacement pump can pump liquids containing vapor but its volumetric efficiency will be greatly reduced, even if the weight percentage of vapor is a very small proportion of total flow.

The pressure gradient along the liquid path through a pump has characteristics similar to those of a control valve or orifice. Instead of a pressure loss, however, the downstream pressure will become much greater than the suction pressure. In both the pump-suction and the pump discharge, the minimum pressure should not drop below the vapor pressure of the liquid at the flowing temperature.

#### NPSH<sub>r</sub>

If the pressure in a centrifugal pump falls below the vapor pressure of the liquid at the flowing temperature, bubbles form just before the inlet edge of the impeller vanes. Because of the rapid increase in pressure within the impeller, the bubbles then collapse. This phenomenon is called cavitation. It can reduce pump efficiency and cause noise, vibration, impeller wear and damage.

To avoid cavitation, pump manufacturers require a positive suction pressure called net positive suction head,  $NPSH_r$ .  $NPSH_r$  is the pressure above the vapor pressure of the liquid being pumped, measured as the pump-suction flange on a gage at the pump centerline. It is the pressure necessary to overcome friction and turbulence losses between the suction nozzle and the entrance to the impeller, and is usually given as head of water on pumps' head-capacity curves.  $NPSH_r$  for a pump will increase with flow rate, and considerably increases with rotating speed of the impeller:

$$(NPSH_r)_2 = \left( \frac{n_2}{n_1} \right)^2 (NPSH_r)_1 \quad \dots (1)$$

For flows below about 20 to 25% of rated capacity (near the maximum efficiency points),  $NPSH_r$  requirements can be higher than those for normal flow. This is especially true with high-capacity, low-head pumps.

**NPSH<sub>a</sub>**

Available net positive suction head,  $NPSH_a$ , is the potential energy of an installation, expressed in feet of liquid head, and can be calculated from:

$$P = \frac{(NPSH_a)\rho}{144} \quad \dots (2)$$

If we let,

$$\rho = (SG)\rho_w$$

Equation 2 becomes:

$$(NPSH_a) = 2.31 \frac{P}{SG} \quad \dots (3)$$

NET POSITIVE SUCTION HEAD NOMENCLATURE	
$(NPSH_r)_2$	The required net positive suction head if pump speed is increased to $n_2$ rpm
$(NPSH_r)_1$	Obtained from the pump's head-capacity curve
$n_1$	Obtained from the pump's head-capacity curve
$n_2$	Obtained from the pump's head-capacity curve
P	Pressure above vapor pressure at the centerline of the suction flange, psia.
$(NPSH_a)$	Available net positive suction head
$\rho$	Density of fluid at flowing temperature, lb/ft <sup>3</sup>
SG	Specific gravity of liquid at flowing temperature
$\rho_w$	Density of water at 60 °F, 62.37 lb/ft <sup>3</sup>

To ensure reliable pump operation,  $NPSH_a$  should equal or exceed  $NPSH_r$ . Thus, process equipment must be elevated at least as high as the sum of the pump's  $NPSH_r$ , plus the friction-head losses in the suction line when saturated liquids are pumped.

At the suction nozzle, velocity head is also a positive energy component. However, this is not included in NPSH diagrams for pumps made in the U.S. Consequently, in calculations for  $NPSH_a$ , velocity head does not have to be considered.

According to the Hydraulic Institute "The NPSH requirements of centrifugal pumps are normally determined on the basis of handling water. It is recognized that when pumping hydrocarbons, the  $NPSH_r$  required to obtain satisfactory operation may be reduced for certain conditions. However, it is customary to use manufacturers' data for the minimum required  $NPSH_r$  when handling hydrocarbons having specific gravities less than 1. This means that:

1. Both available and required  $NPSH_r$  do not vary with the liquid's specific gravity.
2. For a given centrifugal pump, one curve of required  $NPSH_r$  can be used with all normal specific gravities.

The effect of specific gravity can be explained as follows: Looking at the analogy of NPSH as a force that pushes liquid into a pump, and assuming a pump that has an  $NPSH_r$  of 8 ft. at a given capacity, we can show that a head of 8 ft. of water equals 3.45 psi, while a head of 8 ft. of gasoline equals 2.60 psi. It would appear that the force pushing liquid into the pump decreases with specific gravity. However, the mass of liquid being accelerated has decreased by the same proportion.

When choosing between two pumps of equal capacity, the one having the higher  $NPSH_r$  will cost less. But the less-expensive pump will be the economic choice only if its cost advantage is not offset by the expense of elevating the process equipment. Often, a vertical pump will be specified to bring equipment closer to grade elevation.

If a reciprocating pump is to transfer saturated liquid and if we desire maximum volumetric efficiency, we make analyses and calculations for NPSH conditions similar to those for centrifugal pumps. Resistance at the inlet valve of a reciprocating pump is greater than the entrance loss to an impeller. Due to pulsating liquid flow, we must also take into account increased line resistance.

## PUMPING SATURATED LIQUIDS

For subcooled liquids, the available pressure at the suction nozzle is usually more than adequate for NPSH and line losses. Consequently, suction vessels can be close to grade elevation, and the pipe configuration is not as critical.

