pipelinestudio®

GAS NETWORK TRAINING

MODULE : TUNING

energysolutions®

PipelineStudio Training: Module Tuning I
The Module I
The User I
The Training I
Case Study 1 (Simple Model) 1
Case Study 1A (Thermal Model) 11
Case Study 1B (Pipe Leg Tuning) 13
Case Study 1C (Data Sensitivity) 18

PipelineStudio Training: Module Tuning

The Module

The module contains a series of cases; starting with a simple case that will be used as starting point to generate additional cases.

The cases will illustrate the sensitivity of the input values compares to the output values, the Gas equations available upon selection and the different equations of state.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1 (Simple Model)

Purpose

The purpose of this case study is to model a simple A-B configuration of a pipeline in ***pipelinestudio®***, validate the model, run it for steady state and analyze the results. The model consists of a single pipe, supply and delivery. A detailed explanation about modeling of each element is presented in this case study. This case study becomes the basis when modeling the elements in the rest of the case studies.

Important Elements

Starting Point

A case model will be constructed using the Graphical Configurator. In actual practice, ***pipelinestudio***, models are usually started by copying another similar model.

Default and Assumed Values

The amount of engineering data needed to simulate even a Minimal model is considerable. In this case study, for simplicity sake many program defaults will be used.

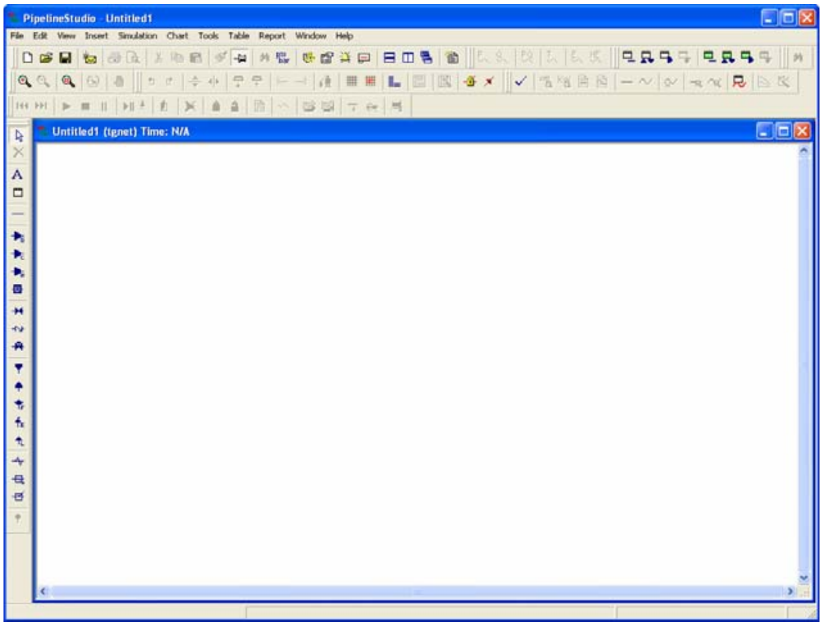
INTRODUCTION

This model represents a Minimal pipeline system consisting of one input (supply) and one output (delivery) connected by a 50-mile pipe segment. The pipeline flow and delivery pressure are the only control data required.

Case Study 1

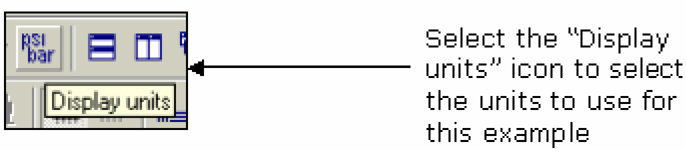
Open the program by double clicking on the ***pipeline****studio* icon on the desktop
Select TGNET configuration
The network view appears as shown below

Network View



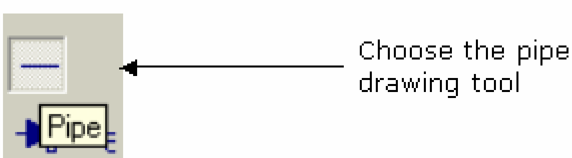
Choosing Units

Choose the units that you want to work on with from Simulation->Units or by clicking the display units icon in the tool bar

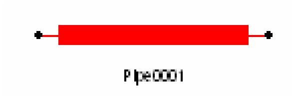


Drawing a pipe

Step1 (Selecting the pipe)

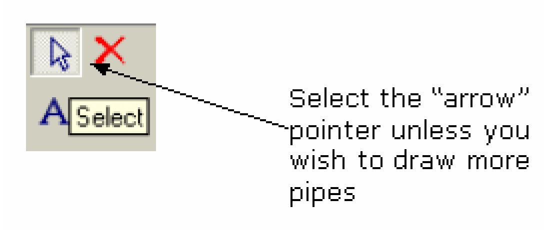


Step2 (Drawing the pipe)



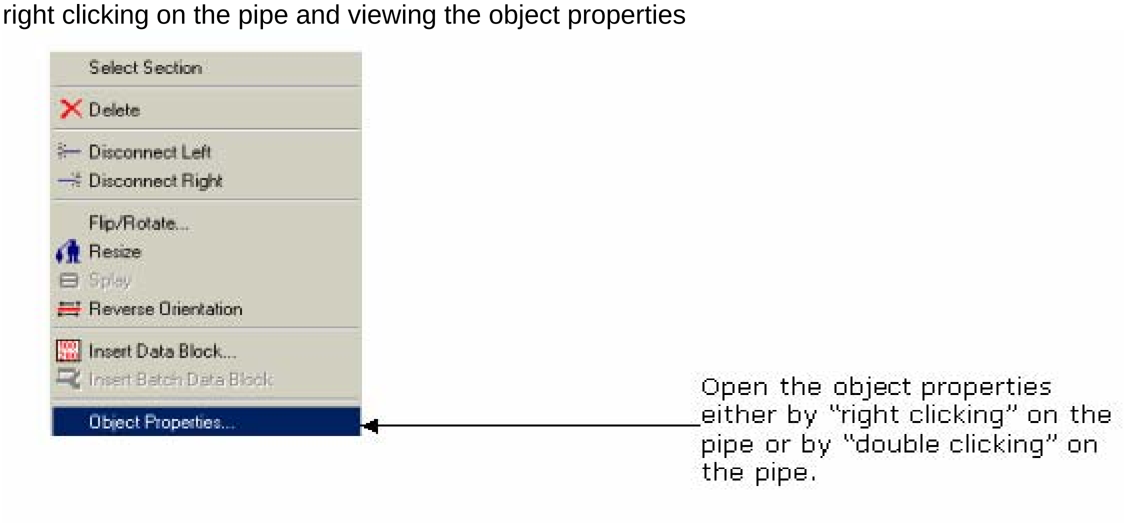
Draw a pipe in the network view

Step3 (De-selecting the pipe)

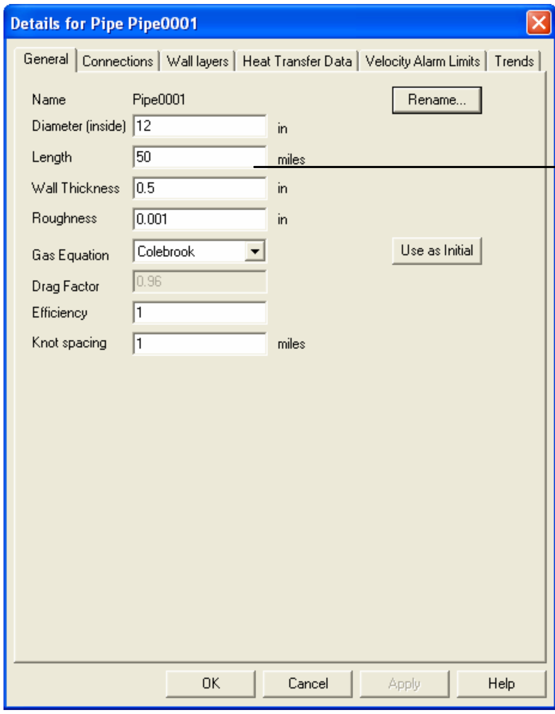


Step4 (Accessing the properties of the pipe)

Change the properties for the pipe either by double clicking on the pipe or by right clicking on the pipe and viewing the object properties



Step5 (Editing the pipe properties)



Enter 50 miles here

External regulators

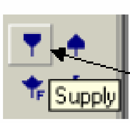
An external regulator is a device, which models flow into or out of a pipeline network at a node. All flows into and out of the network must be modeled with an external regulator. In ***pipelinestudio***, you can include external regulators in a pipeline network that represent supplies, deliveries, and leaks. An external regulator is always connected to just one node. An external regulator is connected to other items in the configuration by a single node.

Adding a supply

Supply points are locations where a source of fluid enters a pipeline network configuration. A supply external regulator allows you to specify the fluid properties and temperature of the flow entering the system.

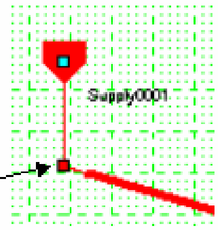
At least one constraint must be defined for a supply external regulator. The recommended constraints are either maximum inlet pressure or maximum inlet flow rate.

Step1 (Selecting the supply tool)



Select the supply tool

Step2 (Adding the supply at the upstream of the pipe)

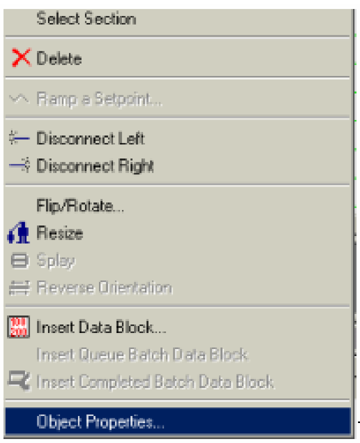


Draw a supply tool attached to upstream node of the pipe

Step3 (De-select the supply)

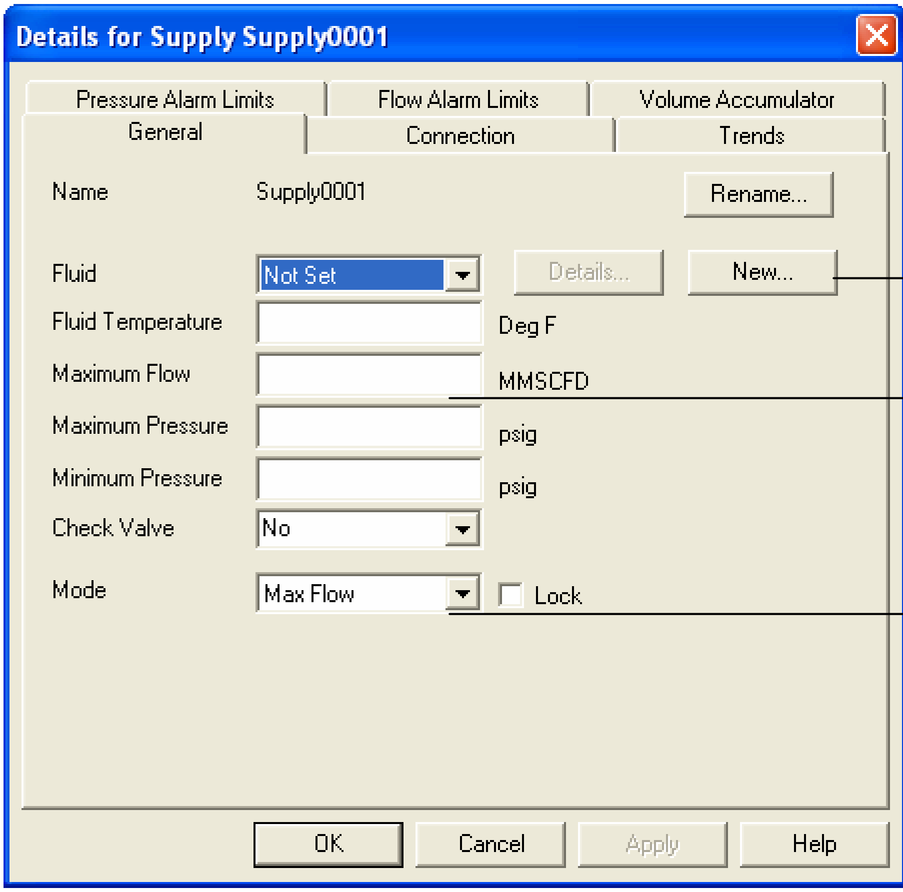
Select the "arrow" pointer unless you wish to draw more supplies

Step4 (Accessing the supply properties)



Select object propertied either by a "right click" on the supply tool or by a "double click" on the supply tool

Step5 (Editing the supply properties)



Tabular fluid to be created from here SG = 0.6, HV=1000 btu/cf CO2= 0.0%

Enter a max flow here = 100 mmscfd

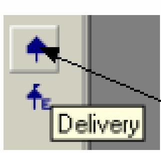
Change the mode here

Adding a delivery

Delivery points are locations where fluids leave a pipeline network, that is, the fluids are delivered to points outside the simulated system.

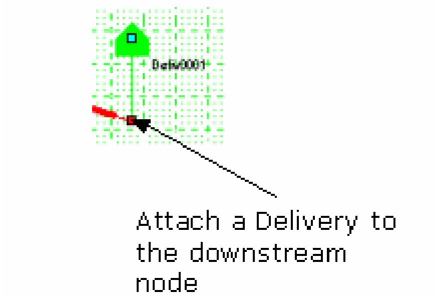
At least one constraint must be specified for a delivery external regulator. The recommended constraints are either minimum delivery pressure or maximum delivery flow rate.

Step1 (Selecting the delivery tool)

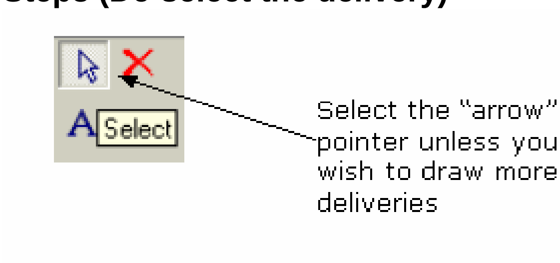


Select the "Delivery" Drawing tool

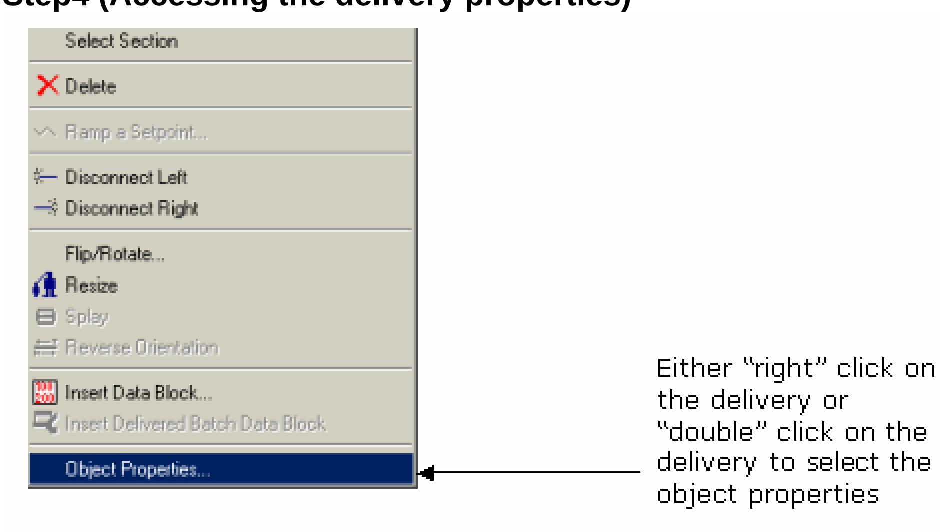
Step2 (Attaching the delivery at the downstream of the pipe)



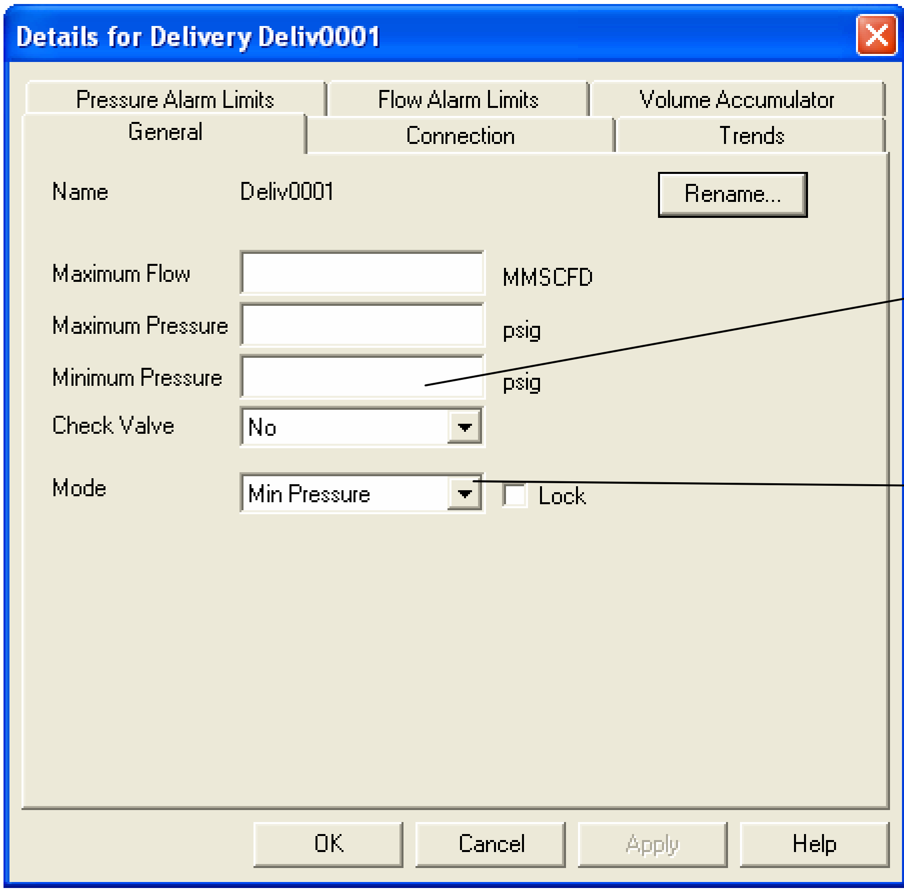
Step3 (De-select the delivery)



Step4 (Accessing the delivery properties)



Step5 (Editing the delivery properties)

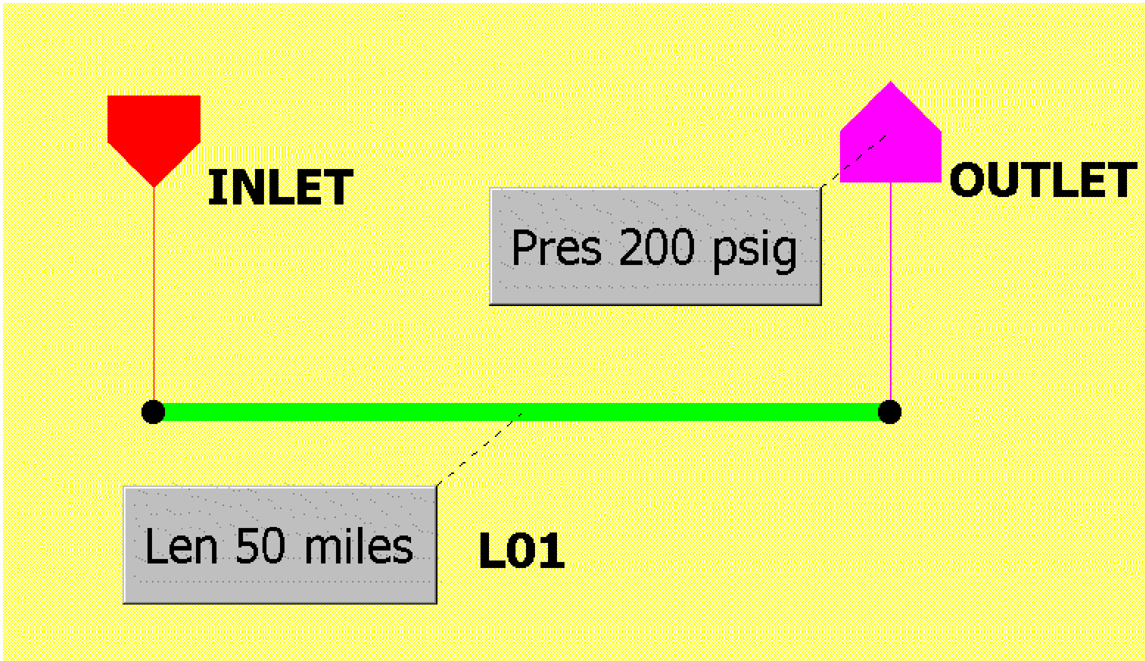


Enter a Min. Delivery pressure of 200 psig

Change the mode to Min pressure here

Validating the network (To check for errors and warnings)

Running the network for Steady state



Case Study 1

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions.

1) Based on the flow requested, what is the required inlet pressure for this pipeline?

2) What default pipe inside diameter did the program use?

3) What gas equation the program has used?

4) What are the head and tail temperatures of the pipeline

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions.

1) Based on the flow requested, what is the required inlet pressure for this pipeline?

943.942 psig

2) What default pipe inside diameter did the program use?

12.00 INCHES

3) What gas equation the program has used?

Colebrook

4) What are the head and tail temperatures of the pipeline

Head Temperature = 60F, Tail Temperature = 60F

Case Study 1A (Thermal Model)

Purpose

In this case effects of temperature on flow are studied. This case is the same as case1 but with temperature tracking "On". The simulator will perform heat balance calculation at every knot only when the temperature tracking is 'on'. If the temperature tracking is 'off', then the simulator will assume a constant system wide temperature. By default, the system wide temperature value is 60 F. User can override this value from **Simulation**->**Options**->**Fluid**->System wide temperature menu.

Instructions

1. Save case1 as case1A
2. Turn "On" temperature tracking
3. This can be done from **Simulations**->**Options**->**General**->Temperature On
4. Everything else will remain same
5. Run the case for steady state
6. Start with Colebrook gas equation

1) Based on the flow requested, what is the required inlet pressure for this pipeline?

2) What fluid temperature and ambient temperatures has the program used?

3) What are the head and tail temperatures of this pipeline

4) What are the head and tail temperatures of the pipeline for a fluid temperature of 50 F and an ambient temperature of 70 F and what is the inlet pressure

5) Turn temperature tracking off, set the system wide temperature to 70 F and determine what is the inlet pressure. Use Colebrook gas equation

Case Study 1A (continued)

1) Based on the flow requested, what is the required inlet pressure for this pipeline?

924.817PSIG

2) What fluid temperature and ambient temperatures has the program used?

Fluid temp= 60 F; Ambient Temp= 50 F

3) What are the head and tail temperatures of this pipeline

Head Temp=60 F & Tail Temp=34.30 F

4) What are the head and tail temperatures of the pipeline for a fluid temperature of 50 F and an ambient temperature of 70 F and what is the inlet pressure? Use Colebrook gas equation

Head Temp= 50 F & Tail Temp= 47.63 F, inlet pressure=926.83 psig

5) Turn temperature tracking off, set the system wide temperature to70F and determine what is the inlet pressure. Use Colebrook gas equation

956.563psig

Case Study 1B (Pipe Leg Tuning)

Purpose

This case study examines some of the procedures used to "tune" a pipeline model to better match calculated results with actual field data.

Important Elements

Starting Point

This case study and subsequent models are created by copying input from pre-existing model files.

Adjustment to Roughness

Pipe roughness is one of the primary input variables used to adjust calculated pressure drop results. Pipeline roughness can be difficult to measure accurately for an entire pipeline due to the inherent differences experienced down the length of a pipeline. Roughness in one segment can be quite different than another. As a result, many pipelines choose an average figure for pipeline roughness that accounts for these factors over an acceptable distance.

One way that pipeline roughness can be determined is by tuning the roughness such that calculated pressure drop closely matches actual observed pressure drop.

INTRODUCTION

The input file for this case study is essentially the same as for CASE 1, with the following exceptions.

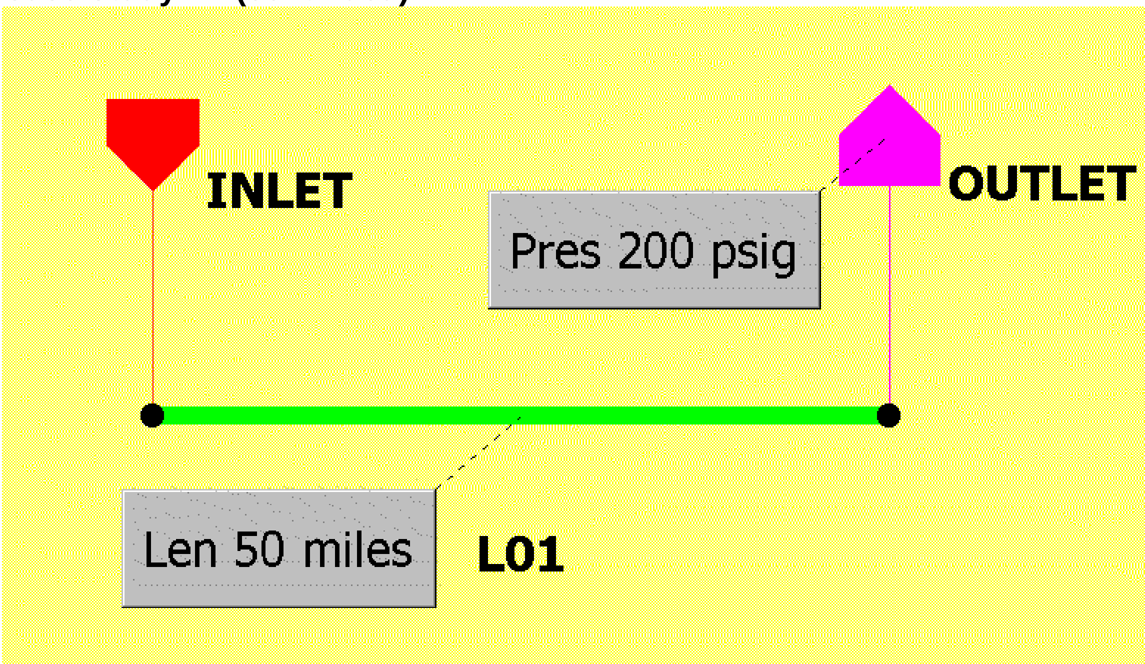
PRESSURE - PRESSURE CONTROLS

The leading flow control in CASE 1 has been removed and replaced by a maximum pressure setpoint of 1000 psig. In this configuration the resulting flow is calculated from the pressure setpoint and the pipe specifications.

ADDITIONAL PIPE SPECIFICATIONS

To permit adjustments of the pipe leg, additional pipe parameters are employed in this case study

Case Study 1B (continued)



Case 1B

Additional Information:
Pipe = L01
Length = 50 miles
ID = 35.5 inches
WT = 0.25 inches
Start with the default roughness value
Use Colebrook gas equation

INSTRUCTIONS

1) Save the CASE1 as CASE1B. Rename the Title of the case "CASE 1B, TUNING A MODEL".
2) Use the same simple fluid
3) Edit the network file to incorporate the above new information plus the following: MODE at INLET - Max Pressure
4) Validate network and run steady state simulation
5) View steady state output to obtain your new flow value
6) Your desired flow rate is 2000 mmscfd
7) Using trial and error methods, "tune" the roughness value to generate the required flow

Case Value Input Table:

Trial Number	Roughness Entered (inches)	Flow Obtained (mmscfd)
1		
2		
3		
4		
5		
6		
7		
8		

Case Study 1B (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions.

1) Record the flow rate to which you tuned your pipe roughness

2) What roughness value did you obtain?

Case Study 1B (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions.

1) Record the flow rate to which you tuned your pipe roughness

2000 MMSCFD

2) What roughness value did you obtain?

0.000171 inches

Case 1B Study (continued)

INSTRUCTOR'S SOLUTION

Given a flow rate of 2000 MMSCFD the model input was adjusted in a series of runs, each of which improves tuning.

Trial Number	Roughness Entered (inches)	Flow Obtained (mmscfd)
1	0.010000	1433.09
2	0.000010	2177.34
3	0.000100	2055.07
4	0.000150	2014.56
5	0.000200	1982.68
6	0.000175	1997.78
7	0.000169	2001.64
8	0.000171	2000.3

Case Study 1C (Data Sensitivity)

Purpose

When constructing a model, the sensitivity of some of the data items is far more sensitive than others. This case study experiments with the sensitivity of the input data for a pipe leg.

Important Elements

Importance of Inside Diameter

Note carefully the significance of pipe inside diameter in this case study.

Adjustment and Input Variables

Although roughness is the major variable used to tune a hydraulic model, All of the input variables must be determined to ensure a complete model. Due to the insensitivity of some variables it is acceptable to simply use a default value. For others much care should be given in obtaining a value as close as possible to the actual facility being modeled.

INTRODUCTION

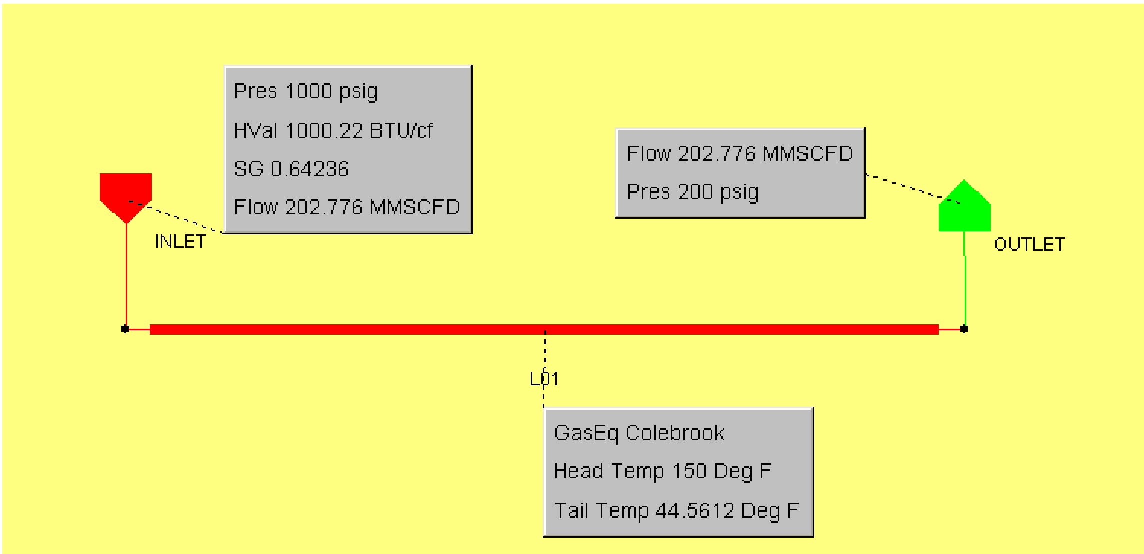
This is essentially the same as CASE 1, except that all of the primary pipe parameters are entered.

ADDITIONAL PIPE SPECIFICATIONS

To permit adjustments of the pipe leg, additional pipe keywords are employed in this case study. A compositional fluid is created in this case.

Each of the input variables' base value will be doubled and steady state values computed against the change in value. A table is provided to record the results.

Case Study 1C (continued)



Case1C

INSTRUCTIONS

1. Save CASE1 save as CASE1C. Rename the Title of the case "CASE 1C, DATA SENSITIVITY".
2. Use the same fluid and input the baseline conditions.
3. Run a Steady Simulation and obtain the flow value at the supply. Enter this value as the BASELINE flow in the table on the next page.
4. One-by-one change the pipe leg inputs to the values shown in the second column Record the resulting flow into the table on the next page. **Be sure to return the input value to its original state before changing the next value!**
5. Determine which data value is most sensitive to input error.
6. Gas composition: C1=85.6, C2=9, C3=2, NC4=1, C02=0.2, N2=2.2
7. Turn "On" temperature tracking

Case Study 1C (continued)

CASE VALUE INPUT TABLE (Physical properties)

Input Variable Changed	Baseline Input	Altered Input	% change	Flow (mmscfd)	% change
Length (miles)	50.0	25			
Inside Diameter (inches)	15.50	7.75			
Roughness (inches)	0.00015	0.000075			
Heat Transfer Coefficient"U" (BTU/hr.ft^2.^{o}F)	1.0	0.5			
Gas Temperature (^{o}F)	150	75			
Pipe Efficiency (%)	90	45			
Ambient Temperature	50	25			
Wall Thickness	0.5	0.25			

Baseline Flow (mmscfd)

CASE VALUE INPUT TABLE (Gas Equation)

Gas Equation	Baseline Input	Altered Input	Flow (mmscfd)	% change
	Colebrook	AGA		
		GSO		
		PAN(A)		
		PAN(B)		
		Spitz glass		
		Weymouth		

Baseline Flow (mmscfd)

CASE VALUE INPUT TABLE (Equation of state)

Equation of state	Baseline Input	Altered Input	Flow (mmscfd)	% change
	Sarem	BWRS		
		Peng-Robinson		

Baseline Flow (mmscfd)

Case 1C Study (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions.

1) Which data value is the most sensitive to input accuracy?

2) Which data value is the least sensitive to input accuracy?

3) Does the EOS affect the flow to a large extent

4) Which gas equation has the highest effect on flow

Case Study 1C (continued)

INSTRUCTOR'S SOLUTION

Altering the input values as per the instructions, the values below were obtained:

Input Variable Changed	Baseline Input	Altered Input	% change	Flow (mmscfd)	% change
Length (miles)	50.0	25	50	277.95	36.18
Inside Diameter (inches)	15.50	7.75	50	33.93	-83.04
Roughness (inches)	0.00015	0.000075	50	207.31	2.78
Heat Transfer Coefficient"U" (BTU/hr.ft^2.^{o}F)	1.0	0.5	50	197.352	-2.15
Gas Temperature (^{o}F)	150	75	50	208.36	3.30
Pipe Efficiency (%)	90	45	50	99.06	-50.88
Ambient Temperature	50	25	50	208.64	3.44
Wall Thickness	0.5	0.25	50	207.12	2.68

Baseline Flow (mmscfd)
201.589

Gas Equation	Baseline Input	Altered Input	Flow (mmscfd)	% change
	Colebrook	AGA	205.3	1.78
		GSO	208.76	3.50
		PAN (A)	218.95	8.55
		PAN (B)	216.17	7.18
		Spitz glass	137.23	31.96
		Weymouth	166.98	17.21

Baseline Flow (mmscfd)
201.699

Equation of state	Baseline Input	Altered Input	Flow (mmscfd)	% change
	BWRS	SAREM	202.9	0.59
		Peng-Robinson	203.99	1.13
		Peng-78	203.99	1.13
		Ideal	188.86	-6.35

Baseline Flow (mmscfd)
201.699

Case 1C Study (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions.

1) Which data value is the most sensitive to input accuracy?

Pipe Diameter

2) Which data value is the least sensitive to input accuracy?

Heat transfer coefficient

3) Does the EOS affect the flow to a large extent

The effect is minimal

4) Which gas equation has the highest effect on flow

Spitz glass

pipelinestudio®

GAS NETWORK TRAINING

MODULE : CAPACITY

energysolutions®

PipelineStudio Training: Module Capacity ..... **I**
The Module I
The User I
The Training I
Case Study 1 (Steady State Capacity) **1**
Case Study 1A (Steady State Capacity Part II) 7
Case 1B (Steady State Capacity Part III) 13
Case 1C (Steady State Capacity Part IV) 16
Case Study 2 (Transient Capacity) **20**
Case Study 2A 25

PipelineStudio Training: Module Capacity

The Module

The module contains a series of cases; starting with a case that will be used as starting point to generate additional cases.

The cases will illustrate the difference between steady state and transient capacity.

The User

This ***pipeline**studio* Training Class is intended to the users of the product or someone who wants to become more productive using ***pipeline**studio*.

The goal of the ***pipeline**studio* training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

***pipeline**studio* Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

***pipeline**studio* training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1 (Steady State Capacity)

Purpose

The determination of pipeline capacity is one of the primary objectives for using ***pipelinestudio*** Gas (TGNET). **The capacity of a pipeline is the amount of gas that a pipeline is rated to transport** (* The concept of "capacity" can vary unless it is assumed that gas flowing into the system equals that which flows out of the system*).
A steady-state simulation calculates time-invariant pressure, temperature and flow profiles throughout a pipeline network using specified boundary conditions and network element setpoints. In other words, the steady-state run calculates the hydraulic state of a pipeline system operating at equilibrium. A transient simulation models the dynamic response of the pipeline network to changes in one or more system variables, such as source/delivery rates or network element setpoints.
(*If one were to consider the time element, then the condition becomes what is known as "transient" or unsteady state. This case study will examine steady state capacity. Transients will be considered in a later case.*)

The objective of this case study is to gain an appreciation of those elements of pipeline design, which are important for performing pipeline capability evaluation under steady state conditions.

Important Elements

Steady State Behaviour

This case model deals with steady state behavior only. Steady state hydraulics is suitable for determining pipeline capacity under reasonably stable operating conditions. Pipeline capacity is normally assumed to be that determined under steady state conditions unless otherwise noted. If a pipeline exhibits large load variations, then determination of pipeline capacity using steady state simulations may not properly determine the true operating capacity of that pipeline. Under such conditions it might be more prudent to perform transient hydraulic simulations. Whether or not one needs to run transient simulations is determined on a case by case basis.

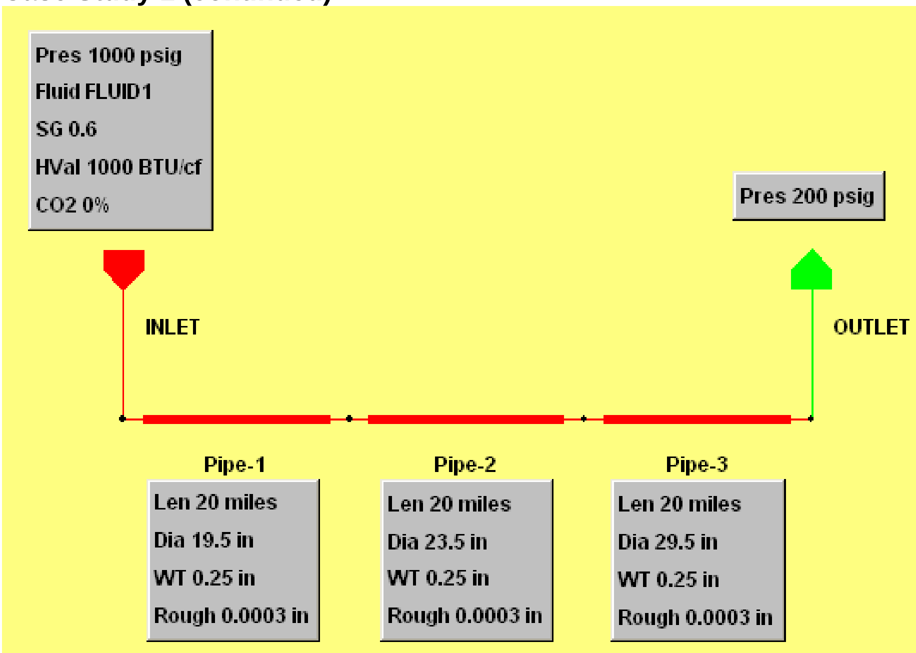
Design Concerns

Each case study in this series deals with pipeline design. In most cases, design concerns can be satisfied using steady state hydraulics.

Introduction

The pipeline in this study is comprised of three pipe segments of different diameter, each 20 miles in length. Expansion of the pipeline's steady state capacity will be made by the addition of a single loop segment. The challenge will be to determine the most effective location for the loop.

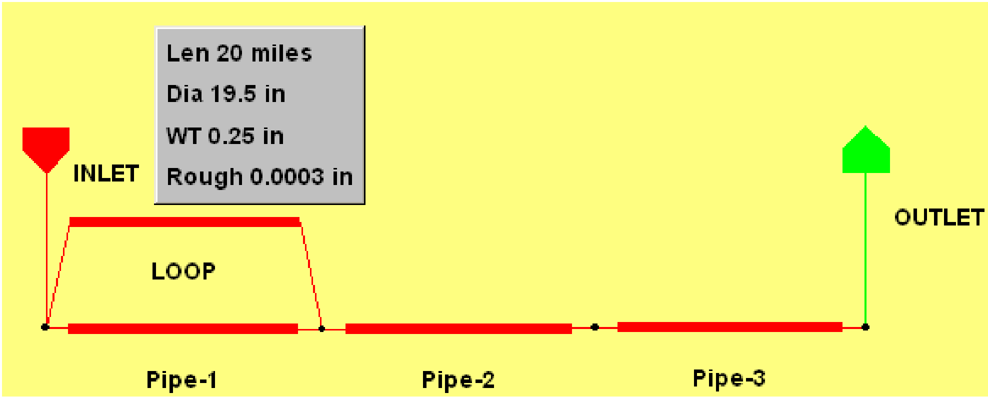
Case Study 1 (continued)



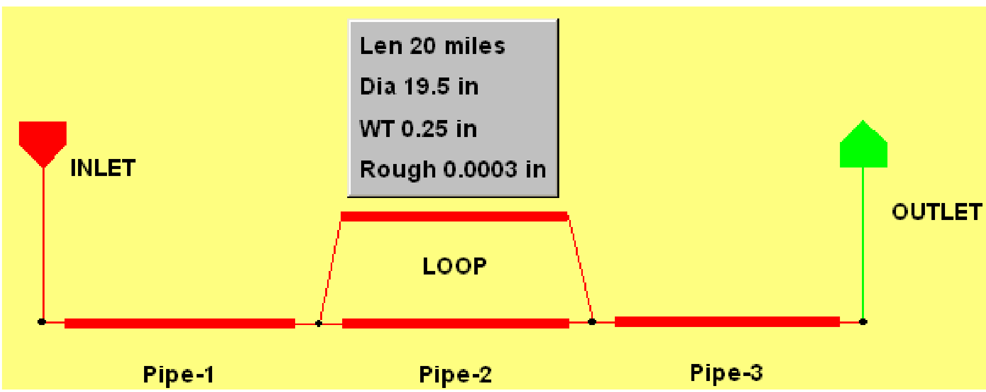
Case 1 (Steady State Capacity)

INSTRUCTIONS

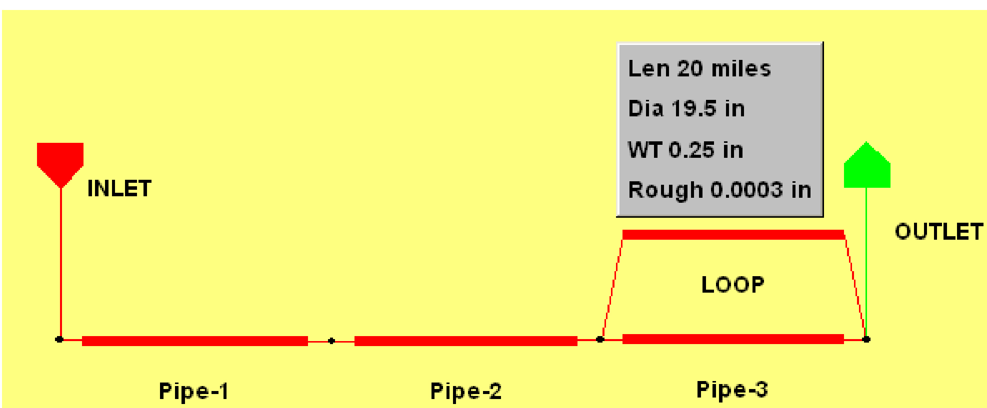
1. Create a new **base** case and call it Case1.
2. Construct a model of the pipeline shown above (turn temperature tracking off).
3. Set the System-Wide Temperature to 150°F
4. Run a Steady Simulation on the base model (without the loop) and record the flow rate.
5. Create a new case by copying Case1 as shown. Call it “Case1_1”.
6. Add the LOOP segment shown below in parallel to the first pipe Pipe-1. Run a Steady Simulation and record the new flow rate.
7. Repeat the steps, for cases CASE1_2 and CASE1_3 attaching the LOOP segment to the pipes Pipe-2 and Pipe-3. Record the respective flows in each instance.



Case1_1 (Steady State Capacity-Loop added to the first segment)



Case1_2 (Steady State Capacity-Loop added to the second segment)



Case 1_3 (Steady State Capacity-Loop added to the third segment)

Case Study 1 (continued)

CASE VALUE INPUT TABLE

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment		
Parallel with Pipe-1		
Parallel with Pipe-2		
Parallel with Pipe-3		

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) Which loop location provides the greatest capacity increase?