Many process pumps handle saturated liquids. Here, the suction vessel and draw off nozzle are elevated to meet the NPSH requirement, plus a distance equal to the static height of the line losses.

We now repeat an important rule for the design engineer. At no point in the suction line should pressure fall (due to piping and components resistance) below the vapor pressure of the liquid at the flowing temperature. A horizontal pipe run immediately after the draw off nozzle is undesirable because a gradual pressure reduction (in proportion to the length of horizontal pipe) can cause the liquid to vaporize. Also, if the liquid level in the suction vessel is low, the available static head might not be sufficient to compensate for the exit loss and the friction loss in the horizontal pipe section. This is especially true with light, volatile liquids.

A horizontal-pipe segment after a vertical drop is not as bad because static head is available at the beginning of the horizontal section. In a vertical segment of suction piping, the potential energy is usually much greater than the energy consumed by friction. Thus, when pumping hot liquids, a vertical pipe run after the draw off nozzle is most desirable; and with light hydrocarbons, it can become essential.

To avoid vaporization at the vessel's draw off nozzle, the minimum static head of liquid above the nozzle should be greater than the entrance loss to the suction pipe. Cavitation at the draw off nozzle can cause undesirable pressure surges and inconsistent control conditions.

The exit resistance of a nozzle can be expressed:

$$h_L = \frac{Lv^2}{2g} \quad \dots (4)$$

PUMPING SATURATED LIQUIDS NOMENCLATURE	
$h_L$	Liquid level above draw off nozzle
L	Length, ft
K	0.5, the resistance coefficient for a sharp-edged pipe entrance
$\rho_w$	Density of water at 60o F, 62.37 lb/ft <sup>3</sup>
v	Velocity, ft/s
g	Gravitational constant, 32.2 ft/s <sup>2</sup>

Using a safety factor of four, the minimum liquid head above the draw off nozzle will be

$$h_L = 4K.$$

## PIPE SIZING USING THE KENT METHOD

If you choose the general pipe sizing method of CHEMCAD, the diameter is determined based upon "typical" velocities which are a function of fluid density. These typical velocities result in the following equations for diameter:

For liquids:

$$D = 2.607 \left( \frac{W}{\rho} \right)^{0.434}$$

For gases:

$$D = 1.065 \frac{W^{0.408}}{\rho^{0.343}}$$

D Pipe internal diameter, in.

W Flow rate 1000 lb/h

$\rho$  Density lb/ft<sup>3</sup>

**PIPING DESIGN CONSIDERATIONS**

Industry practice for initial design of piping systems is based on economic velocity or allowable pressure drop  $\Delta P/100\text{ft}$ . Once detailed isometrics are available the design will be adjusted to satisfy local site conditions.

Reasonable Velocities for Flow of Fluids through Pipes (Reference Crane 410M)				
Service Conditions	Fluid	Reasonable Velocities		Pressure Drop
		m/s	ft/s	kPa / m
Boiler Feed	Water	2.4 to 4.6	8 to 15	
Pump Suction and Drain	Water	1.2 to 2.1	4 to 7	
General Service	Liquids pumped, non viscous	1.0 to 3.0	3.2 to 10	0.05
Heating Short Lines	Saturated Steam 0 to 1.7 bar	20 to 30	65 to 100	
Process piping	Saturated Steam 1.7 and up	30 to 60	100 to 200	
Boiler and turbine leads	Superheated Steam 14 and up	30 to 100	100 to 325	
Process piping	Gases and Vapours	15 to 30	50 to 100	0.02% line pressure
Process piping	Liquids gravity flow			0.05

Reasonable velocities based on pipe diameter (Process Plant Design, Backhurst Harker p235)

Pump suction line for  $d$  in  $(d/6 + 1.3)$  ft/s and  $d$  mm  $(d/500 + 0.4)$  m/s  
 Pump discharge line for  $d$  in  $(d/3 + 5)$  ft/s and  $d$  mm  $(d/250 + 1.5)$  m/s  
 Steam or gas  $d$  in  $20d$  ft/s and  $d$  mm  $0.24d$  m/s

Heuristics for process design (Reference W.D.Seader, J.D.Seider and D.R.Lewin, "Process Design Principles") are also given:

Liquid Pump suction  $(1.3 + d/6)$  ft/s 0.4 psi /100 ft  
 Liquid Pump discharge  $(5.0 + d/3)$  ft/s 2.0 psi / 100 ft  
 Steam or gas  $(20d)$  ft/s 0.5 psi / 100 ft

## INSTRUMENTATION

### Control Valves

Control valve pressure drop needs to be at a reasonable % of total system pressure drop to provide good control. If too low, ie valve oversized, the control valve opening will be small leading to unstable control; if too high flow could be limited leading to throughput concerns. A general "Rule of Thumb" is for a full sized trim control valve to be half the line size.

To model piping systems, involving special items such as flow meters, CHEMCAD provides a facility under the Valve Data entry Window in the Pipe UnitOp for resistance parameters to be entered in various formats. The following guidelines should be considered when selecting flow meter sizes.