2) Why do you think this is?

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

Altering the input values as per the instructions, the values below were obtained.

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment	474.525	
Parallel with Pipe-1	670.605	41.32
Parallel with Pipe-2	517.172	8.99
Parallel with Pipe-3	483.214	1.83

Deleted: 474.196

Deleted: 670.14

Deleted: 516.813

Deleted: 482.879

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) Which loop location provides the greatest capacity increase?

The first segment

2) Why do you think this is?

This segment represents the greatest bottleneck to the system because of its smaller diameter

Case Study 1A (Steady State Capacity Part II)

Purpose

This case study is a modified version of CASE1. In this case all of the pipe segments have the same internal diameter, and the effects of this will be observed.

The objective of this case study is to gain an appreciation of the impact of placing loop segments in different locations on the pipeline

Important Elements

Steady State Behavior

This case model concerns steady state behavior. Since all of the pipes are the same size, no individual pipe forms a hydraulic bottleneck.

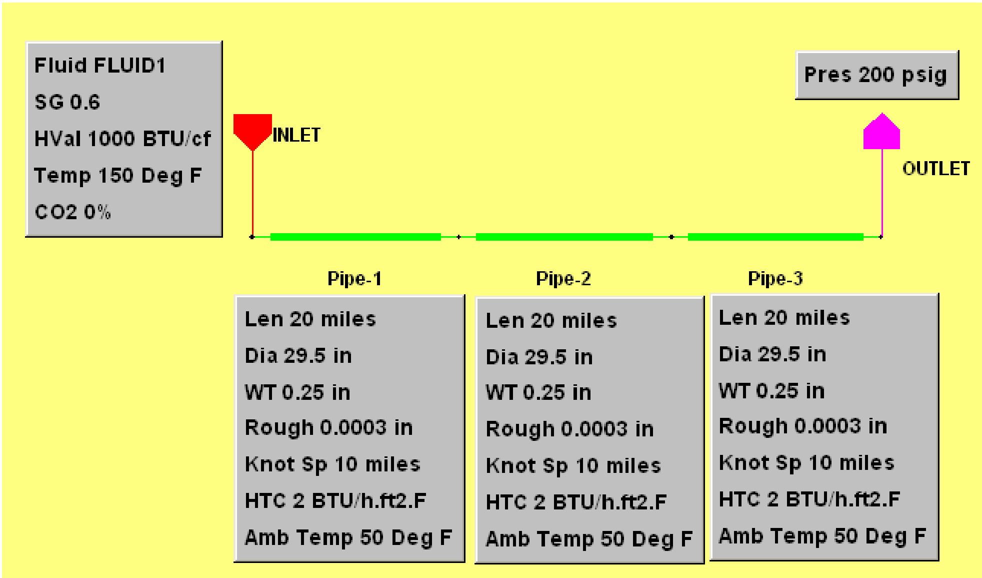
Design Concerns

Each of the case studies in this series is concerned with the design of a pipeline. The implications and limitations of Steady State capacity will be investigated.

Introduction

The pipeline in this study is comprised of three segments of pipe, each of identical diameter. For convenience, each segment will be twenty miles long. Expansion of the pipeline's steady state capacity will be made by the addition of a single loop segment. The challenge will be to determine the most effective location for the loop.

Case 1A (continued)



Case1A

INSTRUCTIONS

1) Create a new case from CASE1 and call it CASE1A (Temperature Tracking On).

2) Update the model to reflect the diagram above

3) Run a steady state simulation on the base model and record the flow rate.

4) Create a new case by copying CASE1A. Call it CASE1A_1

5) Add a LOOP segment (with the same pipeline properties as the other segments shown) in parallel to the first pipe Pipe-1.

6) Run a steady state simulation and record the resulting flow rate.

7) Repeat steps to create cases CASE1A_2 and CASE1A_3 by attaching the LOOP segment to the pipes Pipe-2 and Pipe-3. Record the respective flows in each instance in the table provided.

8) Now try to change the knot length spacing from 10 miles to 0.5 miles.

Case 1A (Steady State Capacity Part II)

Case Value Input Table

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment		
Loop with Leg L01		
Loop with Leg L02		
Loop with Leg L03		

Case Value Input Table (Knot Spacing = 0.5)

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment		
Loop with Leg L01		
Loop with Leg L02		
Loop with Leg L03		

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) Which loop location provides the greatest capacity increase?

2) Why do you think this is?

3) Does reducing the knot spacing change your answer? Why?

4) How does temperature tracking “turned on” affect pressure drop and the resulting flow?

Case Study 1A (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) Which loop location provides the greatest capacity increase?

Marginally at the furthest upstream segment

2) Why do you think this is?

A loop will allow more efficient cooling of the gas (by effectively doubling the pipe's surface area). The cooler the gas is, the more gas can be packed into the pipe, and the resulting flow will be higher. Placing this loop at the upstream end of the system allows more gas to be packed into the whole pipeline. To verify this, run the simulation again with temperature tracking disabled. The change in flow rate should now be negligible.

3) Does reducing the knot spacing change your answer? Why?

Yes, by reducing the knot spacing, the calculation accuracy improves.

4) How does temperature tracking "turned on" affect pressure drop and the resulting flow?

Temperature tracking off assumes isothermal conditions (i.e. 150 ^{o}F gas down the pipeline), this causes higher pressure drop and thus less capacity than when temperature tracking is on (however, more calculations are required)

Case 1A (Steady State Capacity Part II)

INSTRUCTOR'S SOLUTION

Altering the input values as per the instructions, the following values were obtained.

Case Value Input Table

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment	1057.5	
Loop with Leg L01	1247.77	17.99
Loop with Leg L02	1212.4	14.64
Loop with Leg L03	1205.51	14.09

Deleted: 1056.8
Deleted: 1246.9
Deleted: 1211.56
Deleted: 14.65
Deleted: 1205.67

Case Value Input Table (Knot Spacing = 0.5)

Location of Loop Segment	Resulting Flow (mmscfd)	Percent Change
No Loop Segment	1071	
Loop with Leg L01	1273.7	18.93
Loop with Leg L02	1230.95	14.93
Loop with Leg L03	1220.2	13.93

Deleted: 1070.25
Deleted: 1272.81
Deleted: 1230.09
Deleted: 1219.36

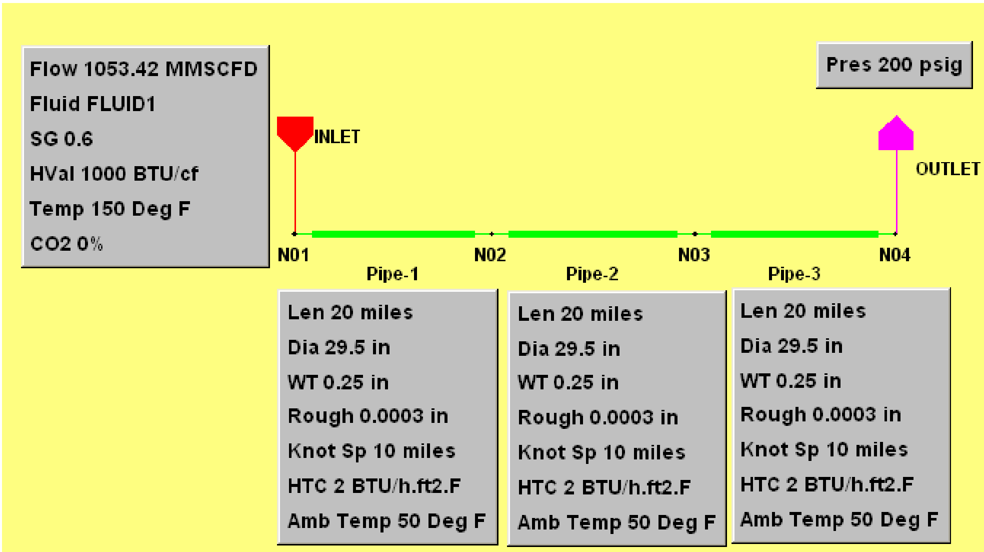
In this case, judging from the steady state results, the precise location of the looped segment is unimportant.

Since no particular pipe segment is more restrictive than another, the increase in flow capacity across that segment represents the same relative importance in all three positions.

The very insignificant flow differences are due primarily to the changes effected by the loop line addition on the overall flow velocity in the pipe legs. A reduction in velocity means lower friction and thus less pressure loss.

Case 1B (Steady State Capacity Part III)

This case study is an extension of case study 1A wherein we enter elevation details at the nodes and change the control mode for the supply



Instructions

1) Save case1A as case1B
2) Name the nodes as shown above
3) Remove the pressure control in the supply and input Maximum Flow = 1053.42 MMSCFD
4) Verify that the temperature tracking is “On” and the fluid temperature = 150 F
5) Verify for each pipe the ambient temperature = 50 F and the HTC = 2 btu/hr.ft2 oF
6) Enter the following elevations for the nodes
7) Use a knot spacing of 10 miles

Formatted: Bullets and Numbering

	Name	Elevation
Initial	**Node**	**0**
Units		ft
0001	N01	415
0002	N02	3000
0003	N03	4000
0004	N04	5000

8) Run the case for steady state

Formatted: Bullets and Numbering

Case Study 1B (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What is the supply pressure for a flow of 1053.42 MMSCFD

2) What is the total inventory in the pipeline in this case

3) Now remove the elevations at the nodes i.e. use a flat elevation and determine the pressure at the supply for a flow of 1053.42 MMSCFD

4) Why do you think less supply pressure is required when there is a flat elevation for the pipeline

Case Study 1B (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) What is the flow at the supply for a pressure of 1000 psig

1060.56psig

Deleted: 1061.25

2) What is the total inventory in the pipeline

78.0104 MMCSF

Deleted: 78.0029

3) Now remove the elevations at the nodes i.e. use a flat elevation and determine the pressure at the supply for a flow of 1053.42 MMSCFD

996.348 psig. The required pressure at the supply is slightly less when the elevation is flat

Deleted: 997.002

5) Why do you think less supply pressure is required when there is a flat elevation for the pipeline

Less pressure (effort) is required to move the amount of flow

Case 1C (Steady State Capacity Part IV)

Purpose

The purpose of this case study is to determine how far an existing system can meet new flow nominations from additional deliveries that become part of the existing system from time to time

Important Elements

Additional deliveries

Initially the case is modelled with a single delivery. However after achieving steady state convergence with this, new deliveries are added and their impact is studied.

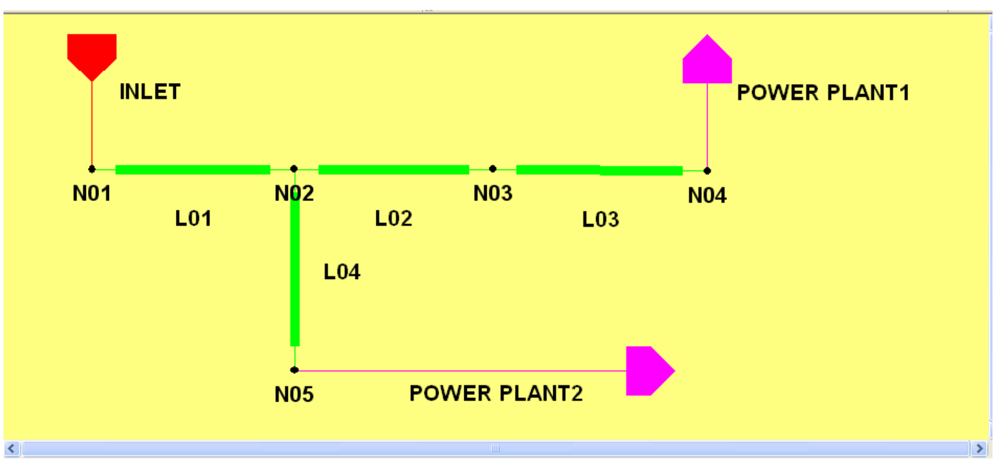
Design Concerns

The implications of adding additional deliveries to the existing system is considered in each case

Introduction

The original system consists of an Inlet (Supply) and a delivery (Power Plant1). Model this and run it for Steady state convergence.

A new power station is being proposed and it requires gas from the 29" line. The tie-in location will be at the N02 node. The new power plant is at a distance of 18 miles and the pipeline to this power plant is of 18" diameter. Make the necessary changes to case 1A and save it as 1C.



Pipeline Information

Name	Length (miles)	Inside Dia (in)	Roughness (in)	Knot spacing (miles)	Wall thickne
Pipe-1	20	29.5	0.0003	10	0.25
Pipe-2	20	29.5	0.0003	10	0.25
Pipe-3	20	29.5	0.0003	10	0.25
Pipe-4	18	18	0.0003	10	1

Formatted Table

Inlet
Max pressure =1000 psig & Gas temperature = 150 F

Power Plant 1
Minimum pressure = 200 psig

Power Plant 2
Minimum pressure = 200 psig

Case Study 1C (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What is the maximum flow at the Power Plant1 and Power Plant2

Deleted: Power Plant 1 = 371. 277 MMSCFD Power Plant 905.366 MMSCFD

2) A new power plant is proposed and it requires gas from the 29" pipeline. The tie-in location for this pipeline will be at node N02. The new power plant is at a distance of 18 miles and the pipeline diameter will be 18". What is the flow at the power plant 2 for a minimum pressure of 200 psig

3) Power plant 2 wants to receive gas at a higher flow rate of 500 MMSCFD. How do we achieve this? A minimum pressure of 200 psig is still required. What is the flow to power plant 1 under these circumstances

4) A new power plant 3 is being proposed after the power plant 1 at a distance of 20 miles. The pipeline to this new power plant 3 will have a diameter of 29.5" at a minimum pressure of 150 psig. Conduct a SS simulation to determine the flow to power plant 3

5) Power plant 1 wants to reduce its daily flow to 500 MMSCFD and power plant 3 raises its daily consumption to 500 MMSCFD. Run the simulation to find out the maximum flow that is possible at power plant 3. Minimum pressures at the power plants remain the same.

Case Study 1C (continued)

INSTRUCTOR'S SOLUTIONS

1) What is the maximum flow at the Power Plant-1 and Power Plant-2 for a minimum pressure of 200 psig

Power Plant-1 = 905.994 MMSCFD Power Plant2= 371.534 MMSCFD

Deleted: 371. 277

Deleted: 905.366

2) Power Plant-2 needs to receive gas at a higher flow rate of 500 MMSCFD. How do we achieve this? A minimum pressure of 200 psig is still required. What is the flow to power plant 1 under these circumstances

A parallel line needs to be added to the system between the nodes N02 to N05. Flow to Power Plant-1 = 846.335 MMSCFD

Deleted: 845.582

3) A new Power Plant-3 is being proposed after the Power Plant-1 at a distance of 20 miles. The pipeline to this new Power Plant-3 will have a diameter of 29.5" and requires at a minimum pressure of 150 psig. Conduct a Steady state simulation to determine the flow to Power Plant- 3.

Flow = 238.522 MMSCFD

Deleted: 238.359

4) Power Plant-1 needs to reduce its daily flow to 500 MMSCFD and Power Plant-3 raises its daily consumption to 500 MMSCFD. Run the simulation to find out the maximum flow that is possible at Power Plant-3. Minimum pressures at the power plants remain the same.

A flow of 338.134 is possible at Power Plant-3.

Deleted: 337.431

Case Study 2 (Transient Capacity)

Purpose

This case study is a modified version of CASE1 (Base case). In this case, the user will get an introduction to the use of transient analysis.

Important Elements

Transient Behavior

This case study deals with pipeline behavior under non-steady state or transient conditions. While steady state assumes that the summation of pipeline input equals output, transient will examine the change in pipeline conditions as a function of time. In real life, we know that the instantaneous input does not equal the output unless the pipeline was to operate under very stringently controlled conditions. What relationship is there between the Steady State Capacity and the Transient Capacity?

Design Concerns

Each of the case studies in this series deals with pipeline operation under non-steady state or transient conditions. The implications and limitations of Transient capacity will be investigated.

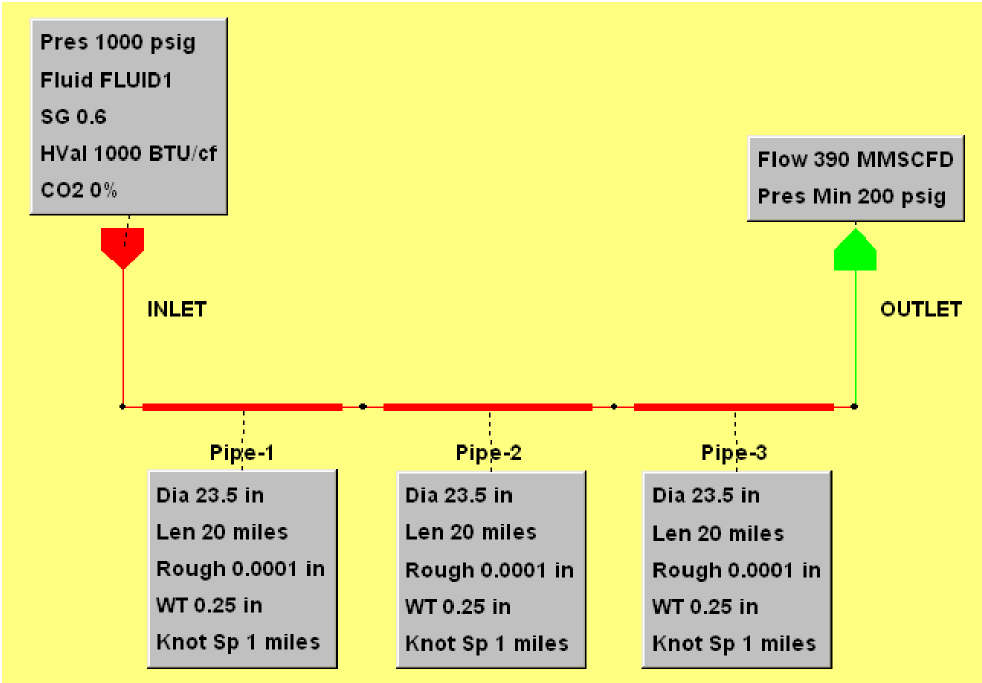
Transient Value Ramping

Transient events in ***pipelinestudio*** are controlled and specified through setpoint value "ramps". These ramps are compiled in the Transient Scenario Table.

Introduction

Expansion of the pipeline's capacity will be made by the addition of a single loop segment. Again, the challenge will be to determine the most effective location for the loop, considering transient operating conditions.

For the purpose of these studies, an additional control will be added to the delivery external regulator. The constraint "maximum flow" will be implemented to control the flow outlet from the pipeline. A demand will then be simulated by inputting a schedule of delivery flow rates expected through this device.



Case2

Case Study 2 (continued)

INSTRUCTIONS

1. Modify the base configuration from Case1. Call it Case2.
2. Construct a model of the pipeline shown with the data and configuration shown.
3. Select the “Temperature Tracking On”
4. Run a Steady Simulation on this base model and record the flow rate.
5. Run a transient simulation using the Transient Scenario Table in the next page.
6. Failure of the pipeline to meet the above schedule will be signaled by a MODE change. View the transient report to see if “OUTLET” changes mode from flow control to pressure control. Note the simulation time at which this occurs. Create another case by copying CASE2. Call it CASE2_1. Add the LOOP segment to the first pipe (Pipe-1). Run a transient simulation and record the time at which “OUTLET” fails to receive the full requested flow.
7. Repeat steps for to assemble cases CASE2_2 & CASE2_3, attaching the LOOP segment to the pipes Pipe-2 and Pipe-3. Record the respective times in each instance.
8. The diameter for the loop line is 31.5 in. All other properties are similar to the main line pipes

Transient Scenario Table: Case2

Time (Hours)	Daily Demand (mmscfd)
1	390
2	391
3	670
4	670
5	780
6	780
7	850
8	750
9	600
10	770
11	600
12	600
13	720
14	720
15	600
16	600
17	720
18	600
19	515
20	432
21	215
22	550
23	620
24	390

Case Study 2 (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

Case Value Input Table

Location of Loop Segment	Time of First Mode Change (Hours)
No Loop Segment	
Loop with Leg L01 <u>(Loop diameter =31.5 inches)</u>	
Loop with Leg L02	
Loop with Leg L03	

1) At what location will adding the loop line provide the most flexibility for handling transient flow variation?

2) Why is this?

Case Study 2 (Transient Flow)

INSTRUCTOR'S SOLUTION

After entering and executing the various model runs, the following answers were obtained:

Location of Loop Segment	Time of First Mode Change (Hours)
No Loop Segment	4:16:28
Loop with Segment LO1	5:29:12
Loop with Segment LO2	6:07:58
Loop with Segment LO3	No mode change

Deleted: 4:16:10

Deleted: 5:28:35

Deleted: 6:07:20

In this Case the transient flow exceeds the steady state capacity of the pipeline. At times, pipe legs show "Drafting", meaning that more gas is leaving the pipeline than entering it. During drafting operations the amount of gas and the length of time drafting can continue is determined by the VOLUME of gas immediately available. The volumetric capacity of the loop segment can be considered to act as a valuable storage device for such "emergencies". When the loop is located at the start or inlet of the pipeline, the additional "storage" capacity is less useful as a "reservoir" because of the distance away from the delivery.

Answers:

1) At what location will adding the loop line provide the most flexibility for handling transient flow variation?

The last segment

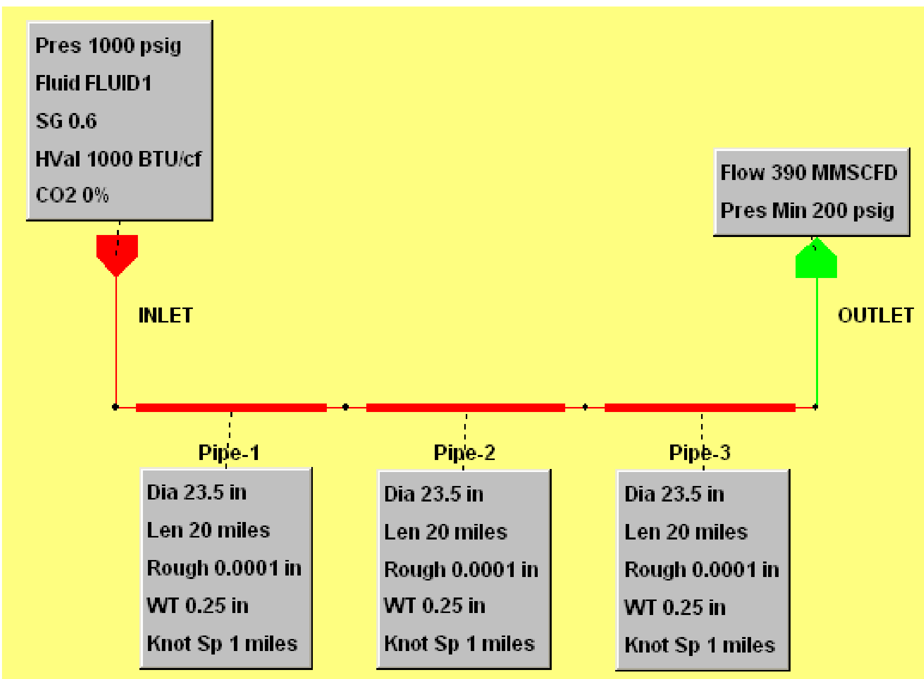
2) Why is this?

Located immediately upstream of the demand, the loop can be used as an effective reservoir.

Case Study 2A

INSTRUCTIONS

1) Create a new case from CASE2 and call it CASE2A.
2) Select trending options as follows (From **Simulation**->**Options**->**Controls**):
    - Trending frequency = maximum
    - Trend the pressure and flow for the supply and the delivery
3) Execute the simulation described in CASE2 with same Transient Scenario Table.
4) Double-click the Trend icon to view the trend output. (Note how the trend output shows the delivery failures)
5) Repeat steps to create files CASE2A_1, 2 & 3 to show looping.
6) Activate the trend viewer. Select for viewing the flow for xreg "OUTLET" from the three cases.



Case2A

Case Study 2A (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) At what loop location are we able to sustain the demand for a longer time
   Use PipelineStudio trends in answering this

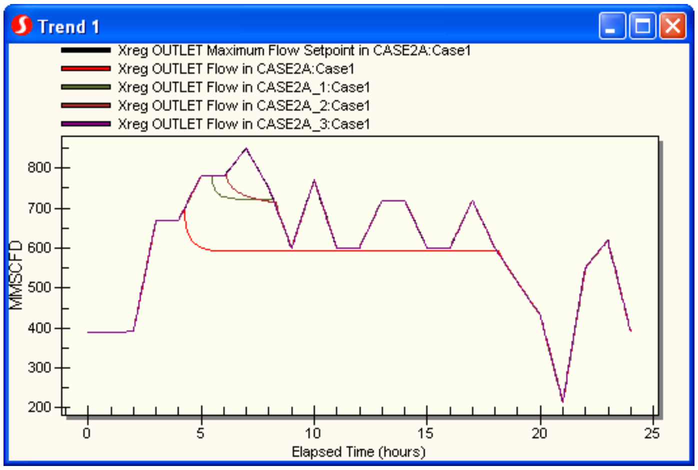
2) Copy the case where the loop is placed over the last segment as case 2B. Model a new delivery at a distance of 15 miles from the downstream of Pipe-2 (T-line). This delivery requires gas at a rate of 200 MMSCFD and at a minimum delivery pressure of 200 psig. Can we meet the flow schedule at the first delivery in this case? Use PipelineStudio trend capability

3) At what time does the mode change happen at the first delivery? Compare this with the earlier case when the second delivery is non-existent.

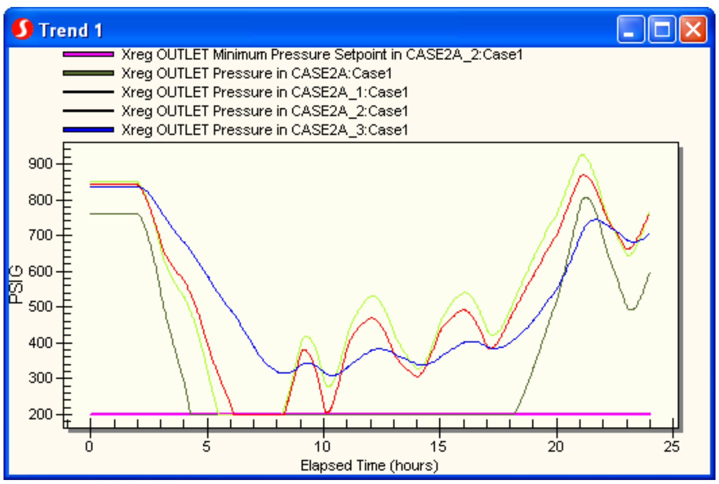
Case Study 2A (continued)

INSTRUCTOR'S SOLUTION

1) At what loop location are we able to sustain the demand for a longer time
Use PipelineStudio trends in answering this



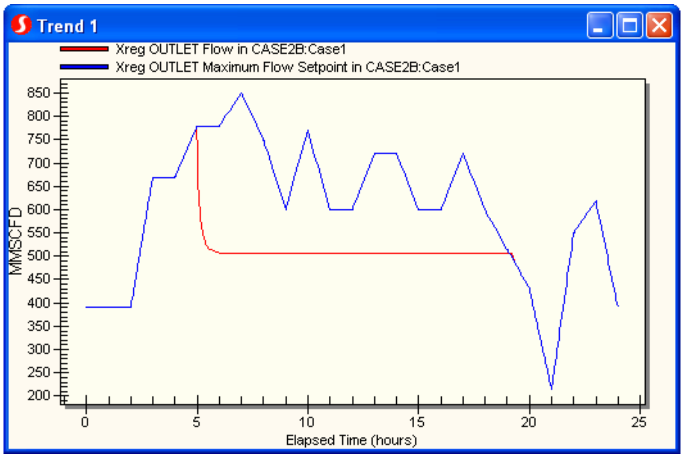
Flow trends at the deliveries



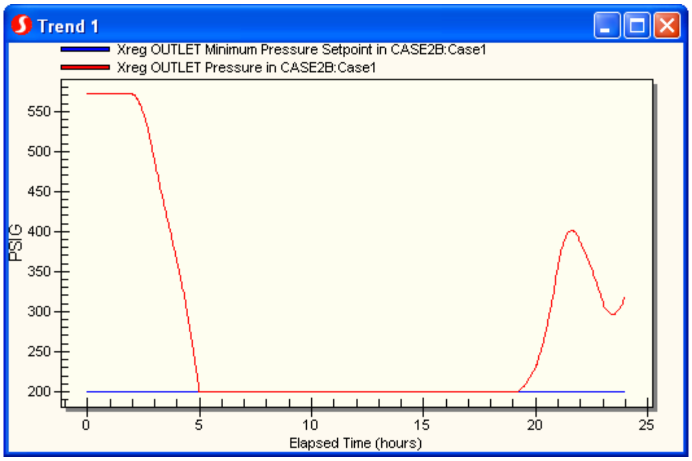
Pressure trends at the deliveries

From the trends it can be seen that the flow schedule can be sustained in the case when the loop is over the last segment

2) Copy the case where the loop is placed over the last segment as case 2B. Model a new delivery at a distance of 15 miles from the down stream of Pipe-2 (T-line).This delivery requires gas at a rate 200 MMSCFD and at a minimum pressure of 200 psig. Can we meet the flow schedule at the first delivery in this case? Use PipelineStudio trend capability



Flow trend at the delivery



Pressure trend at the delivery

It can be seen from the above trends that the flow schedule at the first delivery cannot be attained as the minimum delivery pressure is reached after 5 hours

3) At what time does the mode change happen at the first delivery?
Compare this with the earlier case when the second delivery is non-existent.

The mode change happens at the first delivery after 4:58:59. In the previous case there was no mode change meaning we could attain the flow schedule. However with the presence of a new delivery, the flow nomination at the first delivery cannot be attained.

GAS NETWORK TRAINING

MODULE : SURVIVAL TIME

PipelineStudio Training: Module Survival Time .... **I**
The Module I
The User I
The Training I
Survival Time **1**
Case 1 (Survival Time) **3**

PipelineStudio Training: Module Survival Time

The Module

The module will illustrate the survival time condition or how long does the network can handle a failure trying to comply the parameters (minimum pressure and maximum flow) before changing to critical conditions.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

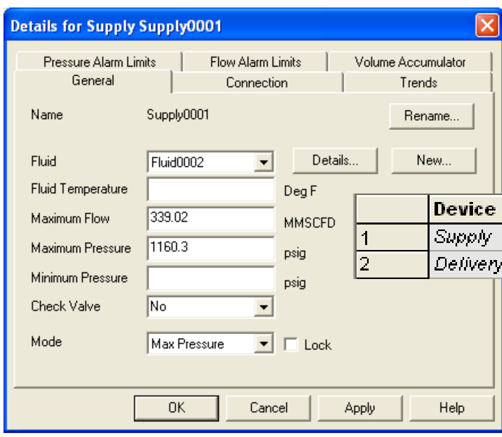
pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Survival Time

Purpose

In addition to generate capacity cases ***pipelinestudio*** could be use to create survival time analysis
To configure a survival time study is very similar to transient capacity case. Taking as our reference figure 1, and the transient developed in figure 2, we can take this a step further.
Let us assume that the source is actually the downstream pressure setpoint of a compressor station. What would happen if that compressor was taken out of service for maintenance? Could we still maintain the nomination or would we have to cut back the flow?
To do this we need to consider how to "switch" the compressor off, since we are just using a source. The easiest way to do this is to place a flow constraint on the source which will not be the initial controlling mode, then during the transient simulation we will drop that flow rate to 0, forcing a mode change. To ensure the flow constraint will not be the initial controlling mode we choose a value which is significantly higher than the maximum steady-state flow figure.
This will simulate the turning off of the compressor. So:



	Device Type	Name	Setpoint	Units	Initial...	0.1	1
1	Supply	Supply0001	Flow Maximum	MMSCFD	339.02	0	
2	Delivery	Deliv0001	Flow Maximum	MMSCFD	254.265	254.265	296.642

Figure 1: Supply and transient scenario dialogue to configure the supply for a survival time example.

What we would expect to see here is that the nomination would be delivered until the pipeline can no longer maintain delivery, and depressurized.

Figure 2 shows the trend output for this case at the delivery, and indicates the system can survive for a period of around 140 minutes. At this point the choice could be made to cut back the demand to preserve the existing line-pack, or to continue until the pipeline is depleted.

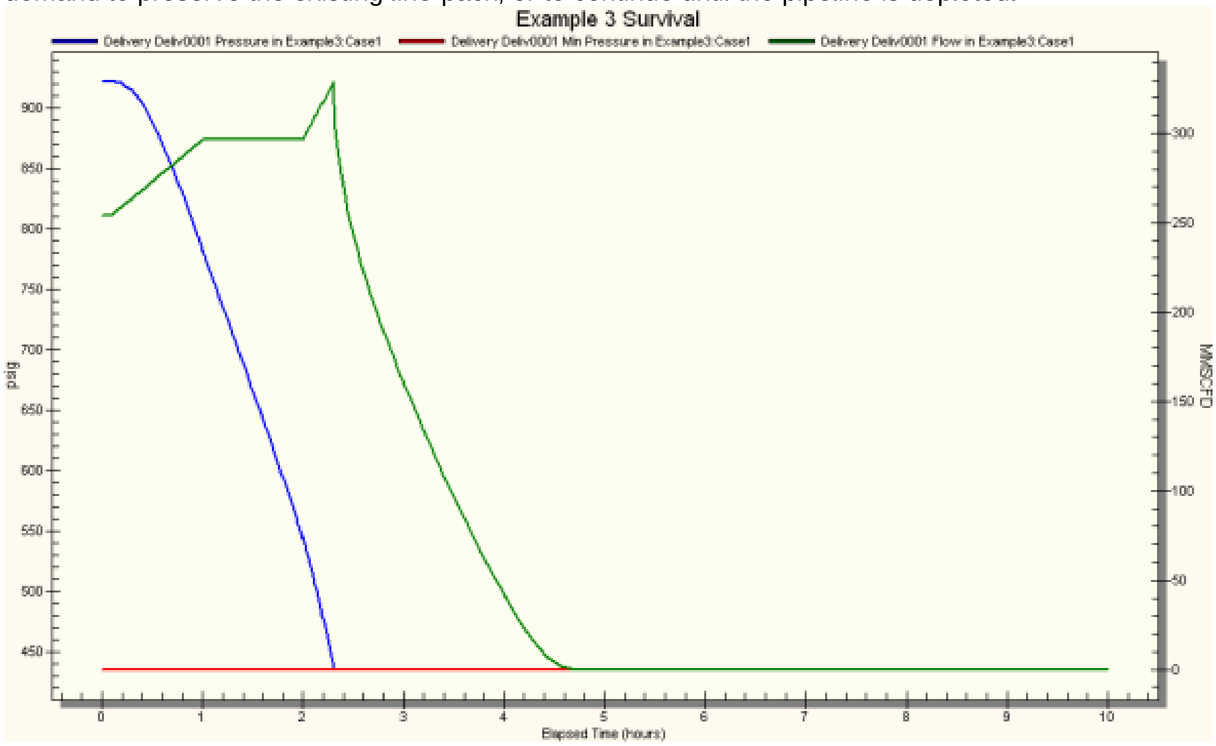


Figure 2: Survival Time example, showing a survival time of around 140 minutes.

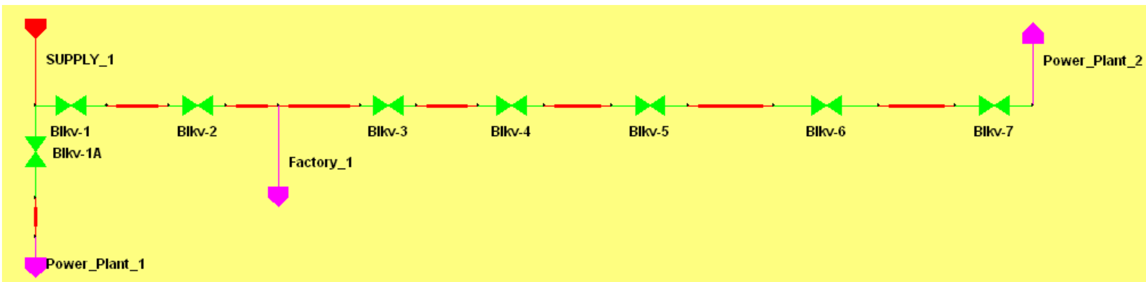
Case 1 (Survival Time)

Purpose

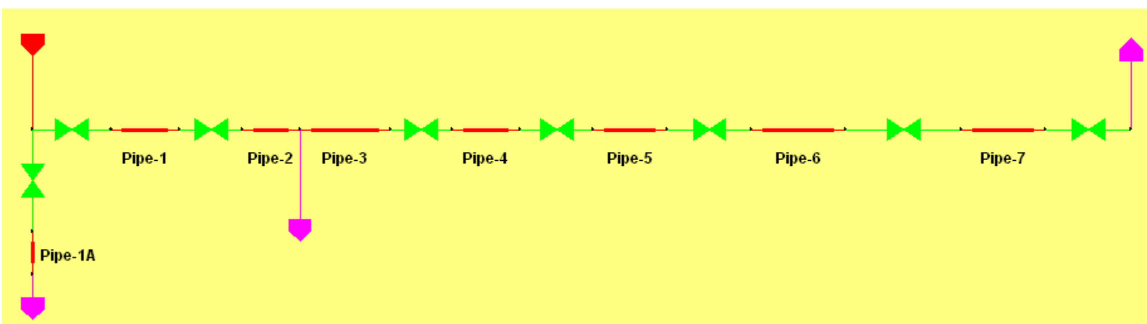
This case study is designed to investigate the Survival Time for a delivery point in a pipeline network; when a section of the pipeline is isolated and the inventory is the only option to continue the operation in that particular network.

Use the configuration layout as a reference to configure the network and use the configuration name: Case1.

Supply, deliveries and block valves names



Pipes names



INSTRUCTIONS

1. Create a new case and call it Case1.
2. Construct a model using the information provided
3. Turn “On” the temperature tracking.
4. Create the transient scenario
5. Select the all necessary trends to review the results
6. Run a Steady State and Transient.

Use the following data to configure the network:

Fluid1 properties:

Specific Gravity	0.694	
Heating Value	1007	BTU/cf
Carbon Dioxide	6.93	percent

Supply_1 details:

Fluid Temperature	80	Deg C
Maximum Pressure	700	Psig

Delivery **Power_Plant_1** details:
(Location: Downstream of the Pipe-1A)

Maximum Flow	28	MMSCFD

Delivery **Factory_1** details:
(Location: Downstream of the Pipe-2 and Upstream of Pipe-3)

Maximum Flow	0.5	MMSCFD
Minimum Pressure	60	Psig

Delivery **Power_Plant_2** details:
(Location: Downstream of the valve Blkv-7)

Maximum Flow	160	MMSCFD

Valves details:

Valve Name	CV	Size in inches	Valve Name	CV	Size in inches
Blkv-1	12000	20	**Blkv-4**	12000	20
Blkv-1A	12000	20	**Blkv-5**	12000	20
Blkv-2	12000	20	**Blkv-6**	12000	20
Blkv-3	12000	20	**Blkv-7**	12000	20

Elevation details:

Network Element Name	Elevation in meters	Network Element Name	Elevation in meters
SUPPLY_1	1200	**Blkv-5**	850
Blkv-2	1100	**Blkv-6**	100
Factory_1	1100	**Power_Plant_2**	0
Blkv-3	1560	**Blkv-1A**	1200
Blkv-4	1270	**Power_Plant_1**	1200

Pipes details:

Pipe Name	Length in Miles	Diameter (inside) in inch	Wall Thickness in inch	Roughness in inch	Knot Spacing in Miles	Efficiency
Pipe-1A	1.242	12	0.375	0.0018	1	0.9
Pipe-1	5.716	19.25	0.375	0.0018	1	0.9
Pipe-2	0.344	19.25	0.375	0.0018	0.344	0.9
Pipe-3	1.519	19.25	0.375	0.0018	1	0.9
Pipe-4	3.447	19.25	0.375	0.0018	1	0.9
Pipe-5	3.111	19.25	0.375	0.0018	1	0.9
Pipe-6	3.437	19.25	0.375	0.0018	1	0.9
Pipe-7	0.621	19.25	0.375	0.0018	0.621	0.9

Use the following table to create a transient scenario. The scenario considers a flow restriction to the customer Power_Plant_2 and normal flow programmed to the Factory_1.

Time (hours)	Block Valve Blkv-1 (%)	Block Valve Blkv-2 (%)	Delivery Power_Plant_2 (MMSCFD)	Delivery Factory_1 (MMSCFD)
1	100	100	160	0.5
1.1	0	0	160	0.5
1.5	0	0	160	0.5
2.1	0	0	0	0.5
3	0	0	0	0.5
4	0	0	0	0.5
5	0	0	0	0.5
6	0	0	0	1
7	0	0	0	2
8	0	0	0	2
9	0	0	0	2
10	0	0	0	2
11	0	0	0	2
12	0	0	0	2
13	0	0	0	2
14	0	0	0	2
15	0	0	0	1
16	0	0	0	0.5
17	0	0	0	1
18	0	0	0	2
19	0	0	0	2
20	0	0	0	2
21	0	0	0	2
22	0	0	0	1
23	0	0	0	0.5
24	0	0	0	0.5
25	0	0	0	0.5
26	0	0	0	0.5

Case 1 (Survival Time) (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) How many hours the delivery Factory_1 could operate before reaching minimum pressure?

2) Use Pipeline Studio graphical capabilities to support the answer to the previous question

Case 1 (Survival Time) (continued)

OBSERVATIONS AND QUESTIONS

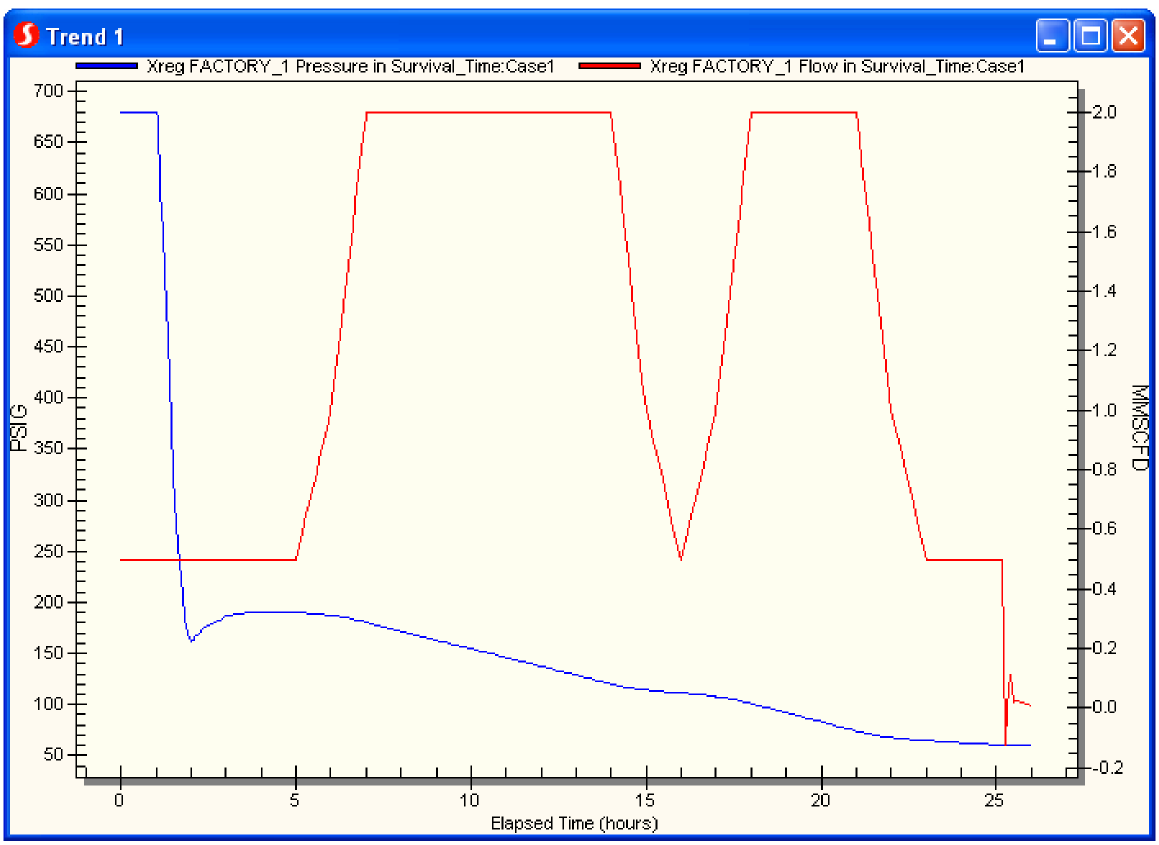
INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) How many hours the delivery Factory_1 could operate before reaching minimum pressure?