### Magnetic Flow Meter

Velocity Limits for Flow of Fluids through Magnetic Flow Meters (Foxboro Bulletin)				
Service Conditions	Fluid	Reasonable Velocities		Pressure Drop
		m/s	ft/s	kPa / m
General Service and Process Piping	Liquids pumped, non viscous	0.9 to 4.6	3.0 to 15	Line size meter Same ΔP as pipe
	Erosive Slurries	0.9 to 4.6	3 to 6	
	Coating forming liquids	1.8 to 4.6	6 to 16	

### Mass Flow Meter

Because of the wide turndown capability of Coriolis flowmeters (30:1 to as high as 200:1), the same flow can be measured by two or three different sized flow tubes subject to accuracy requirements. Using the smallest possible meter lowers the initial cost and reduces coating build-up, but increases erosion/corrosion rates and head loss.

Using a meter that is smaller than line size is acceptable if the process fluid is clean with a low viscosity. However on corrosive, viscous, or abrasive slurry services, this practice may cause reduced operational life. Flow tube sizes and corresponding pressure drops, inaccuracies, and flow velocities can be obtained from software provided by the manufacturer.

Different Coriolis meter principles incur different pressure drops, but in general they require more than traditional volumetric meters, which usually operate at less than 10 psi. This higher head loss is due to the reduced tubing diameter and the circuitous path of flow. Head loss can be of concern if the meter is installed in a low-pressure system, or if there is a potential for cavitation or flashing, or if the fluid viscosity is very high.

### Vortex Shedding Meter

Measurable flow velocities on liquids are in the general range of 0.5 to 9.0 m/s ( 1.5 to 32 ft/s).

On gas or steam the flow velocities are in the range  $\sqrt{\frac{74}{\rho}}$  to 79 m/s (  $\sqrt{\frac{50}{\rho}}$  to 260 ft/s)

Where  $\rho$  fluid density (kg/m<sup>3</sup> or lb/ft<sup>3</sup>)

Process fluid viscosity requires the Reynolds Number to be greater than 20000

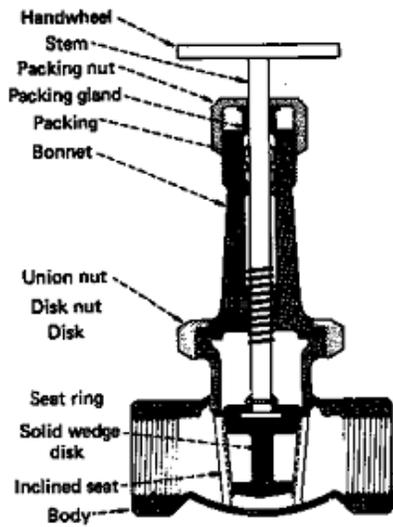
Linear performance is achieved for Reynolds Number in the range 20000 to 7.0 E06

**CHAPTER 8****DESCRIPTION OF VALVES AND FITTINGS**

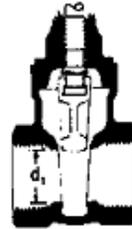
VALVES		FLANGED FITTINGS	
1	Gate valve	27	Standard elbow 90 deg
2	Globe seat flat, bevel, plug	28	Standard elbow 45 deg
3	Globe wing/pin guided disk	29	Standard elbow 90 long R
4	Angle, no obstruction	30	Return bend 180 deg close
5	Angle wing/pin guided disc	31	Standard T, flow-through run
6	Y-pattern globe 60 deg	32	Standard T, flow-through branch
7	Y-pattern globe 45 deg	33	45 deg T, flow-through run
8	Ball valve	34	45 deg T, flow-through branch
9	Butterfly 2-8 inches	35	Elbow 90 deg, R/D = 1.0
10	Butterfly 10-14 inches	36	Elbow 90 deg, R/D = 1.5
11	Butterfly 14-24 inches	37	Elbow 90 deg, R/D = 2.0
12	Plug, straight-way	<b>WELDED FITTINGS</b>	
13	Plug, 3-way (straight way)	38	Elbow 45 deg, R/D = 1.0
14	Plug, 3-way through branch	39	Elbow 45 deg, R/D = 1.5
15	Foot valve, poppet disk	40	Elbow 45 deg, R/D = 2.0
16	Foot valve, hinged disk	41	Return bend 180, R/D = 1.0
17	Swing check, clearway	42	Return bend 180, R/D = 1.5
18	Swing check, tilting seat	43	Return bend 180, R/D = 2.0
19	Tilt disk 5 deg 2-8 in	44	Tee 100% flow-through run
20	Tilt disk 5 deg 10-14 in	45	Tee 100% flow-out branch
21	Tilt disk 5 deg 16-48 in	46	Tee 100% flow-in branch
22	Tilt disk 15 deg 2-8 in	47	Reducer
23	Tilt disk 15 deg 10-14 in	<b>MISCELLANEOUS</b>	
24	Tilt disk 15 deg 16-48 in	48	Entrance, inward project
25	Lift of stop check, globe	49	Entrance, sharp edged
26	Lift or stop check, angle	50	Entrance, slightly round
		51	Entrance, well rounded
		52	Exit from pipe
		53	Sudden contraction ( $D_{SMALL}/D_{LARGE}$ )
		54	Sudden expansion ( $D_{SMALL}/D_{LARGE}$ )