23 hours (24 -1)

2) Use Pipeline Studio graphical capabilities to support the answer to the previous question



pipelinestudio®

GAS NETWORK TRAINING

MODULE : THERMAL EFFECTS

energysolutions®

PipelineStudio Training: Module Thermal Effects ..... **I**
The Module I
The User I
The Training I
Introduction To Thermal Calculations in PipelineStudio **1**
Thermal Equations **6**
Case Study – Thermal Effects **12**
Case1 ..... **12**
Case1A **15**
Case1B **18**

PipelineStudio Training: Module Thermal Effects

The Module

The module contains a series of cases; starting with a case that will be used as starting point to generate additional cases.

The cases will illustrate how the thermal effects have influence into the simulations.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Introduction To Thermal Calculations in PipelineStudio

pipelinestudio supports a set of options which permit the configuration of thermal pipeline models; from simple isothermal cases, where an overall system temperature may be used to develop the PVT relationship, up to highly detailed configurations which calculate changes in the fluid properties due to variations in the pipe thermal properties, burial depth and ambient temperature.
The equation which ***pipelinestudio*** uses to govern these changes in temperature is the energy equation, thus:

$$\rho C_v(T_t + uT_x) = -T\left.\frac{dP}{dT}\right|_\rho u_x + \frac{\rho f}{2D}|u|^3 - \frac{4U_w}{D}(T - T_g) \text{ ....1}$$

Where

= Density (slug/ft3)
C_v = Heat capacity of gas (ft.lbf/slug.R)
x = position along the pipe (ft)
t = time (s)
u = Fluid Velocity (ft/s)
T = Fluid Temperature (R)
P = Pressure (psia)
f = Friction Factor
D = Internal Diameter of pipe (ft)
U_w = Overall Heat Transfer Coefficient of material(ft.lbf/slug.R)
T_g = Ground Temperature (R)

Taking each term, the first term to consider is:

$$\rho C_v(T_t + uT_x) \text{ ....2}$$

This term accounts for the convection of heat within the gas.

$$-T\left.\frac{dP}{dT}\right|_\rho u_x \text{ ....3}$$

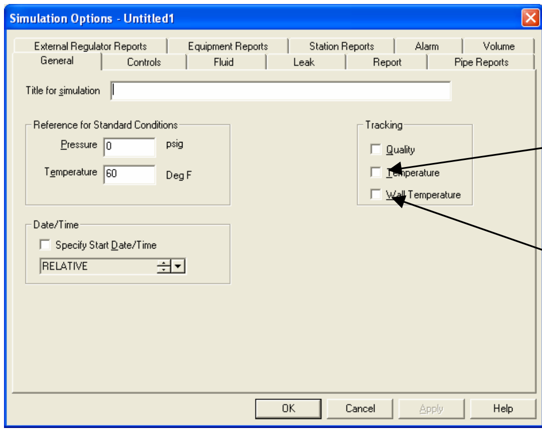
This term accounts for the Joule-Thompson effect. The Joule-Thompson effect is the name given to the decrease in temperature of a gas as it is allowed to expand (this is in accordance with PV=ZRT). The Joule-Thompson effect may also be described as the temperature drop which accompanies the throttling process.
The next term is:

$$+\frac{\rho f}{2D}|u|^3 \text{ ....4}$$

This term accounts for the heat gain of the fluid due to friction caused by the pipe wall.
The final term:

$$-\frac{4U_w}{D}(T - T_g) \text{ ....5}$$

accounts for the temperature loss or gain across the pipeline wall due to heat transfer.
Generally the majority of thermal effects are due to heat transfer, though in, for instance, the situation where there is a high pressure drop a large Joule-Thompson effect may be seen.
Generally there is very little change in fluid temperature due to friction in a gas.
The main option to choose when modelling pipeline thermal effects is to set the temperature tracking. Temperature Tracking status is set in the **Simulation Options | General** dialogue. To set the temperature tracking on, place a check mark in the box by left-clicking with the mouse.

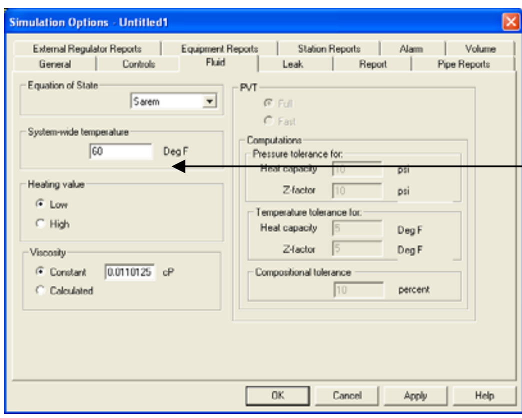


Turn **Temperature** tracking on to investigate thermodynamics

Wall Temperature tracking may be turned on to permit more detailed thermal calculations of pipeline wall layers

Figure 1: Dialogue for Simulation Options | General tab; showing location of temperature tracking and wall temperature tracking

In the case of an isothermal study (where you simply want to use an average gas temperature for the fluid) an overall system temperature may be entered in the **Simulation Options | Fluids** tab. ***pipelinestudio*** uses a default temperature of 60 F or 15.5556 C



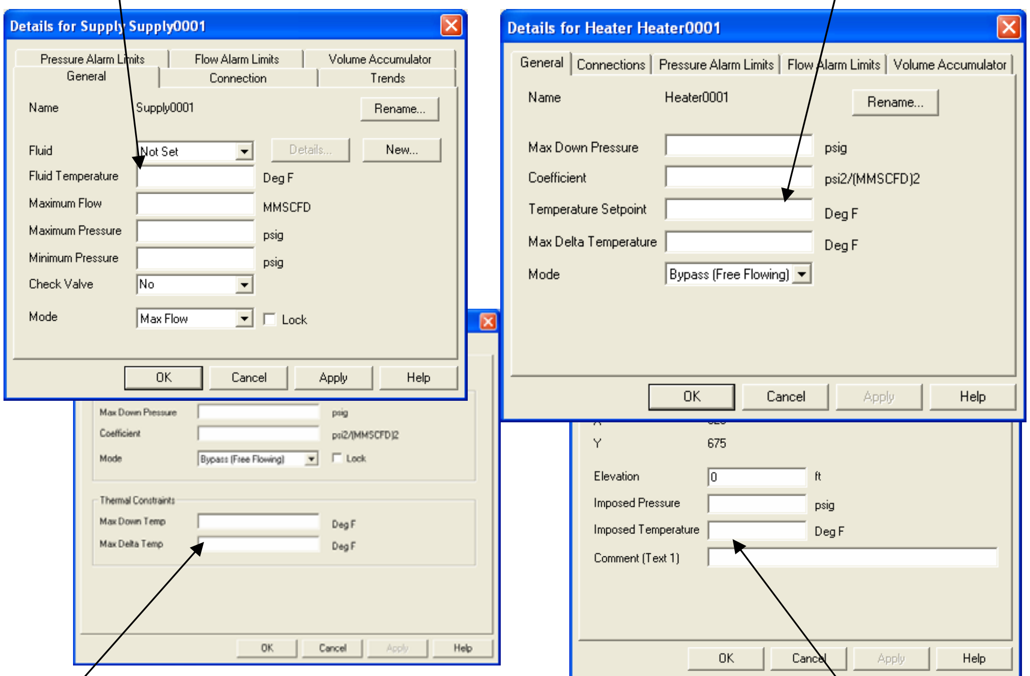
System-wide temperature may be entered here, used if not using temperature tracking and an average fluid temperature is to be used.

Figure 2: Dialogue for Simulation Options | Fluid tab; showing location of system wide temperature

Investigations using temperature tracking require that a fluid temperature be supplied at the source, else the system-wide temperature will be used as the supply fluid temperature. Other equipment which has options permitting temperature variation include compressors, heaters & coolers and nodes.

The **Fluid Temperature** may be entered here

A **Temperature** may be set downstream of a heater or cooler



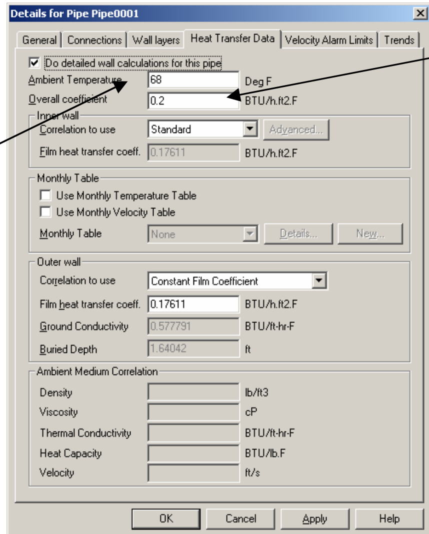
A **Temperature Change** of a certain number of degrees may be entered here

Imposed Temperature at node (meaning the fluid temperature will change to the new temperature)

Figure 3: Dialogue for equipment showing where fluid temperature data may be changed.

The dialogue for pipes permits entry of basic thermal property data, or greater detail in the form of wall layers and heat transfer coefficients.
With only temperature tracking on the pipe dialogue for heat transfer is mainly greyed out, permitting entry of data for only ambient temperature and overall pipe heat transfer coefficient.

Figure 4: Dialogue for general pipe heat transfer data.

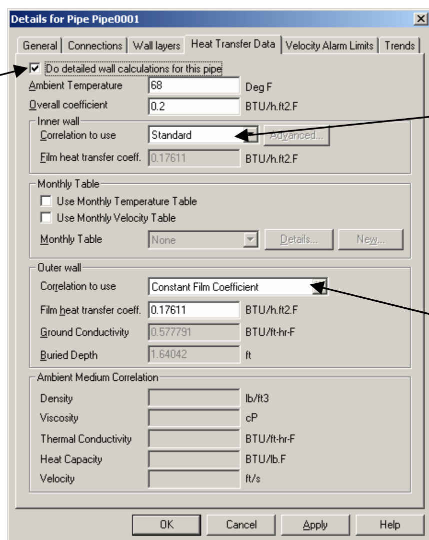


An **overall coefficient** for heat transfer may be entered here

Ambient Temperature may be entered here

If wall temperature tracking is switched on (and note, the simulation speed will be impaired significantly if it is) the rest of the dialogue box becomes active.

Detailed Wall calculations may be specified, in which case wall layers should be specified.



The inner wall **correlation** may be standard, **constant film** (where a film heat transfer coefficient must be supplied, or **user defined**, where the **Nusselt Number** is defined by the user.

The outer wall **correlation** may, **constant film** (where a film heat transfer coefficient must be supplied, or **shallow burial or deep burial**, where ground conductivity and buried depth must be supplied

Figure 5: Dialogue for pipe heat transfer tab, with wall temperature tracking enabled.

The wall layers dialogue permits a more detailed "map" of the pipeline wall layer thermal properties to be built. For example a typical pipe may consist of an inner drag reducing layer, a steel shell and an outer concrete casing.

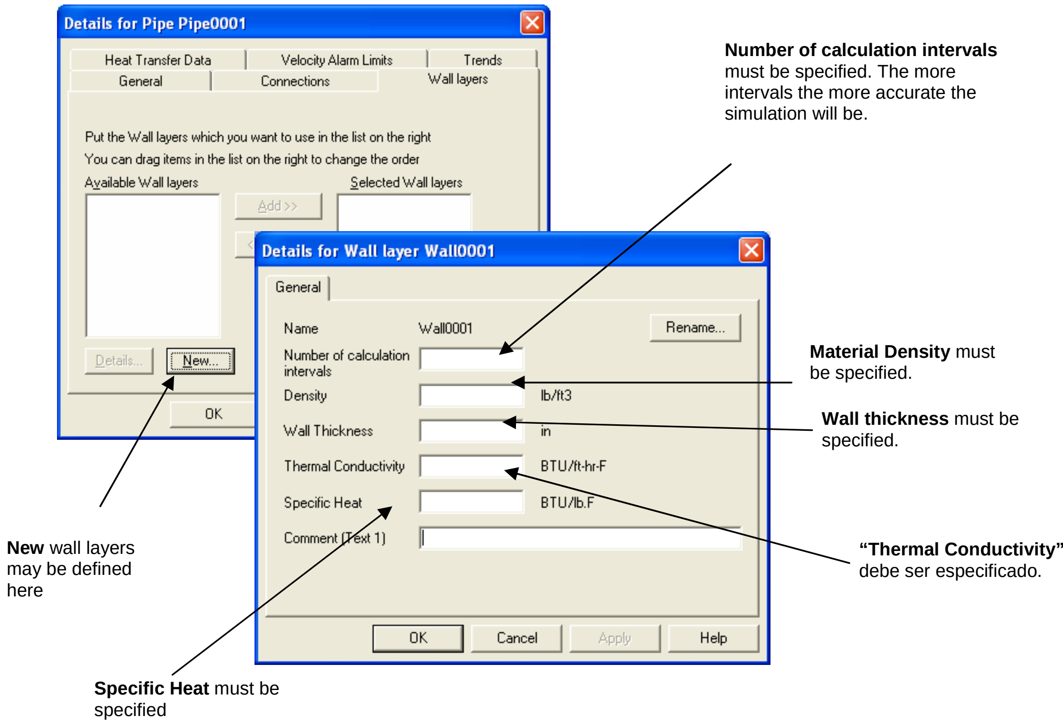


Figure 6: Dialogue for wall layers

Thermal Equations

General Heat Transfer

Given a fully buried bare pipe, the overall heat transfer coefficient is computed with thermal conductivity figures as follows:

$$\frac{1}{U} = \frac{R_i}{R_o} \times \left[\frac{1}{k_{env}} + R_o \times \ln\left(\frac{R_o}{R_i}\right) \frac{1}{k_{steel}} \right]$$

Where:

U = overall heat transfer coefficient, BTU/hr·ft^{2}·°F
R_o = radius to outside of pipe wall, feet
R_i = radius to inside of pipe wall, feet
k_{steel} = thermal conductivity of steel, BTU/hr·ft^{2}·°F
k_{env} = thermal conductivity of environment (air, water, soil, etc.), BTU/hr·ft^{2}·°F

Heat Transfer with Multiple Wall Layers

If the pipe is insulated or coated, thermal energy must pass through several "shells" to exit the pipe. These multiple resistances may be combined into a single heat transfer coefficient by the following:

$$\frac{1}{U} = \frac{R_i}{R_o} \times \left[\frac{1}{k_{env}} + R_o \times \sum_{n=1}^{\#ofshells} \left(\ln\left(\frac{R_n}{R_{n-1}}\right) \frac{1}{k_n} \right) \right]$$

Where:

U = overall heat transfer coefficient, BTU/hr·ft^{2}·°F
R_o = radius to outside of pipe wall, feet
R_i = radius to inside of pipe wall, feet
R_n = radius to outside of layer "n", feet
k_n = thermal conductivity of layer "n", BTU/hr·ft^{2}·°F
k_{env} = thermal conductivity of environment (air, water, soil, etc.), BTU/hr·ft^{2}·°F

Note: in this equation, $R_o = R_n$ and $R_i = R_1$

Given a fully buried bare pipe, the overall heat transfer coefficient can be computed from conductivity as follows:

$$\frac{1}{U} = \frac{1}{h_f} + \frac{1}{h_p} + \frac{1}{h_s} + \frac{1}{h_i}$$

Where:

U = overall heat transfer coefficient, BTU/hr·ft^{2}·°F
h_f = pipe wall film resistance, BTU/hr·ft^{2}·°F
h_p = pipe thermal resistance, BTU/hr·ft^{2}·°F
h_s = soil (surroundings) thermal resistance, BTU/hr·ft^{2}·°F
h_i = pipe coating or insulation thermal resistance, BTU/hr·ft^{2}·°F

The Overall Heat Transfer Coefficient used to calculate heat transfer per unit pipe length is calculated based on a combination of terms:

- Heat conduction through the surroundings (i.e. rock, soil, water)
- Heat conduction through pipe wall (often includes insulating layer)
- Convection heat transfer from the pipe to the fluid
- Heat conduction through the pipe coating or insulation

To calculate the thermal resistances associated with these different forms of heat transfer, the following formulae are used:

$$h_p = \frac{2k_p / d_o}{\ln(r_o / r_i)}$$

$$h_f = C_h N_{re} a$$

"h_s" for Deep Burial (invoked by specifying Deep Burial, should be used for modelling heat transfer in pipes buried deeper than twice their outside diameter):

$$h_s = \frac{2k_s / d_o}{\ln(\frac{4h}{d_o})}$$

"h_s" for Shallow Burial (invoked by specifying Shallow Burial, should be used for pipes buried less than twice the pipeline diameter)

$$h_s = \frac{2k_s / d_o}{\ln(x + \sqrt{x^2 - 1})}$$

$$h_i = \frac{2k_c / d_c}{\ln(r_c / r_o)}$$

Where:

C_h	= Pipe wall film coefficient, BTU/hr·ft^{2}·°F (3.6 x 10^{-4} default)
N_{re}	= Reynolds number
a	= Reynolds number exponent (0.8 default)
k_p	= Pipe thermal conductivity, BTU/hr·ft·°F (26 default)
k_s	= Soil (surroundings) thermal conductivity, BTU/hr·ft·°F (1.0 default)
k_c	= Pipe coating or insulation thermal conductivity, BTU/hr·ft·°F
d_o	= Pipe outside diameter, feet
r_o	= pipe outside radius, feet
r_i	= pipe inside radius, feet
r_c	= pipe coating or insulation outside diameter, feet
h	= Burial depth, to centerline of pipe, feet (d/2 default)
x	= $2h/d_o$

If more precise heat transfer and temperature profile information is required, it is necessary to do detailed wall temperature tracking calculations. This is done by selecting Wall Temperature Tracking (in **Simulation | Options | General** tab) and checking the "Do detailed wall calculation for this pipe" box in the Details for Pipe dialogue box. Un-checking this box gives you a way to override the wall temperature tracking for the particular pipe. Note, if wall temperature tracking (in **Simulation | Options | General** tab) is off, this entry has no effect.

There are a number of different ways to set up the detailed wall tracking option. A single wall layer that represents the pipe wall or multiple wall layers may be specified. ***pipelinestudio*** (gas) is capable of simulating up to 5 different wall layers.

For heat transfer between the gas and the inside pipe wall and between the outside wall layer (be it pipe or another coating) and the surrounding medium, the following different options are available:

Inside Wall Heat Transfer

1. **Constant Film Coefficient** – Simply select this correlation as the one to use and insert the inner wall heat transfer coefficient that you wish to use for all inside wall heat transfer calculations.

2. **Standard Film Coefficient** – If the standard correlation is selected, **pipelinestudio** (gas) uses the following equations to calculate the inner wall heat transfer coefficient:

$$Nu = 0.023\,\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.40}$$ For turbulent flow (Re > 2100)

$$Nu = 1.62\left(\frac{\mathrm{Re}\,\mathrm{Pr}\,D_i}{L}\right)^{1/3}$$ For laminar flow (Re $\leq$ 2100)

Where:

Nu = Nusselt Number *(=* $h_f d_i/k_f$*)*
Re = Reynolds Number *(=* $\rho_f u_f d_i/\mu_f$*)*
Pr = Prandtl Number *(=* $Cp_f \mu_f/k_f$*)*
d_i = inside pipe diameter, feet
k_f = fluid thermal conductivity*, (default = 0.015 BTU/hr ft$^{2 \cdot o}$F)*
ρ_f = fluid density*, lb$_m$/ft^3*
u_f = fluid velocity*, ft/s*
μ_f = fluid viscosity, *(default = lb$_m$/ft.s)*
Cp_f = fluid heat capacity, *BTU/lb$_m$$\cdot^o$F*

3. The **User Defined** Correlation – For the user defined correlation, the user has the option of defining their own coefficients in the following equation:

$$Nu = A\,\mathrm{Re}^{B}\,\mathrm{Pr}^{C} + D$$ For fully turbulent flow (Re >2100)

$$Nu = 1.62\left(\frac{\mathrm{Re}\,\mathrm{Pr}\,D_i}{L}\right)^{1/3}$$ For laminar flow (Re $\leq$ 2100)

Where:

Nu = Nusselt Number *($h_s d_o/k_s$)*
Re = Reynolds Number *(= $\rho_s u_s d_o/\mu_s$)*
Pr = Prandtl Number *(= $Cp_s \mu_s/k_s$)*
d_o = outside pipe diameter*, feet*
k_s = thermal conductivity of surrounding air/water, *BTU/hr·ft$^{2\cdot o}$F*
ρ_s = density of surrounding air/water, *lb$_m$/ft^3*
u_s = velocity of surrounding air/water, *ft/s*
μ_s = viscosity of surrounding air/water*, lb$_m$/ft.s*
Cp_s = heat capacity of surrounding air/water*, BTU/lb$_m$·^{o}F*

External conditions are based on the external medium properties. The velocity is the local velocity over the pipeline, assumed to act as an isolated cylinder with flow vertical to the pipe axis.

Outside Wall Heat Transfer:

1. Select the **Constant Film Coefficient** – Simply insert the heat transfer coefficient for the outside wall to be used for all calculations.

2. **Shallow Burial Method** – Simply invoke the Shallow Burial Coefficient and enter the ground conductivity and burial depth. Shallow burial should be specified for modelling heat transfer in pipes buried less than twice their outside diameter. The following equation is then used to determine the outside heat transfer coefficient.

$$h_s = \frac{2k_s / d_o}{\ln(x + \sqrt{x^2 - 1})}$$

Where:

k_s = Soil (surroundings) thermal conductivity, BTU/hr·ft·^{o}F (1.0 default in ***pipelinestudio*** (gas))
d_o = Pipe outside diameter, ft
h = Burial depth, to center line of pipe, ft (d/2 default)
x = $2h/d_o$

3. **Deep Burial Method** – Simply invoke the Deep Burial Coefficient and enter the ground conductivity and burial depth. Deep burial should be specified for modelling heat transfer in pipes buried deeper than twice their outside diameter. The following equation is then used to determine the outside heat transfer coefficient.

$$h_s = \frac{2k_s / d_o}{\ln(\frac{4h}{d_o})}$$

Where:

k_s = Soil (surroundings) thermal conductivity, BTU/hr·ft·°F (1.0 default in ***pipeline*studio** (gas))
d_o = Pipe outside diameter, ft
h = Burial depth, to center line of pipe, ft (d/2 default)

4. The **User Defined** Correlation – For the user defined correlation, the user has the option of defining their own coefficients in the following equation (for fully turbulent flow):

$$Nu = A\,\mathrm{Re}^B\,\mathrm{Pr}^C + D$$ (For fully turbulent flow Re >2100)

$$Nu = 1.62\left(\frac{\mathrm{Re}\,\mathrm{Pr}\,D_i}{L}\right)^{1/3}$$ (For laminar flow Re $\leq$ 2100)

External conditions are based on the surrounding external properties. In the case of seawater, the velocity is the local velocity over the pipeline, assumed to act as an isolated cylinder with flow vertical to the pipe axis.

Example coefficients for modelling a sub-sea pipeline are as follows:

$$Nu = 0.86\,\mathrm{Re}^{0.43}\,\mathrm{Pr}^{0.30}$$ (For 0.2 < Re $\leq$ 200)

$$Nu = 0.26\,\mathrm{Re}^{0.60}\,\mathrm{Pr}^{0.30}$$ (For Re > 200)

Where:

Nu = Nusselt Number *(*$h_s d_o / k_s$*)*
Re = Reynolds Number *(=* $\rho_s u_s d_o / \mu_s$*)*
Pr = Prandtl Number *(=* $Cp_s \mu_s / k_s$*)*
d_o = outside pipe diameter*, feet*
k_s = thermal conductivity of surrounding air/water*, BTU/hr·ft*^{2}*·°F*
ρ_s = density of surrounding air/water, *lb*$_m$*/ft*^{3}
u_s = velocity of surrounding air/water*, ft/s*
μ_s = viscosity of surrounding air/water*, lb*$_m$*/ft.s*
Cp_s = heat capacity of surrounding air/water, *BTU/lb*$_m$*·°F*

The heat lost or gained per unit pipe length due to heat transfer is determined using the following equation:

$$Q = 2\pi r_i U\left(T_f - T_a\right)$$

Where:

Q = heat loss/gain from the fluid per unit pipe length, *BTU/ft*
T_f = mean fluid temperature, *^oF*
T_a = mean ambient temperature, *^oF*

The energy balance used in Temperature Tracking determines the mean fluid temperature. The ambient temperature is input directly into **pipelinestudio** (gas).

Case Study – Thermal Effects

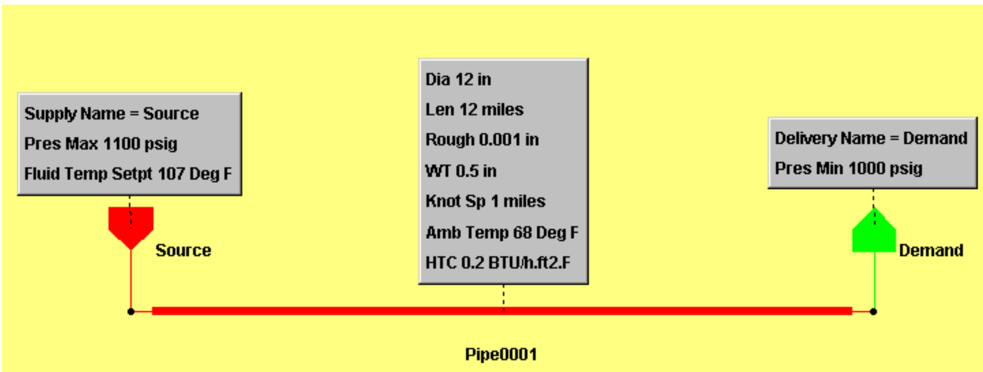
Case1

Purpose

This case study is designed to investigate the effects of pipeline thermal options.

The Scenario

This Case Study will require the use of one supply, pipe and delivery point, the thermal basic options will be use to demonstrate the effects.



The configuration network is as follow: The pressure from the injection point is 1100 psig (Source), and the pressure in the delivery (Demand) is 1000 psig, the supply and the delivery are separated by 12 miles of pipeline.

The fluid properties provided are compositional

Component	Percentage
C1	80%
C2	12%
C3	4%
IC4	2%
IC5	2%

The ambient temperature is 68 °F, and the fluid inlet temperature is 107 °F.

INSTRUCTIONS

1) Create a new case based on the schematic shown and the information given for CASE1

2) Select the equation of state BWRS

3) Input the thermal information and turn the temperature tracking “On”

4) Make the necessary changes to answer the questions

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) Determine the approximate flow rate through the pipeline, assuming Pipeline Studio default data where none is given

Flow at delivery Demand =

2) The minimum delivery temperature must be no less than 95 F. Determine the required overall HTC, assuming the demand is set to flow control. What is the pressure at the delivery?

Pressure at delivery Demand = Heat Transfer coefficient (HTC) =

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) Determine the approximate flow rate through the pipeline, assuming ***pipelinestudio*** default data where none is given

Flow at delivery Demand = 93.1935 MMSCFD

2) The minimum delivery temperature must be no less than 95 F. Determine the required overall HTC, assuming the demand is set to flow control. What is the pressure at the delivery?

Pressure at delivery Demand = 999.91 psig Heat Transfer coefficient (HTC) = 0.18669 BTU/h.ft^2.F

Case1A

Purpose

The configuration created in case1 will be augmented to include wall layers to investigate additional pipeline thermal options.

The Scenario

This Case Study will require the use of one supply, pipe and delivery point, the thermal basic options will be use to demonstrate the effects.

Use a copy of the configuration CASE1 and return the overall heat transfer coefficient to the default value = 0.2 BTU/h.ft^2.F.

Formatted: Superscript

Use the following table that shows the materials available to protect the pipe and the thickness. The pipe will be coated with a protective coating to limit corrosion, and the pipe itself is constructed of steel.

The pipe will then be buried in a special insulating mixture.

Deleted: insulative

Material	Density (lb/ft^3)	Wall Thickness (inch)	Thermal Conductivity (BTU/ft-hr-F)	Specific Heat (BTU/lb.F)	Number of calculations intervals
Coating-1	81.1563	0.023622	0.231116	0.453807	3
Steel-1	486.938	0.5	3.00451	0.191077	5
Mixture-1	156.07	?	0.0866686	2.26904	5

Formatted: Superscript

INSTRUCTIONS

1) Create the new case CASE1A based on the CASE1 and update it using the information given

2) Turn the wall temperature tracking “On”

3) Input the wall layer information

4) Make the necessary changes to answer the question

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What is the minimum thickness of this mixture to maintain a delivery temperature 95 °F?

Make the assumption the constant film heat transfer coefficient = 0.17611 BTU/h.ft^2.F.

Formatted: Superscript

Wall Thickness = inches

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) What is the minimum thickness of this mixture to maintain a delivery temperature 95 °F?

Make the assumption the constant film heat transfer coefficient = 0.17611 BTU/h.ft^2.F.

Formatted: Superscript

Wall Thickness = 0.856 inches

Deleted: 0.899

Case1B

Purpose

The configuration created in case1A will be augmented to include wall layers to investigate and compare the additional pipeline thermal options.

The Scenario

Using a copy of the configuration CASE1A make changes according to the following table:

Inner Wall Correlation	Outer Wall Correlation	Inner Wall film HTC (BTU/h.ft^2.F)	Outer Wall film HTC (BTU/h.ft^2.F)	Ground Conductivity (BTU/ft-hr-F)	Buried Depth (ft)	Delivery Temperature (°F)
Standard	Constant Film	-	0.17611	-	-	
Constant Film	Constant Film	0.17611	0.17611	-	-	
User-defined	Constant Film	Default	0.17611	-	-	
Standard	Shallow Burial	-	-	2.88895	1.640	
Standard	Deep Burial	-	-	2.88895	13.123	
Constant Film	Shallow Burial	0.17611	-	2.88895	1.640	
Constant Film	Deep Burial	0.17611	-	2.88895	13.123	
User-defined	Shallow Burial	Default	-	2.88895	1.640	
User-define	Deep Burial	Default	-	2.88895	13.123	

Formatted Table

Formatted: Superscript

Formatted: Superscript

INSTRUCTIONS

1) Create the new case CASE1B based on the CASE1A and update it using the information given
2) Input the information and create individual configurations based on the table provided
3) Make the necessary changes to fill the column "Delivery Temperature" and answer the questions

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) Which type of burial would be better to maintain the temperature in the pipeline and the delivery ?

2) Is the assumption to use a value of 0.17611 BTU/h.ft^2.F a reasonable one?

Formatted: Superscript

OBSERVATIONS AND QUESTIONS

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) Which type of burial would be better to maintain the temperature in the pipeline and the delivery ?

Constant Film Constant Film = 0.17611 Temperature obtained =98.4574 °F

Deleted: 0.17611

2) Is the assumption to use a value of 0.17611 BTU/h.ft^2.F a reasonable one?

Formatted: Superscript

Yes. If no other information is available, it is a good reasonable value

Deleted: t

Inner Wall Correlation	Outer Wall Correlation	Inner Wall film HTC (BTU/h.ft^2.F)	Outer Wall film HTC (BTU/h.ft^2.F)	Ground Conductivity (BTU/ft-hr-F)	Buried Depth (ft)	Delivery Temperature (°F)
Standard	Constant Film	-	0.17611	-	-	95.009
Constant Film	Constant Film	0.17611	0.17611	-	-	98.461
User-defined	Constant Film	Default	0.17611	-	-	95.009
Standard	Shallow Burial	-	-	2.88895	1.640	79.4672
Standard	Deep Burial	-	-	2.88895	13.123	82.4974
Constant Film	Shallow Burial	0.17611	-	2.88895	1.640	96.3589
Constant Film	Deep Burial	0.17611	-	2.88895	13.123	96.5928
User-defined	Shallow Burial	Default	-	2.88895	1.640	79.4672
User-define	Deep Burial	Default	-	2.88895	13.123	82.4974

Formatted Table

Deleted:

Formatted: Superscript

Formatted: Superscript

Deleted:

Deleted: 95.0072

Deleted: 98.4574

Deleted: 95.0072

pipelinestudio®

GAS NETWORK TRAINING

MODULE : COMPRESSORS

energysolutions®

PipelineStudio Training: Module Compressors.. I
The Module I
The User I
The Training I
Case Study 1 (Simple Network) 1
Case Study 1A (Generic Compressor) 7
Case Study 1B (Generic Driver) 14
Case Study 1C (Centrifugal Compressor) 20

PipelineStudio Training: Module Compressors

The Module

The module contains a series of cases; starting with a case that will be used as starting point to generate additional cases.

The cases will illustrate how to use a generic compressor, driver and centrifugal compressor.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1 (Simple Network)

Purpose

With this case study we begin looking at more practical network examples in which the interaction of the various pipeline elements is significant.

Important Elements

Transient versus Steady State Behavior

This case model illustrates the concerns between transient and steady state hydraulic behaviour. A pipeline may be capable of considerably more short-term capacity than might be obvious from the steady state results.

Design Concerns

Each of the case studies in this series deals with pipeline capacity and the ability to match supply and delivery requirements under various operating conditions.

Skill Advancements

As each of the case studies progress, less and less model input will be provided. The student is expected to draw from previous case studies, user guides and online help.

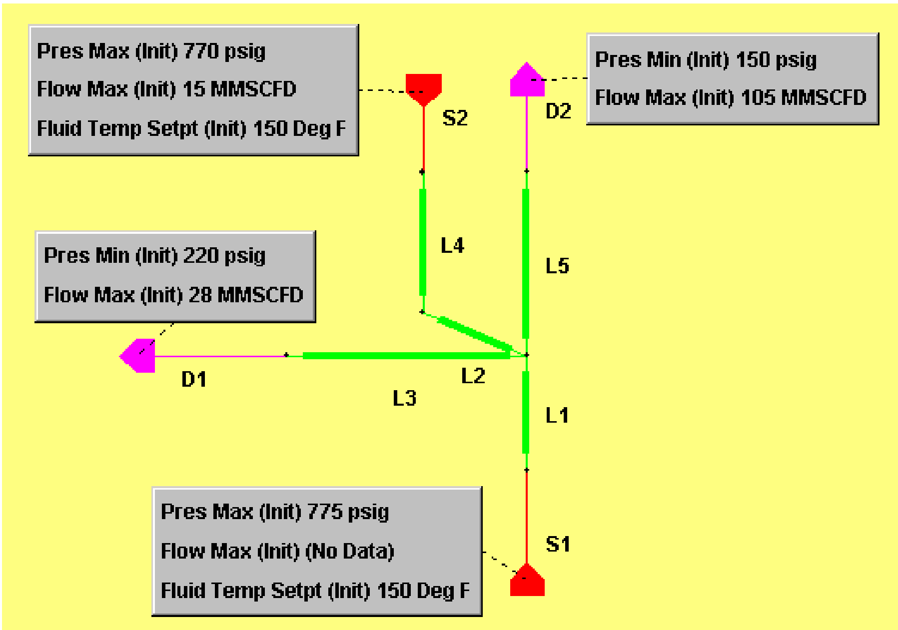
The Scenario

Your main supply, S1, is a gas plant that can deliver an unlimited amount of gas at a constant pressure of 775 PSIG. Another minor supply, S2, can supply up to 15 MMSCFD at a maximum pressure of 770 PSIG.

Case Study 1 (continued)

The system consists of two major gas deliveries, D1 and D2. The power plant (D1) has a constant demand of 28 MMSCFD. The minimum contract delivery pressure for this plant is 220 PSIG. The delivery point (D2) has a constant demand of 105 MMSCFD.

D2 requires an increase in demand to 132 MMSCFD for a 4-hour period. The minimum delivery pressure at D2 is 150 PSIG.



Case1

Supply and Delivery Summary

XREG Name	Flow (MMSCFD	Pressure (PSIG)
S1	?	775
S2	15	770
D1	28	220
D2	105	150

Case Study 1 (continued)

Pipeline Segment Data

Pipe Name	Length (Miles)	ID (Inches)	Roughness (Inches)
L1	37	14.876	0.00250
L2	12	16.876	0.00250
L3	30	10.020	0.00115
L4	17	16.876	0.00050
L5	29	17.350	0.00250

Gas Properties:

Fluid Name	SG	HV (BTU/CF)	CO_2 (%)
Fluid1	0.615	1050	1.4

Use the following data:

GAS TEMPERATURE = 150 ^{o}F
Ambient Temperature = 70 ^{o}F
Wall Thickness = 0.25 inches
Knot spacing = 1.0 miles
Overall heat transfer coefficient = 1 BTU/h.ft2.F

TRANSIENT SCENARIO TABLE: CASE 1 (To be used to answer Q2)

Scheduled Time (hours)	Daily Demand D2 (MMSCFD)
1	132
5	132
6	105
24	105

Case Study 1 (continued)

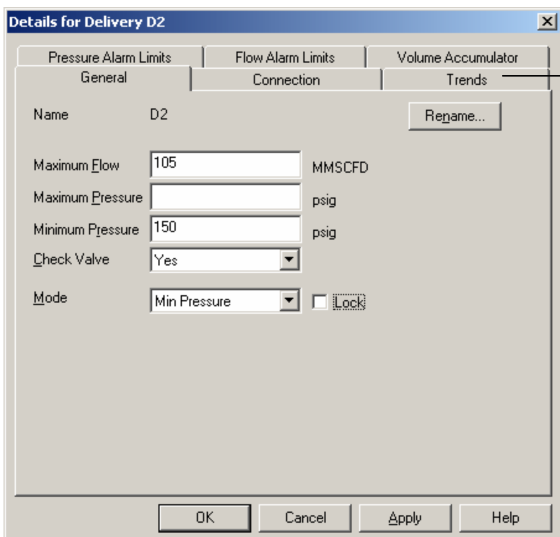
INSTRUCTIONS

1) Configure S1 & S2 with check valves to ensure no gas can return to the supplies. A check valve is a mechanical device, which allows gas flow through it in only one direction. A check valve, as defined by the simulator, is a device that will prevent reverse flow. Reverse flow is defined as flow that passes from the downstream node of the check valve to the upstream node. Upon detecting a negative flow, the check valve closes and remains closed as long as the upstream pressure is less than the downstream pressure. Check valves in the simulator are always either fully open or fully closed.

2) A check valve has two available modes of control: Closed and Open.

3) Use the data provided for the supplies, deliveries and the pipes.

4) Select the appropriate trends at the supplies and deliveries to collect the data needed to answer the questions. The trends have to be selected from the dialogue box by selecting the 'trend' tab for the corresponding external regulator as shown



Trends to be selected from here

5) Modify the base model and make any additional runs needed to answer the questions

6) Turn "on" temperature tracking

7) Create a transient scenario table to match the delivery schedule provided for D2

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the Steady State pressures in the system with D1 and D2 delivery rates of 28 and 105 mmscfd, respectively?

Pressure at D1= Pressure at D2=

2) What is the impact of the delivery increase at D2? (Hint: see trend plot)?

3) Could contract conditions be maintained at D1 if delivery D2 continued taking gas at 132 mmscfd on this schedule during 72 hours?

Yes/No

4) What is the maximum steady state flow that can be delivered on this pipeline system?

Case Study 1 (continued)

INSTRUCTORS' SOLUTION

1) What are the Steady State pressures in the system with D1 and D2 delivery rates of 28 and 105 mmscfd, respectively?

Pressure at D1 = 333.263 psig

Deleted: 333.263

Pressure at D2 = 352.127 psig

Deleted: 351.153

2) What is the impact of the delivery increase at D2? (Hint: see trend plot)?

Initially, no impact. However, as the line pack is reduced, the pipeline cannot continue to deliver 132 mmscfd and meet the minimum delivery pressure.

3) Could contract conditions be maintained at D1 if delivery D2 continued taking gas at 132 mmscfd on this schedule during 72 hours ?

No, after 30 hours you see the minimum pressure at D1 is starting to impact required flow

Deleted: 48

4) What is the maximum steady state flow that can be delivered on this pipeline system?

Total= 143.85 mmscfd D1=26.397 mmscfd and D2=117.452 mmscfd;

Deleted: 143.758

Deleted: 26.381

Deleted: 117.377

Case Study 1A (Generic Compressor)

Purpose

The simple network created in Case1 will be augmented in this Case Study to include a Generic compressor.

Important Elements

Generic Devices

pipelinestudio Gas permits the inclusion of several minimally specified devices into the simulation configurations. Generic devices are those which have limited control and which require minimal unit specifications. They are ideal for preliminary network analysis where detailed information or equipment specifications are unknown.

Controlling Generic Compressors

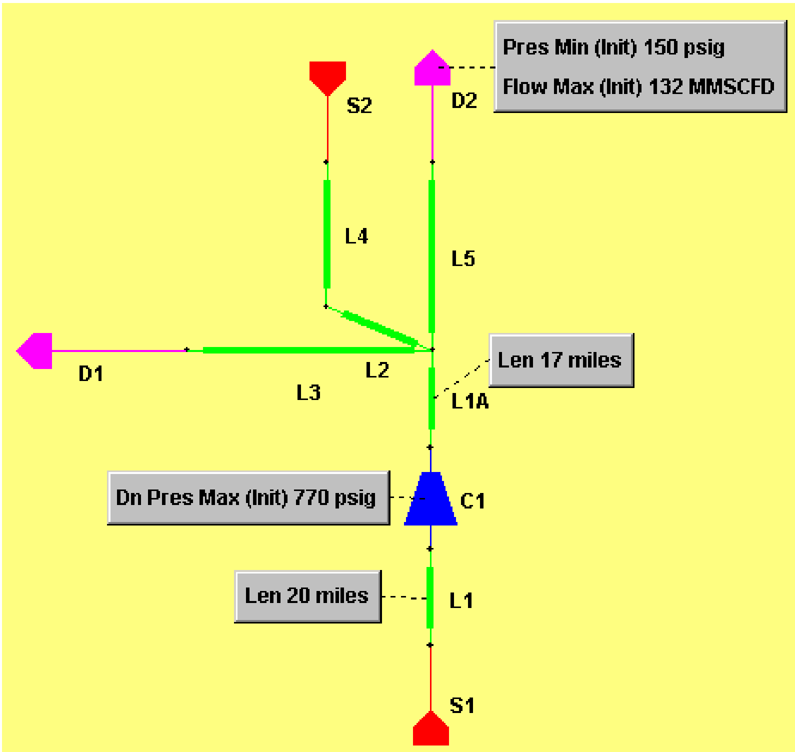
This Case Study will require the use of a generic compressor with controllability for turning the unit on or off. Generic compressors can be controlled, including the ability to "shut down", through their primary pressure or flow controls. In this Case Study we will "ramp" the compressor status to "off". Another method you can try is to ramp the discharge pressure to a small value. This will force the unit into an off condition (since it cannot maintain the required set point).

The Scenario

It has been decided to increase the capacity of the pipeline by adding a compressor station. The only location for this station is 20 miles downstream of the supply S1. The required discharge pressure from this station is 770 PSIG. The delivery point (D2) will have a constant demand of 132 MMSCFD

A problem arises, however, with the compressor station. The compressor has to be shut off everyday for 4 hours.

Case 1A (Generic Compressor)



Case1A

Transient Scenario Table: Case 1A
Maximum Pressure Setpoint for C1(Compressor):

Time (Hours)	Compressor Status
1	ON
1.3	OFF
5	OFF
5.3	ON
24	ON

Case Study 1A (continued)

INSTRUCTIONS

1) Create a new case based on the schematic shown and the information given for CASE1A
2) Create a transient scenario table to match the scheduled operations for the compressor
3) Select the appropriate trend categories to answer the questions
4) Make additional data files and modifications as required

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at D1 and D2?

Pressure at D1

Pressure at D2

2) With the compressor operating, can D2 take gas at the new higher rate without impacting D1?

Yes/No

3) Can minimum gas delivery conditions be maintained at D1 and D2 if the compressor is shut off for four hours?

Yes/No

4) The compressor has gone on annual operational maintenance and it will restore normal operations only after 8 hours. Can the delivery flows be sustained?