VALVE DIAGRAMS

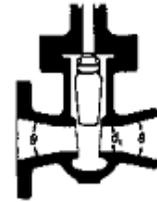
1 GATE VALVE



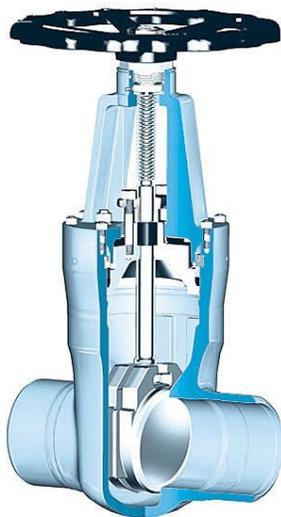
Wedge disc



Double disc

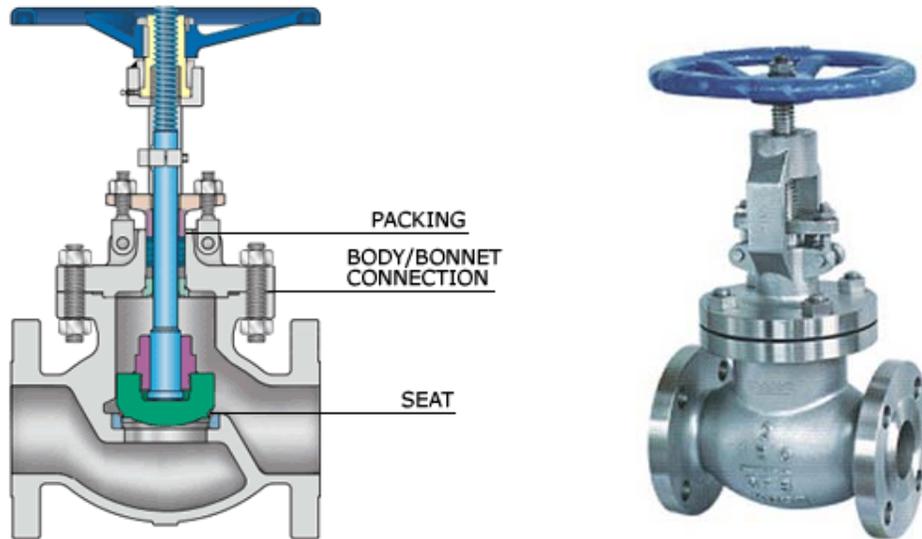


Plug Type

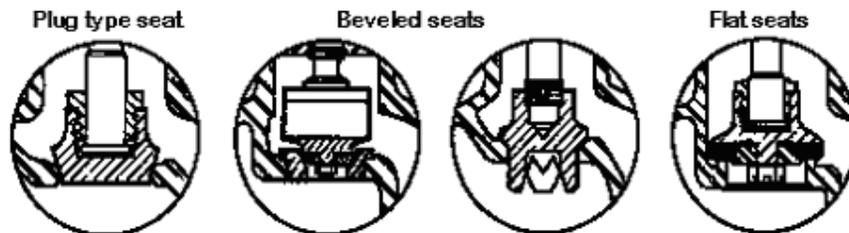


## 2 GLOBE VALVE, BEVEL, PLUG

Globe valve with seat that receives flat, beveled, or plug type disks. All three of these are welded the same.

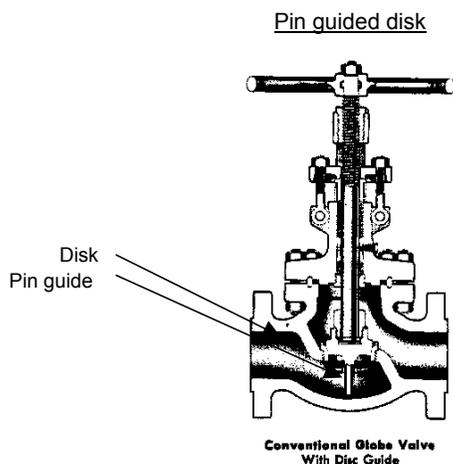


a. Seat arrangement diagrams



### 3 GLOBE VALVE WING/PIN GUIDED DISK

Globe valve with disk guided by pin or wing.



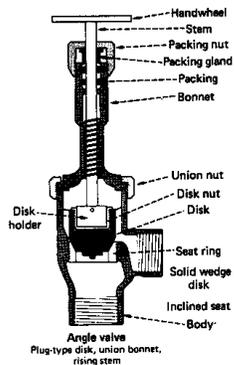
### 4 ANGLE, NO OBSTRUCTION

Angle valve with no obstruction; that is, no disk guides.

Angle valves are basically globe valves having inlet and outlet connections at right angles.

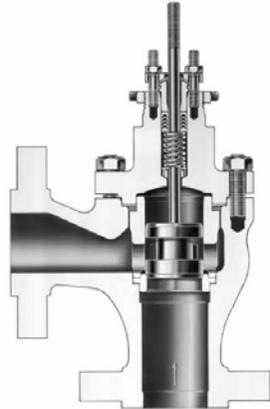
The overall construction elements of an angle valve are basically the same for the stem, disk and seat-rings design as the globe valve.

General diagram of angle valves



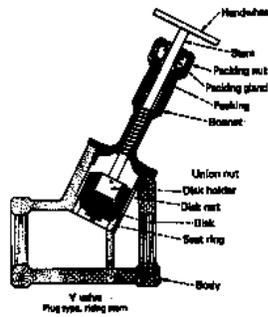
**5 ANGLE, WING/PIN GUIDED DISK**

Angle valve with wing or pin disk guides(not shown)

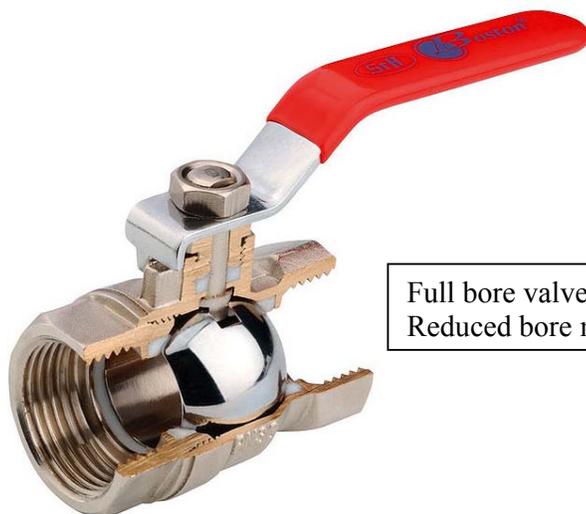


**6 Y-PATTERN GLOBE 60 DEGREES**

**7 Y-PATTERN GLOBE 45 DEGREES**



**8 BALL VALVE**



Full bore valve shown  
Reduced bore more common

The conventional ball valve, as the name implies, has a spherical plug that controls fluid flow. The two basic types are the rotary motion ball valve and the lift or ball-in-cage type. The ball valve provides not only tight shutoff but also good flow characteristics approaching those of an equal percentage valve. For this reason, it is often applied as a combination throttling and shutoff valve. Due to low friction forces, ball valves have been known to open gradually after prolonged exposure to vibration.

The most common body form is the two-way globe body. The spherical plug has a passage that aligns with body parts in the open position. A quarter turn of the plug closes the valve. To achieve tight shutoff, resilient seat rings are embedded in the body. Ball valves can be provided with three or four-way plugs. In some installations this can represent a considerable savings in fittings, valves and space since one unit can replace two or three flow through valves.

- 9 BUTTERFLY, 2-8 IN
- 10 BUTTERFLY, 10-14 IN
- 11 BUTTERFLY, 14-24 IN

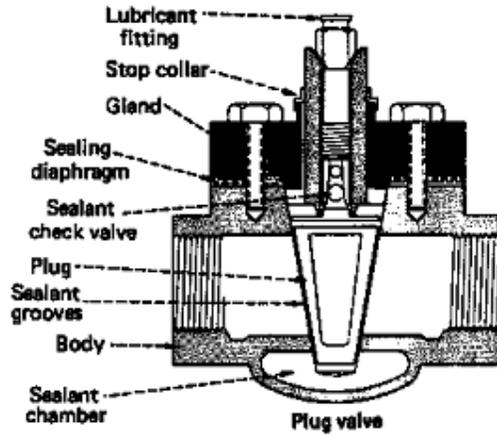


Wafer valve shown  
Direct lever operator shown



Flanged valve shown  
Geared operator

- 12 PLUG, STRAIGHT WAY
- 13 PLUG, 3-WAY (STRAIGHT RUN)
- 14 PLUG, 3-WAY (THROUGH BRANCH)



Straight Way



Straight Run

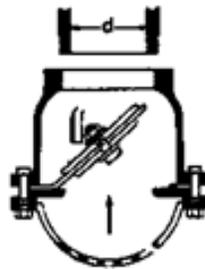
Through Branch



- 15 FOOT VALVE, POPPET DISK
- 16 FOOT VALVE, HINGED DISK



Poppet Disc

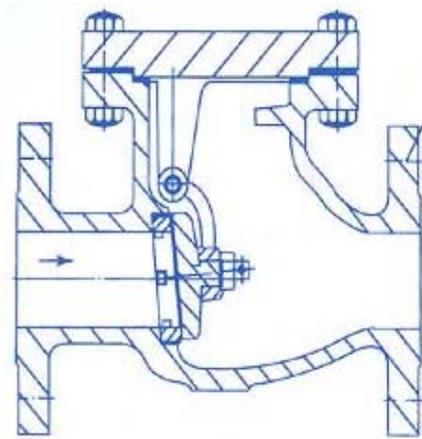


Hinged Disc



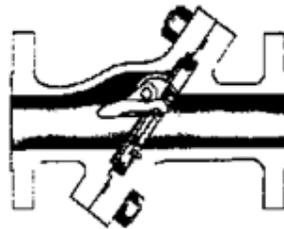
- 17 SWING CHECK, CLEARWAY
- 18 SWING CHECK, TILTING SEAT