5) What is the maximum flow in steady state at D1 & D2 with the addition of the compressor?

Case Study 1A (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at D1 and D2?

Pressure at D1 = 466.377 psig

Deleted: 465.773

Pressure at D2 = 413.349 psig

Deleted: 412.582

2) With the compressor operating, can D2 take gas at the new higher rate without impacting D1?

Yes

3) Can minimum gas delivery conditions be maintained at D1 and D2 if the compressor is shut off for four hours?

Yes

4) The compressor has gone on annual operational maintenance and it will restore normal operations only after 8 hours. Can the delivery flows be sustained?

No, the pipeline is able to maintain the flow until time (8:45:13) before minimum pressure is reached and the delivery flow drops

Deleted: 8.690 hours

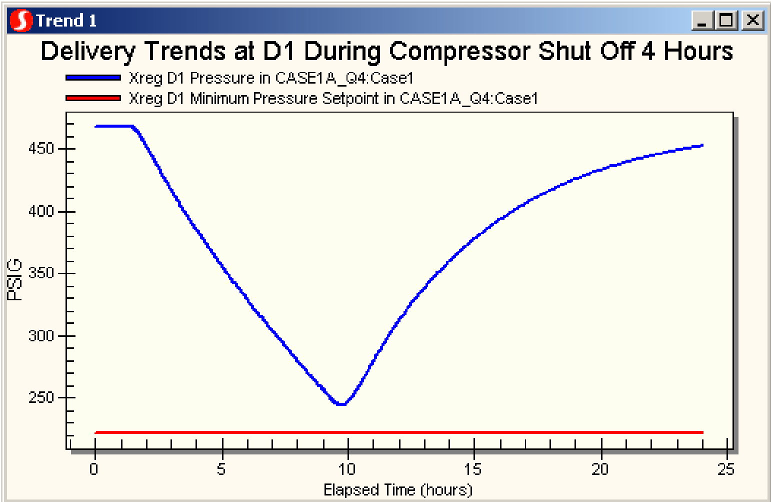
Deleted: 8:41:25

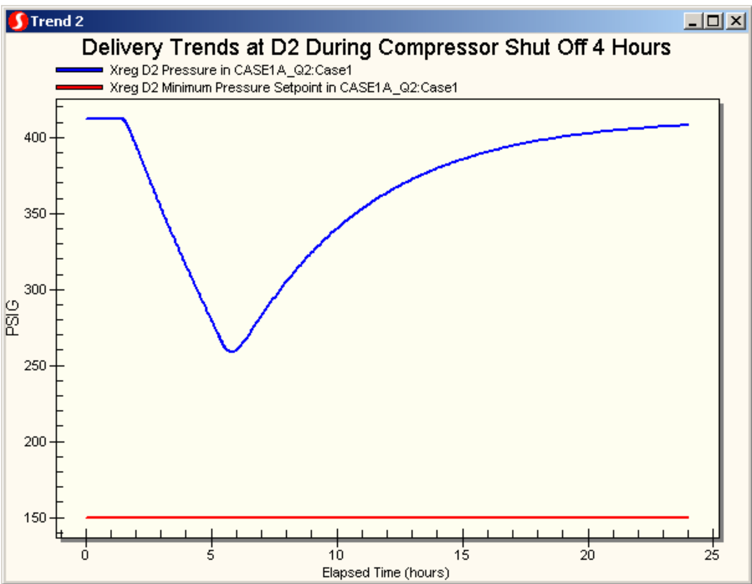
5) What is the maximum flow in steady state at D1 & D2 with the addition of the compressor?

Flow rate at D1 - 34.853 mmscfd Flow rate at D2 - 147.677 mmscfd Total Flow = 182.530 mmscfd

Case Study 1A (continued)

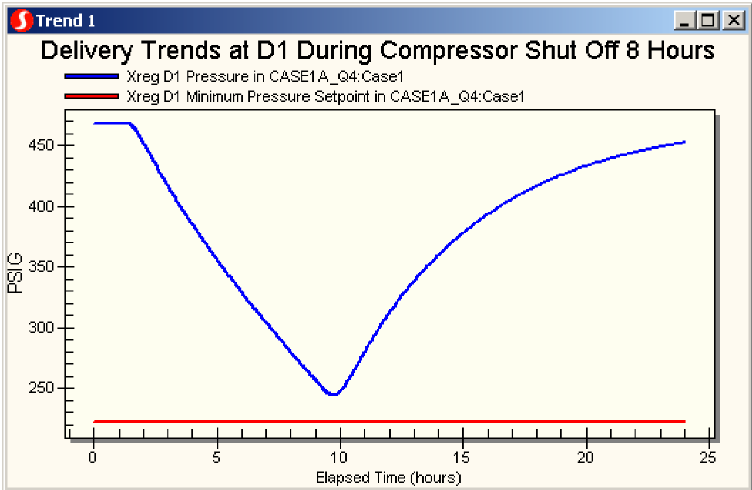
When the compressor is tripped off for 4 hours

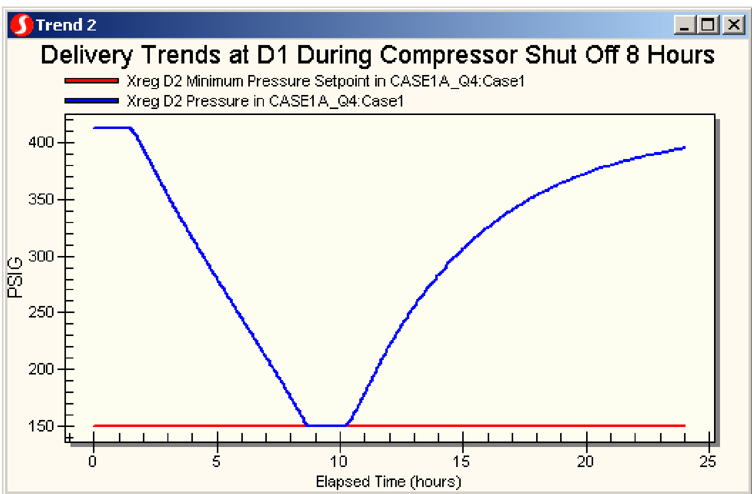




Case Study 1A (continued)

When the compressor is down for 8 hours due to annual operational maintenance





Case Study 1B (Generic Driver)

Purpose

The simple network and compressor created in case1A will be augmented to include a Generic Driver.

Important Elements

Drivers

Drivers are optional devices that provide power to a compressor unit. Studies that require the user to determine compressor fuel gas consumption or for determining the horsepower requirements of a compressor require the use of a driver.

Generic Drivers

A generic driver permits the inclusion of driver limitations and demands on the simulations without the need for detailed particulars of the engines involved.

Fuel Consumption

Specifying a particular driver will allow the user to determine a fuel gas requirement. This volume of gas may be extracted from the pipeline. In this Case Study we will configure a driver and have its fuel gas extracted from the line.

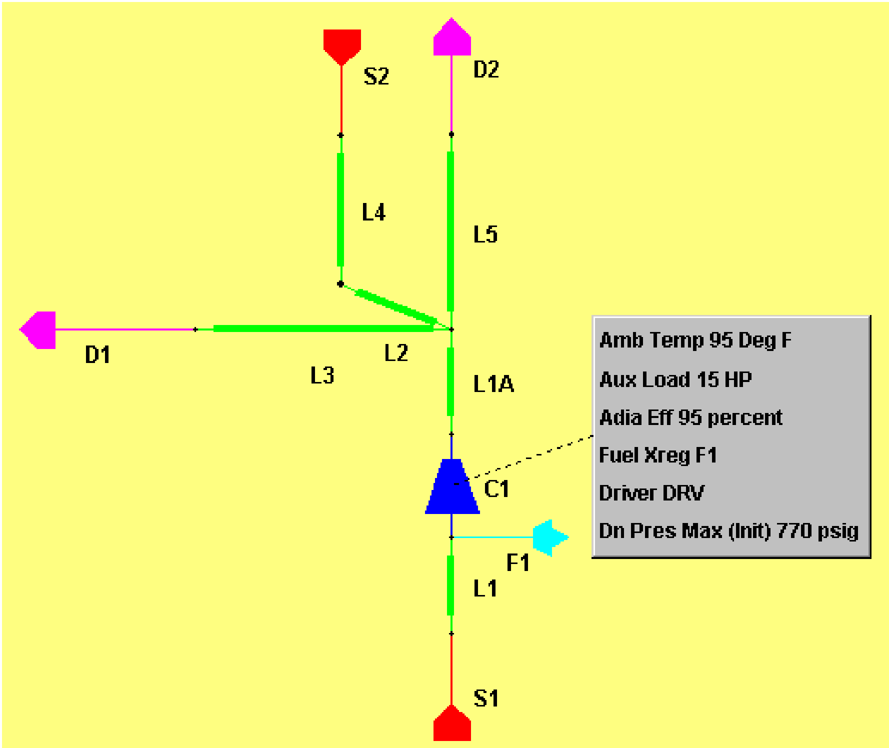
Equipment specifications

Equipment specifications for pipeline facilities such as compressors and drivers (including the associated performance curves) are input into the respective equipment detailed dialogue box.

The Scenario

After considering all the drivers that are available, a unit that is site rated for 1750 HP @ 60^{O}F has been selected. The driver performance coefficients were also provided (see fine print in the manufacturer's specifications - or supplied schematic). Does this unit provide adequate horsepower to allow you to still make your deliveries? Can the compressor be shut off for 4 hours?

Case Study 1B (continued)



Case1B

Driver Performance Data:
A = 1.0, B = 0.0, C = 0.0

Driver:	DRV
Ambient Rating Factor:	0.0005
Rated Temperature:	60 ^{o}F
Rated Fuel Efficiency:	30%
Rated Power:	1750 HP

INSTRUCTIONS

1. Create a new case based on the schematic shown and the information given
2. Add the DPID specification
3. Add a fuel external regulator to the configuration
4. Modify the compressor to reflect the DRIVER reference and the other new data
5. Select the appropriate trends to collect the data needed to answer the questions
6. Modify the model and make additional runs as required to answer the questions

Case Study 1B (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at D1 and D2

Pressure at D1= Pressure at D2=

2) Can we deliver the flow required at D1 and D2 if the compressor is shut off for 4 hours

3) How does this compare to the head for Case1A? Why is there a difference?

4) How much fuel does the compressor consume on an hourly basis during the first 24 hours?

5) What is the maximum flow in steady state at D1 and D2 with the compressor driver addition?

Case Study 1B (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at D1 and D2

Pressure at D1= 324.397 psig
Pressure at D2= 240.801 psig

Deleted: 322.027
Deleted: 237.439

2) Can we deliver the flow required at D1 and D2 if the compressor shut off for 4 hours

No

3) How does this compare to the head for Case1A? Why is there a difference?

It is lower. The compressor in Case 1A has unlimited driver power, whereas the compressor in Case1B has driver limitations

4) How much fuel does the compressor consume in steady state and during the first 24 hours?

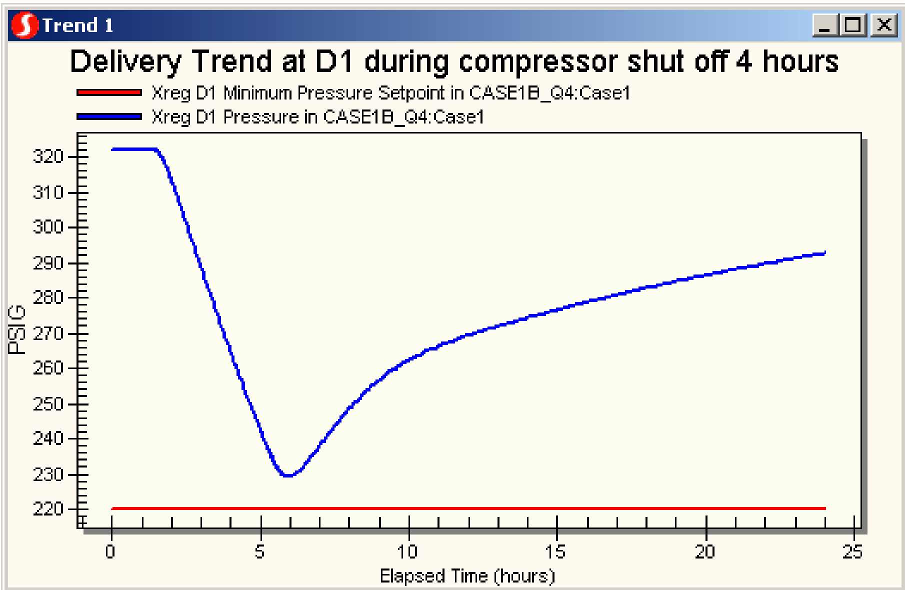
0.333 mmscfd and 0.277 mmscf

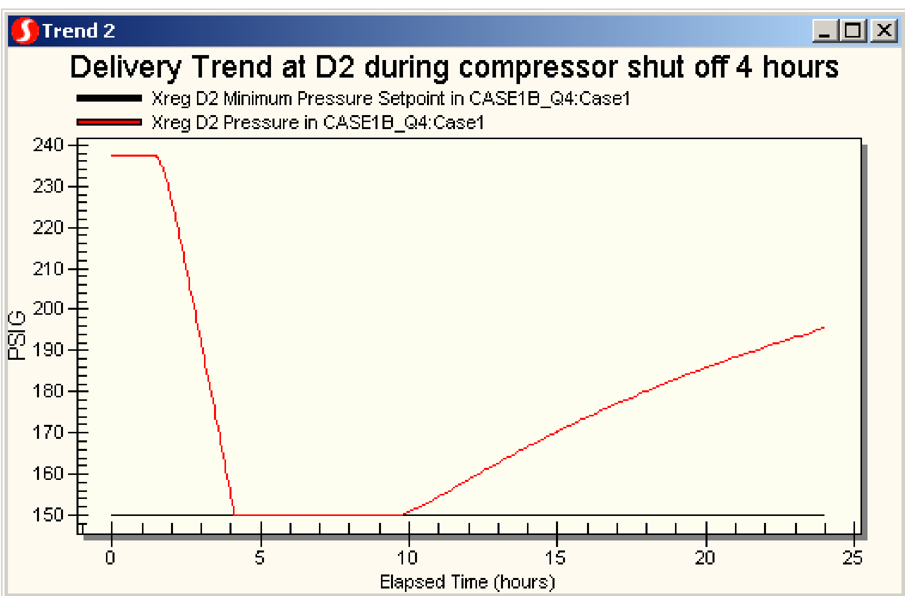
5) What is the maximum flow in steady state at D1 and D2 with the compressor driver addition?

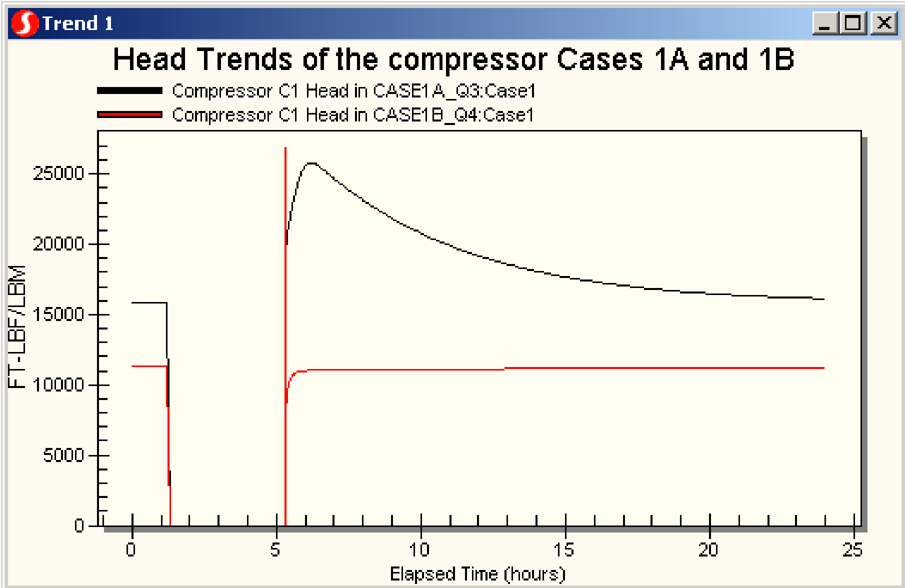
D1 = 30.511 mmscfd, D2 = 132.078 mmscfd
Total= 162.923 mmscfd

Deleted: 30.492
Deleted: 131.993
Deleted: 162.819

Case Study 1B (continued)







Case Study 1B (continued)

During peak demand where the demand exceeds the steady state capacity of the line, the ability of the system to respond is governed almost exclusively by the amount of gas stored in the pipeline as resident linepack. The primary difference between this case study and CASE1A is that the power limitation of the modified compressor has produced lower pressures in the downstream portion of the network resulting in less linepack. Consequently, there is inadequate linepack available to cover for the loss of the compressor.

Case Study 1C (Centrifugal Compressor)

Purpose

The compressor used in Case 1C will be assigned a fully defined centrifugal compressor curve.

Important Elements

Centrifugal Compressors

Centrifugal compressors possess a specified relationship between the flow rate of the gas passing through the unit and the amount of pressure or head that the unit can produce. Additionally, operational boundaries are placed on the unit to prevent the operation of the compressor under unreasonable, inefficient, or hazardous conditions. These limiting boundaries are the surge and stonewall lines.

Head Curve

The head versus flow relationship of the compressor is defined by specifying a collection of performance points. A curve fit is performed on these data points to create a CPID or Compressor Performance ID. At least six data points need to be provided and at least two speeds represented. Values outside of the surge and stonewall limits should be avoided.

Efficiency Curve

The adiabatic efficiency of the compressor is similarly specified. This efficiency term controls how much of the compressor's power is lost as heat into the gas stream. At least six data points need to be provided and at least two speeds represented.

Minimum and Maximum Speeds

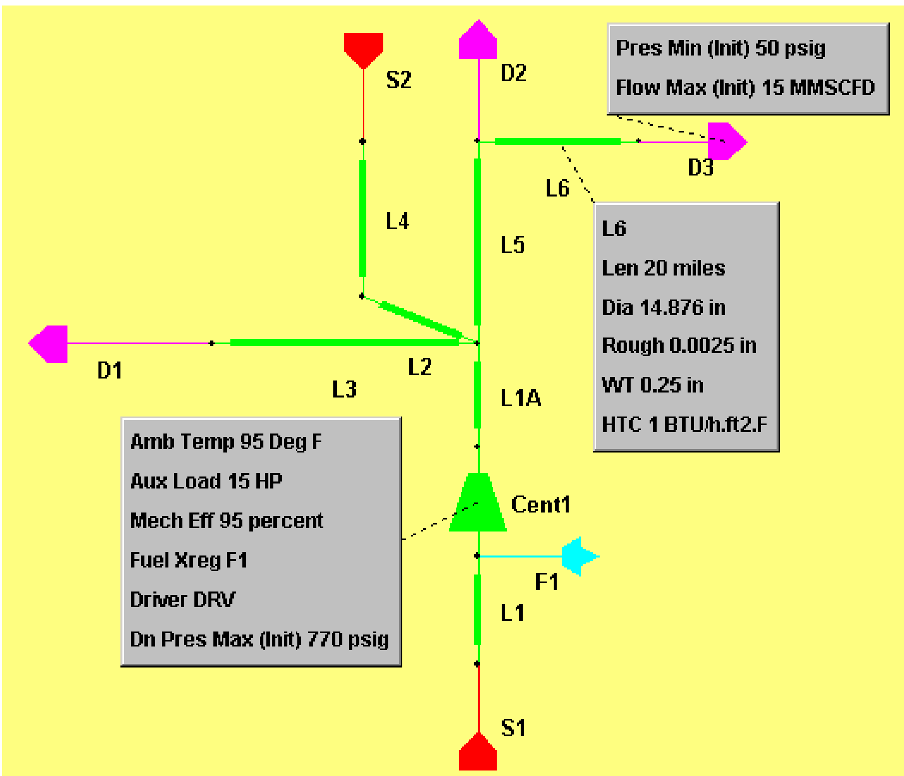
Compressor power is also a function of the unit speed. Each compressor will have a maximum and minimum speed under which the unit should operate (In terms of safety as well as pure mechanical limits).

Case Study 1C (continued)

The Scenario

Having a fairly good idea of power and throughput requirements for a compressor, you start shopping around for a potential vendor. You have finally selected a centrifugal compressor and also got the design data that is required.

A new delivery is likely to come up close to delivery D2.This new delivery will also put a strain on the system. Add the facilities shown and determine the limitations that this centrifugal compressor will impose on the system.



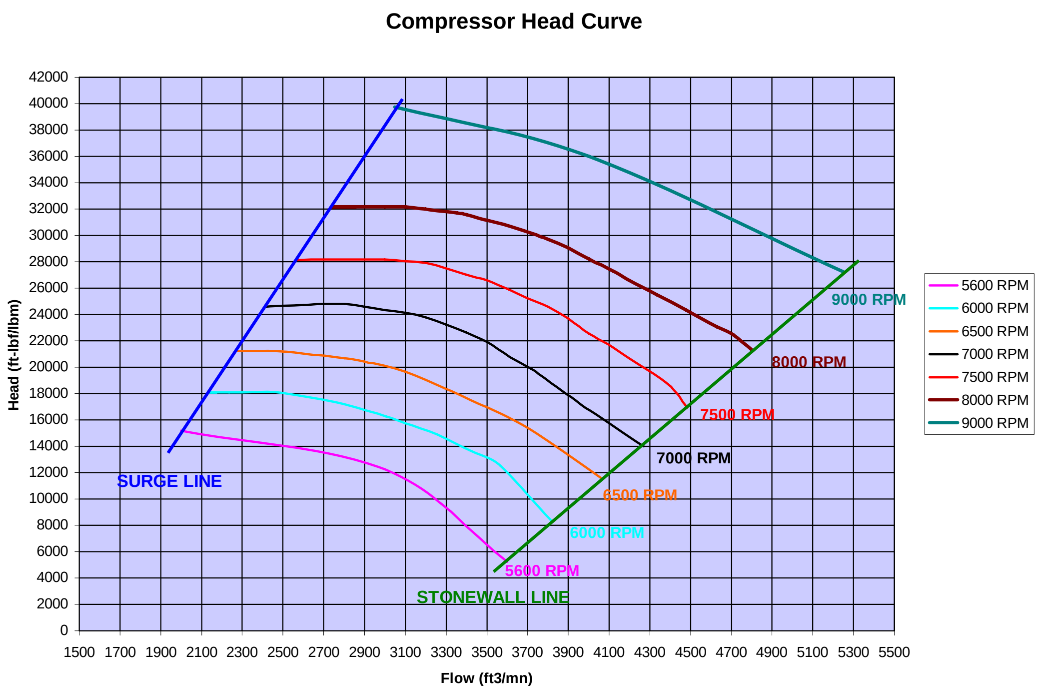
CASE1C

D3 flow= 15 mmscfd, minimum pressure = 50 psig
Driver Performance Data:
Driver: DRV; A = 1.0, B = 0.0, C = 0.0

Ambient Rating Facto: 0.0005
Rated Temperature: 60 °F
Rated Fuel Efficiency: 30%
Rated Power: 8000 HP

Case Study 1C (continued)
Additional Pipe Data:

Pipeline lateral connection - From D2 to the new delivery D3.
Pipe Name - L06
Length - 20 miles
ID - 14.876 inches
Roughness - 0.00250
WT = 0.25 inches
U = 1.0 BTU/ft·hr·°F
Delivery ("D3") Flow - 15 mmscfd
D3 - Minimum Pressure 50 psig



Case Study 1C (continued)

Extract six (minimum recommended) or eight data points from the curve.