Also known as NON-RETURN VALVE

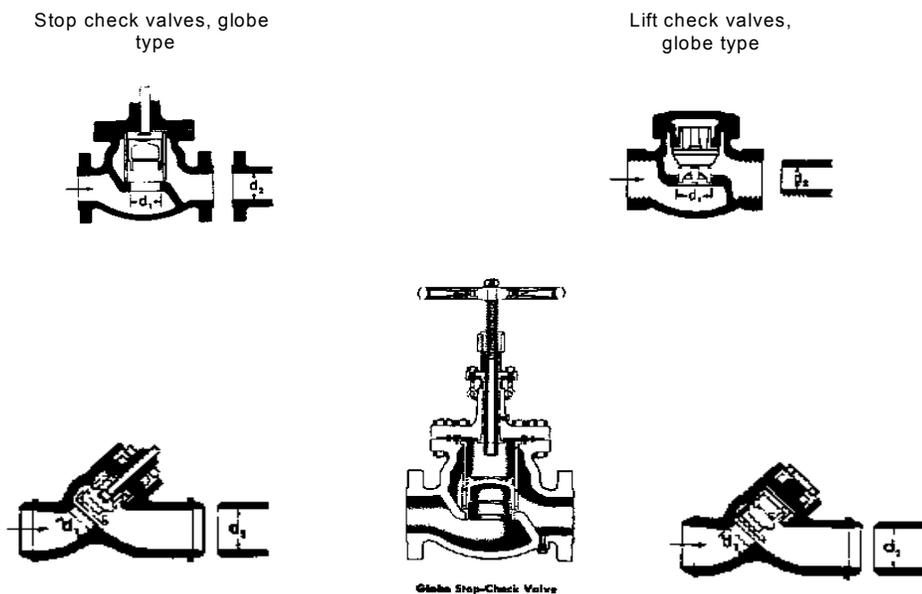


- 19 TILT DISK, 5 DEGREES 2-8 IN
- 20 TILT DISK, 5-DEGREES 10-14 IN
- 21 TILT DISK, 5 DEGREES 16-48 IN
- 22 TILT DISK, 15 DEGREES 2-8 IN
- 23 TILT DISK, 15 DEGREES 10-14 IN
- 24 TILT DISK, 15 DEGREES 16-48 IN

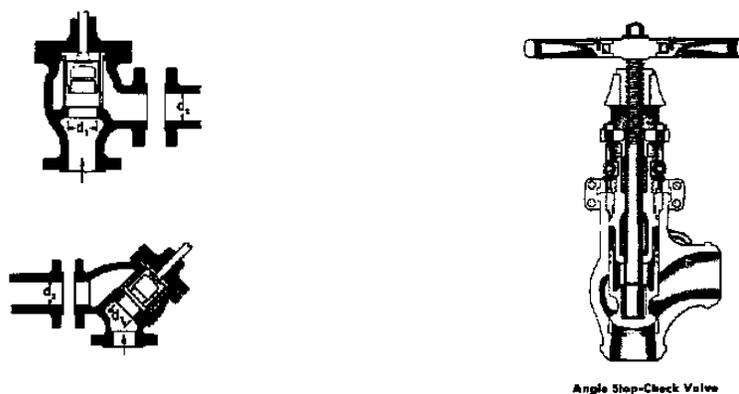
Also known as NON-RETURN VALVE



## 25 LIFT OR STOP CHECK VALVE, GLOBE



## 26 LIFT OR STOP CHECK VALVES, ANGLE



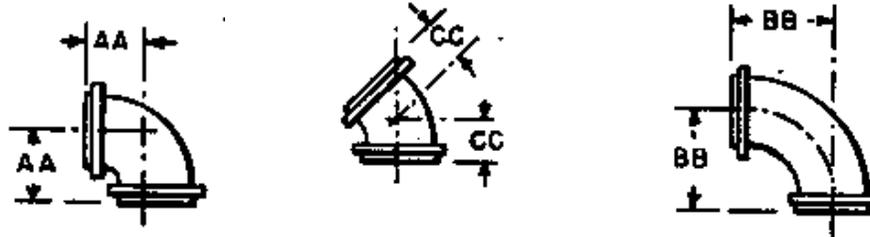
Stop-check valves are usually produced in globe or angle styles, with a stem that is used either to tightly seat or to restrict the lift of the disk. The stem is not attached to the disk, and cannot be used to open the valve. The stop-check valve is used on steam lines from boilers more than in any other application.

Foot valves are specialty check valves used at the bottom of a suction line on a horizontal pump. They maintain the prime on such pumps and usually incorporate strainers to keep foreign matter from entering.

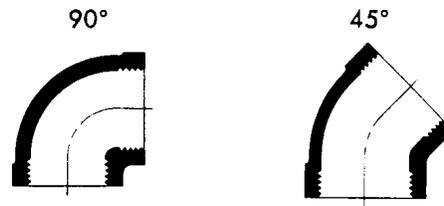
Lift check valves, usually of the globe body form, depend on gravity for operation. They are designed for either horizontal or vertical installation, but the two designs are not interchangeable. Under forward flow, a piston is lifted off the seat by fluid while being retained in the valve by guides. On reverse flow, the piston is forced against the seat to block further flow. Some lift check valves utilize spring loading to assure positive seating. In any case, lift check valves require relatively high pressure drops. They are used on higher pressure services and in smaller pipe sizes than swing checks.

**FLANGED FITTINGS**

- 27 STANDARD ELBOW, 90 DEGREES
- 28 STANDARD ELBOW, 45 DEGREES
- 29 STANDARD ELBOW, 90 LONG R

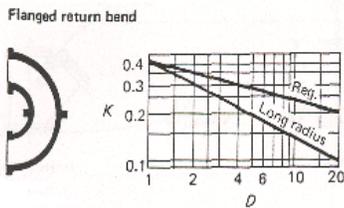


**STANDARD ELBOWS**

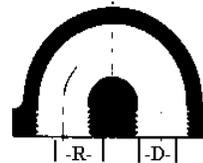


Standard elbows and fittings are built to standard dimensions which are given in mechanical standards like ANSI B16.5, ANSI B16.1, etc. The dimensions change with pressure rating.

- 30 RETURN BEND, 180 DEGREES, CLOSE (FLANGED)

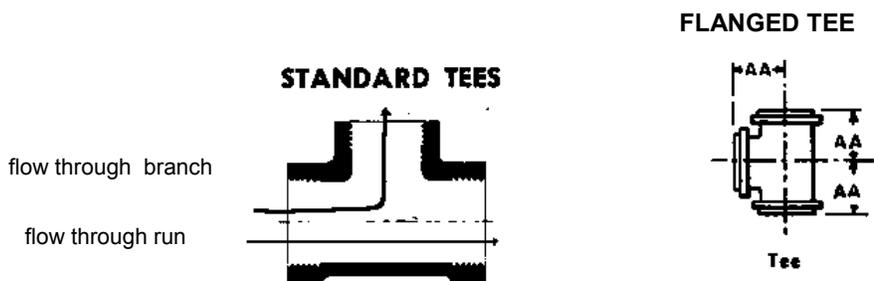


**CLOSE PATTERN RETURN BENDS**



A close return bend means that  $R/D = 1$ .

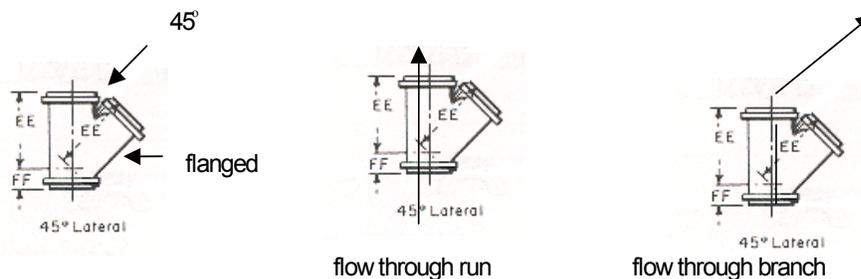
- 31 STANDARD T, FLOW-THROUGH RUN (FLANGED)
- 32 STANDARD T, FLOW-THROUGH BRANCH (FLANGED)



Standard T's are built according to dimensions given in mechanical standards like DIN, ANSI B16.1, ANSI B16.5, etc. The dimensions change with pressure rating.

Nominal pipe size	ANSI B16.5, Class 150 ANSI B16.1, Class 125						ANSI B16.5, Class 300 ANSI B16.1, Class 250					
	AA	BB	CC	EE	FF	GG	AA	BB	CC	EE	FF	GG
1/8	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....
1/4	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....	.....
1/2	3.50	5.00	1.75	3.75	1.75	4.50	4.00	3.00	2.25	6.50	2.00	4.50
3/4	3.75	5.50	2.00	4.25	1.75	4.50	4.25	3.50	2.50	7.25	2.25	4.50
1	4.00	6.00	2.25	7.00	2.00	4.50	4.50	6.00	2.75	8.50	2.50	4.50
1 1/2	4.50	6.50	2.50	5.00	2.50	5.00	5.00	6.50	3.00	9.00	2.50	5.00
2	5.00	7.00	3.00	9.50	2.50	5.50	5.50	7.00	3.50	10.50	2.50	5.50
2 1/2	5.50	7.75	3.00	10.00	3.00	6.00	6.00	7.75	3.50	11.00	3.00	6.00
3	6.00	8.50	3.50	11.50	3.00	6.50	6.50	8.50	4.00	12.50	3.00	6.50
3 1/2	6.50	9.00	4.00	12.00	3.00	7.00	7.00	9.00	4.50	13.50	3.00	7.00
4	7.50	10.25	4.50	13.50	3.50	8.00	8.00	10.25	5.00	15.00	3.50	8.00
5	8.00	11.50	5.00	14.50	3.50	9.00	8.50	11.50	5.50	17.50	4.00	9.00
6	9.00	14.00	5.50	17.50	4.50	11.00	10.00	14.00	6.00	20.50	5.00	11.00
8	11.00	16.50	6.50	20.50	5.00	12.00	11.50	16.50	7.00	24.00	5.50	12.00
10	12.00	19.00	7.50	24.50	5.50	14.00	13.00	19.00	8.00	27.50	6.00	14.00
14	14.00	21.50	7.50	27.00	6.00	16.00	15.00	21.50	8.50	31.00	6.50	16.00
16	15.00	24.00	8.00	30.00	6.50	18.00	16.50	24.00	9.50	34.50	7.50	18.00
18	16.50	26.50	8.50	32.00	7.00	19.00	18.00	26.50	10.00	37.50	8.00	19.00
20	18.00	29.00	9.50	35.00	5.00	20.00	19.50	29.00	10.50	40.50	8.50	20.00
24	22.00	34.00	11.00	40.50	9.00	24.00	22.50	34.00	12.00	47.50	10.00	24.00

- 33 45 DEG. T, FLOW THROUGH RUN (FLANGED)
- 34 45 DEG. T, FLOW THROUGH BRANCH (FLANGED)

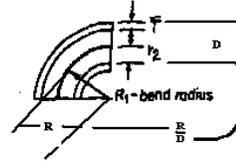


- 35. ELBOW 90 DEGREES, R/D = 1.0 (FOR WELDED, FLANGED, OR SCREWED)
- 36. ELBOW 90 DEGREES, R/D = 1.5 (FOR WELDED, FLANGED, OR SCREWED)
- 37. ELBOW 90 DEGREES, R/D = 2.0 (FOR WELDED, FLANGED, OR SCREWED)

**90° PIPE BENDS AND BUTT-WELDING 90° ELBOWS**



r/d	K	r/d	K
1	20 ft	8	24 ft
1.5	14 ft	10	30 ft
2	12 ft	12	36 ft
3	12 ft	14	38 ft
4	14 ft	16	48 ft
6	17 ft	20	50 ft



**WELDED FITTINGS**

- 38 ELBOW 45 DEGREES, R/D = 1.0 (WELDED)
- 39 ELBOW 45 DEGREES, R/D = 1.5 (WELDED)
- 40 ELBOW 45 DEGREES, R/D = 2.0 (WELDED)



R/D as defined above

- 41 RETURN BEND 180, R/D = 1.0 (WELDED)
- 42 RETURN BEND 180, R/D = 1.5 (WELDED)
- 43 RETURN BEND 180, R/D = 2.0 (WELDED)

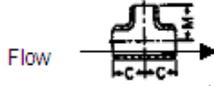


R/D same as for 90° elbow

- 44 TEE 100% FLOW THROUGH RUN (WELDED)
- 45 TEE 100% FLOW OUT BRANCH (WELDED)
- 46 TEE 100% FLOW IN BRANCH (WELDED)



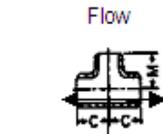
Tee



flowthrough run



flowthrough branch



flow in branch

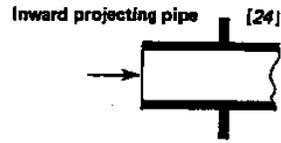
- 47. REDUCER CONCENTRIC OR ECCENTRIC (WELDED)



\

**MISCELLANEOUS**

**48 ENTRANCE, INWARD PROJECTING**



**49 ENTRANCE, SHARP EDGED**

Sharp edged inlet (24)

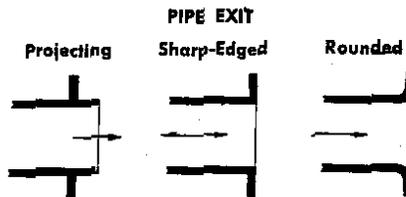


**50 ENTRANCE, SLIGHTLY ROUNDED**  
**51 ENTRANCE, WELL ROUNDED**

$r/d$	$K$
0.00*	0.5
0.02	0.28
0.04	0.24
0.06	0.15
0.10	0.09
0.15 & up	0.04

\*Sharp-edged

**52 EXIT FROM PIPE**



No distinction between types made in CHEMCAD.

<b>REFERENCES</b>
-------------------

1. Deutsch, David J. and Chemical Engineering; *Process Piping Systems*; McGraw-Hill Publications Co.; New York, N. Y., 1980.
2. Dr. Brill, James P., Dr. Beggs, H. Dale; *Two-Phase Flow in Pipes*; Fourth Printing, 1978.
3. Darby, Ron; *Chemical Engineering Fluid Mechanics*; Marcel-Dekker, Inc.; New York, New York; ISBN No. 0-8247-0444-4.
4. Kent , George R.; "*Preliminary Pipeline Sizing*"; Chemical Engineering; September 25, 1978.
5. Crane Co; *Flow of Fluids Through Valves, Fittings, and Pipes*; Technical Paper No. 410; Twenty Fifth Printing, 1991.