	Head (ft·lbf/lbm)	Speed (rpm)	Flow (ft³/mn)
1	17780	6000	2000
2	11500	6000	3500
3	24490	7000	2500
4	16650	7000	4000
5	31420	8000	2500
6	22780	8000	4500
7	40250	9000	3000
8	30000	9000	5000

Deleted: Flow¶
(ft³/mn)

Deleted: Speed¶
(rpm)

Formatted Table

Efficiency data points are:

	Effic. (%)	Speed (rpm)	Flow (ft³/mn)
1	78.0	6000	2335
2	70.0	6000	3450
3	78.0	7000	2750
4	75.0	7000	3750
5	78.0	8000	3110
6	70.0	8000	4585
7	79.5	9000	4000
8	65.0	9000	5480

Deleted: Flow¶
(ft³/mn)

Deleted: Speed¶
(rpm)

Deleted: 2335

Deleted: 6000

Deleted: 3450

Deleted: 6000

Deleted: 2750

Deleted: 7000

Deleted: 3750

Deleted: 7000

Deleted: 3110

Deleted: 8000

Deleted: 4585

Deleted: 8000

Deleted: 4000

Deleted: 9000

Deleted: 5480

Deleted: 9000

Surge and Stonewall data points are:

	SPEED (RPM)	FLOW (FT³/MN)
SURGE	5600	2000
STONEWALL POINT	9000	5250

Case Study 1C (continued)

INSTRUCTIONS

1) Create a new case based on CASE1B
2) Create a CPID specification
3) Extract HEAD data points from the compressor curve provided and input the head/flow curve data
4) Using the EFFICIENCY data points provided, input the efficiency/flow curve data
5) Modify the compressor to reflect the new data and CPID reference. Specify a minimum and maximum speed of 4000 rpm and 9000 rpm
6) Add new lateral and delivery data
7) Perform a steady state simulation and answer the questions
8) Make sure to review any reported constraint violations to make sure they make sense

Case Study 4C (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the Steady State pressures at D1, D2 and D3?

Pressure at D1

Pressure at D2

Pressure at D3

2) What is the operating speed of the compressor?

3) Plot the compressor operating point on your performance curve. Is this a good point to operate at?

4) How much fuel does the compressor consume in the first day?

5) Try running the simulation with a D3 delivery of 20 mmscfd. What happens?

Case Study 1C (continued)

INSTRUCTOR'S SOLUTION

After entering and executing the model described above, answer the following questions:

1) What are the Steady State pressures at D1, D2 and D3?

Pressure at D1 – 379.679 psig

Deleted: 378.718

Pressure at D2 – 238.721 psig

Deleted: 236.903

Pressure at D3 – 231.33 psig

Deleted: 229.443

2) What is the operating speed of the compressor?

7037 RPM

Deleted: 7049

3) Plot the compressor operating point on your performance curve. Is this a good point to operate at?

Yes

4) How much fuel does the compressor consume in the first day?

0.95 mmscf

5) Try running the simulation with a D3 delivery of 20 mmscfd. What happens?

The delivery flow at D2 reduces to 131.168 mmscfd. So with the present system D3 cannot receive 20 mmscfd.

Deleted: 131.057

pipelinestudio®

GAS NETWORK TRAINING

MODULE : QUALITY TRACKING

energysolutions®

PipelineStudio Training: Module Quality Tracking I
The Module I
The User I
The Training I
Case Study 1 (Quality Tracking) 1
Case Study 1A (Quality Tracking) 6

PipelineStudio Training: Module Quality Tracking

The Module

The module contains a series of cases; starting with a case that will be used as starting point to generate additional cases.

The cases will illustrate the mixing or blending between different fluids properties in the network.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1 (Quality Tracking)

Purpose

To determine the composition of the gas that reaches the deliveries

Important Elements

Supply

Sets of 3 supplies are configured and the fluid at each of these supplies has a unique composition for the gas.

Delivery

The deliveries are configured at minimum pressures and maximum flow; one delivery (D2) has a restriction on the percentage CO_2 that it can accept

Deleted: 2

Tracking Options

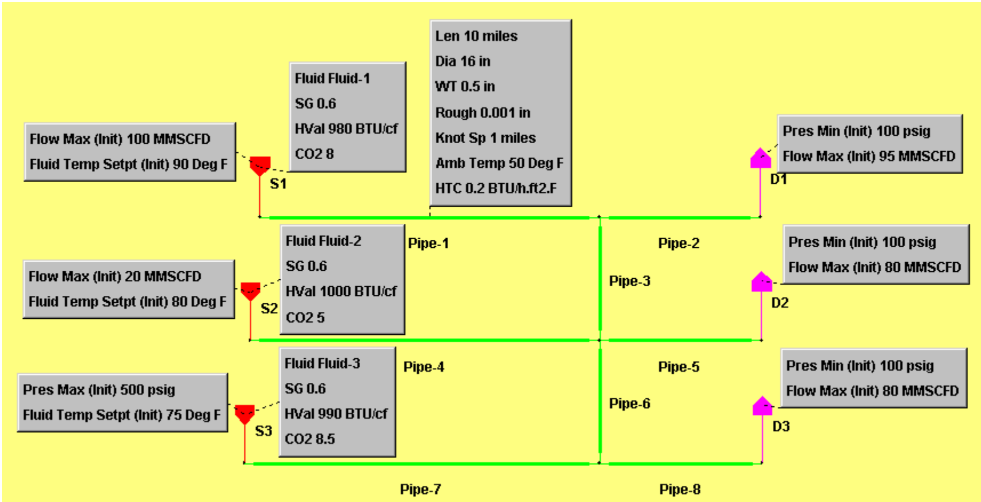
Quality tracking has to be enabled for this case to determine the composition of the gas that the deliveries receive

The Scenario

The system consists of 3 supplies. The fluid at each source has a unique composition. There are contract limitations because of which the CO_2 quantity that is delivered at the delivery D2 and has to be regulated. Each supply will have constraints with regard to the maximum pressure / flow. Steady state scenarios have to be run in order to determine the capacity in the network without violating the contract norms

Deleted: 2

Case 1 (Continued)



Case 1

Supply details

Supply	Max Flow (mmscfd)	Max Pressure (psig)
S1	100	
S2	20 (Can vary between 20 and 100). To start with it is 20	
S3		500

Fluid Properties

Supply	Fluid	Specific Gravity	Heating Value BTU/cf	CO_2 %
S1	Fluid-1	0.6	980	8
S2	Fluid-2	0.6	1000	5
S3	Fluid-3	0.6	990	8.5

Deleted: 2

Delivery details

Delivery	Max Flow (mmscfd)	Min Pressure (psig)
D1	95	100
D2	80	100
D3	80	100

Pipeline data (For all pipelines)

Length (miles)	Inside diameter (in)	Roughness (in)	Wall thickness (in)	Ambient Temperature (F)	Heat Transfer Coefficient (BTU/h.ft^2.F)
10	16	0.001	0.5	50	0.2

Formatted: Font: Arial Bold, Superscript

Instructions

1) Configure the model (Pipes, supplies, deliveries)
2) Turn on Temperature and Quality tracking from **Simulations**->**Options**->**General** tab
3) Run the model for steady state
4) Note the composition of the gas at the deliveries

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

1) What are the Steady State pressures at D1, D2 and D3?

Pressure at D1 =

Pressure at D2 =

Pressure at D3 =

2) What is the % of CO_2 at the deliveries D1, D2 and D3

Deleted: 2

% CO_2 at D1 =

Deleted: 2

% CO_2 at D2 =

Deleted: 2

% CO_2 at D3 =

Deleted: 2

3) There is a contractual condition to deliver gas to D2 with a % CO_2 content is equal or less than 6%. Make the adjustments in the supply "S2" to achieve this.

Deleted: 2

4) What are the flow in supplies S1, S2, S3 and the % of CO_2 at the deliveries D1, D2 and D3

Deleted: 2

S1 = mmscfd % CO_2 at D1 =

Deleted: 2

S2 = mmscfd % CO_2 at D2 =

Deleted: 2

S3 = mmscfd % CO_2 at D3 =

Deleted: 2

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

1) What are the Steady State pressures at D1, D2 and D3?

Pressure at D1 = 353.355 psig

Deleted: 353.11

Pressure at D2 = 369.549 psig

Deleted: 369.343

Pressure at D3 = 385.684 psig

Deleted: 385.504

2) What is the % of CO_2 at the deliveries D1, D2 and D3

Deleted: 2

% CO_2 at D1 = 8

Deleted: 2

% CO_2 at D2 = 8.09

Deleted: 2

Deleted: 7.59

% CO_2 at D3 = 8.5

Deleted: 2

3) There is a contractual condition to deliver gas to D2 with a % CO_2 content is equal or less than 6%. Make the adjustments in the supply "S2" to achieve this.

Formatted: Subscript

Flow in S2 = 57 mmscfd will be enough to have less than 6 %

4) What are the flow in supplies S1, S2, S3 and the % of CO_2 at the deliveries D1, D2 and D3

Deleted: 2

S1 = 100 mmscfd % CO_2 at D1 = 8

Deleted: 2

S2 = 57 mmscfd % CO_2 at D2 = 5.975

Deleted: 2

S3 = 98 mmscfd % CO_2 at D3 = 8.5

Deleted: 2

Case Study 1A (Quality Tracking)

Purpose

To determine the gas composition (H2S) that reaches at the delivery point

Important Elements

Supply

Sets of 3 supplies are configured and the fluid of these supplies has two different fluid gas composition.

Delivery

The delivery is configured using maximum flow; this delivery (D1) has a restriction on the percentage H_2S that it can accept. There is a contractual condition to deliver gas to D1 with a % H_2S content equal or less than 1%. The other three deliveries are configured just to verify the gas quality.

Deleted: HS2

Formatted: Subscript

Formatted: Subscript

Tracking Options

Temperature and quality tracking have to be enabled for this case to determine the composition of the gas that the deliveries receive

The Scenario

The system consists of 3 supplies and 4 deliveries three of those will be use as a check point to verify the fluid quality. The fluid at each source has two different compositions one fluid will contain the H_2S value after processing plant and one fluid will have higher value that represent the H_2S when the process plant is out of service. The scenario is about a failure in the process plant and the H_2S value has to be carefully monitor to determine the new quality and how long does it takes the new quality arrive to the delivery or terminal point.

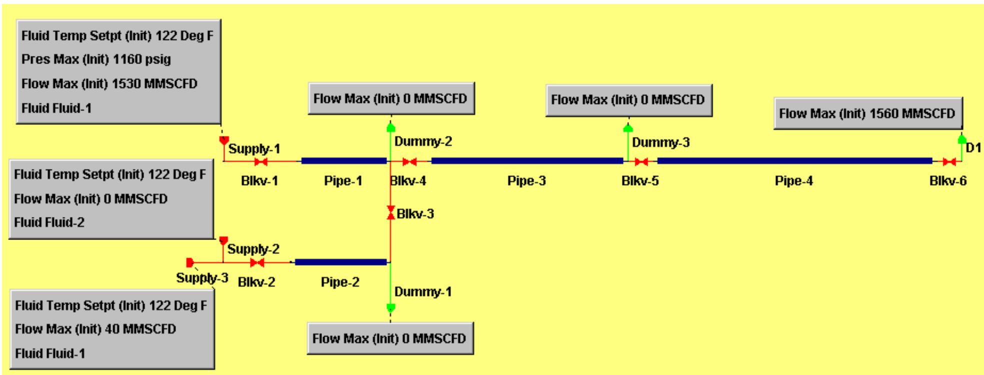
Deleted: HS2

Formatted: Subscript

Formatted: Subscript

Formatted: Subscript

Use the configuration layout as a reference to configure the network and use the configuration name: Case1A.



Use the following data to configure the network:

Fluid-1 properties:

Component	Percent
Methane (C1)	90.1
Ethane (C2)	3.3
Propane (C3)	1.3
Isobutane (IC4)	0.5
Isopentane (IC5)	0.5
Hexane (C6)	0.7
Heptane+ (C7+)	0.6
Carbon Dioxide (CO_2)	2.5
Hydrogen Sulfide (H_2S)	0.5

Formatted: Subscript

Formatted: Subscript

Fluid-2 properties:

Component	Percent
Methane (C1)	79.1
Ethane (C2)	7.3
Propane (C3)	3.3
Isobutane (IC4)	0.5
Isopentane (IC5)	0.5
Hexane (C6)	0.7
Heptane+ (C7+)	4.6
Carbon Dioxide (CO_2)	1
Hydrogen Sulfide (H_2S)	3

Formatted: Subscript

Formatted: Subscript

Supply-1 details:

Fluid	Fluid-1	
Maximum Flow	1530	MMSCFD
Maximum Pressure	1160	psig
Fluid Temperature	122	Deg F
Mode	Max Pressure	

Supply-2 details:

Fluid	Fluid-2	
Maximum Flow	0	MMSCFD
Maximum Pressure		psig
Fluid Temperature	122	Deg F
Mode	Max Flow	

Supply-3 details:

Fluid	Fluid-1	
Maximum Flow	40	MMSCFD
Maximum Pressure		psig
Fluid Temperature	122	Deg F
Mode	Max Flow	

Delivery **Dummy-1** details:

Maximum Flow	0	MMSCFD

Delivery **Dummy-2** details:

Maximum Flow	0	MMSCFD

Delivery **Dummy-3** details:

Maximum Flow	0	MMSCFD

Delivery **D1** details:

Maximum Flow	1560	MMSCFD

Valves details:

Valve Name	CV	Size in inches	Valve Name	CV	Size in inches
Blkv-1	50000	40	**Blkv-4**	50000	40
Blkv-2	50000	40	**Blkv-5**	50000	40
Blkv-3	50000	40	**Blkv-6**	50000	40

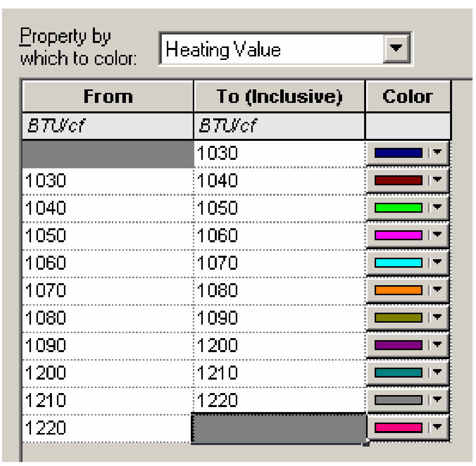
Pipes details:

Pipe Name	Length in Miles	Diameter (inside) in inch	Wall Thickness in inch	Roughness in inch	Knot Spacing in Miles	Efficiency
Pipe-1	1.5	39.37	0.314	0.001	1	1
Pipe-2	8	39.37	0.314	0.001	1	1
Pipe-3	60	39.37	0.314	0.001	1	1
Pipe-4	60	39.37	0.314	0.001	1	1

Pipes details:

Pipe Name	Ambient Temperature	Heat Transfer Coefficient
Pipe-1	104	0.2
Pipe-2	104	0.2
Pipe-3	104	0.2
Pipe-4	104	0.2

Color the Pipes by “Heating Value” according to the following table and colors:


Property by which to color: Heating Value

From	To (Inclusive)	Color
BTU/cf	BTU/cf	
	1030	
1030	1040	
1040	1050	
1050	1060	
1060	1070	
1070	1080	
1080	1090	
1090	1200	
1200	1210	
1210	1220	
1220		

Instructions

1) Configure the model (Pipes, supplies, fluids deliveries)
2) Turn on Temperature and Quality tracking from **Simulations**->**Options**->**General** tab
3) Select the Equation of State "BWRS" from **Simulations**->**Options**->**Fluid** tab
4) Color the pipes by "Heating Value" from **View**->**Network View Properties**->**Coloring** tab
5) Hide the coloring pipe "Legend" from **View**->**Legend**
6) Select "Show Thick Pipes" from **View**->**Show Thick Pipes**
7) Select the trends in delivery D1 (Heating Value and H2S)
8) Run the model for steady state
9) Select the pipes "Pipe-3" and Pipe-4" and open the Heating Value profile
10) Tile the windows horizontally from **Window**->**Tile Horizontally**
11) Start the Interactive Transient from **Simulations** ->**Interactive Transient**
12) Select the supply Supply-3 and ramp the flow value from 40 to 0
13) Select the supply Supply-2 and ramp the flow value from 0 to 300 (the equivalent of the process plant failure)
14) Press the "Start" button from the transient toolbar
15) Verify the new composition moving using the heating value pipes color and the profile
16) If it is necessary adjust the "Time Multiplier" to properly view the simulation
17) Press Pause after the new product passes the delivery Dummy-3 and mouse over to verify the new H2S composition.
18) Answer the questions from the next page

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

1) What is the % of H_2S at the delivery D1 in steady state ?

Formatted: Subscript

2) How long does it take the new product arrive to the delivery point D1 ?

3) What is the % of H_2S at the delivery D1 in transient scenario

Formatted: Subscript

4) Using the Interactive Transient repeat the following steps: 1) Select the supply Supply-3 and ramp the flow value from 40 to “0” 2) Select the supply Supply-2 and ramp the flow value from 0 to 560 (the equivalent of more process plant failure).

5) How long does it take the new product arrive to the delivery point D1 ?

6) What is the % of H_2S at the delivery D1 in transient scenario

Formatted: Subscript

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

INSTRUCTOR'S SOLUTION

1) What is the % of H_2S at the delivery D1 in steady state ?

Formatted: Subscript

0.5 %

2) How long does it take the new product arrive to the delivery point D1 ?

7.06 Hours (Around 7 hours)

3) What is the % of H_2S at the delivery D1 in transient scenario

Formatted: Subscript

0.98 %

4) Using the Interactive Transient repeat the following steps:
1) Select the supply Supply-3 and ramp the flow value from 40 to "0"
2) Select the supply Supply-2 and ramp the flow value from 0 to 560 (the equivalent of more process plant failure).

Formatted: Indent: Left: 18 pt

5) How long does it take the new product arrive to the delivery point D1 ?

Around 6 Hours

6) What is the % of H_2S at the delivery D1 in transient scenario

Formatted: Subscript

1.4 %

GAS NETWORK TRAINING

MODULE : EQUIPMENT

Deleted: EQUIPMENTS

PipelineStudio Training: Module Equipment I
The Module I
The User I
The Training I
Case Study 1 (Equipment) **1**
Case Study 1A (Equipment) **6**

Deleted: Equipments

Deleted: Equipments

Deleted: Equipments

PipelineStudio Training: Module Equipment

Deleted: Equipments

The Module

The module contains a series of cases that help to understand how the equipment could be used to improve the simulations results.

Deleted: equipments

The cases will illustrate the different equipment available in ***pipelinestudio***.

Deleted: equipments

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Deleted: Equipments

Case Study 1 (Equipment)

Purpose

To determine the functionality of various network elements that are available in the simulator. Each network element is added to a base network and its effect is studied.

Important Elements

Regulator

The simulator models various types of regulators found in actual pipeline networks. Specific regulator functions, listed below, are achieved by:

Flow control	Maximum flow
Pressure control	Maximum/minimum pressure differential
Back-pressure control	Minimum upstream pressure
Downstream-pressure control	Maximum downstream pressure constraints

Regulator valves automatically include a check valve to prevent back flow. If the regulator is not controlling flow or pressure, a free-flow constraint is automatically specified. The pressure-flow relationship across a fully opened valve is dependent upon a valve coefficient (CV).

Block valve

A block valve, as defined by the simulator, is a device that provides a variable resistance to flow based on the percent open value. A block valve is attached to other network elements or pipes by its upstream and downstream nodes.

When fully open, a block valve provides some amount of resistance to flow through the valve (which may be in either direction). When closed, the flow is assumed to be zero and the upstream and downstream flows are assumed to be independent of each other.

When placing fully-closed block valves into a pipeline network, you should be careful to avoid creating pressure-unspecified networks.

For block valves, there is only one mode of control: Percent Open.

Heaters

Heaters provide a means of increasing the temperature of a gas flow. The increase in temperature can be specified either as a fixed discharge temperature (Temperature Setpoint) or as a relative change (increase) in temperature (Max Delta Temperature).

If you set a Temperature Setpoint, the simulator will use it unless Max Delta Temperature is set and violated. If you do not set a Temperature Setpoint and Max Delta Temperature is set, the simulator will use the Max Delta Temperature. This prevents heaters from producing a downstream temperature lower than the upstream temperature.

Since a heater also produces hydraulic changes (flow and pressure), you can also specify a flowing resistance coefficient or a fixed discharge pressure setpoint.

A heater is connected to other items in the configuration by an upstream node and a downstream node.

Flow through a heater from the upstream node towards the downstream node is reported as positive; flow in the other direction is reported as negative.

Coolers

Coolers provide a means of reducing the temperature of a gas flow. The reduction in temperature can be specified either as a fixed discharge temperature (Temperature Setpoint) or as a relative change (drop) in temperature (Max Delta Temperature).

If you set a Temperature Setpoint, the simulator will use it unless Max Delta Temperature is set and violated. If you do not set a Temperature Setpoint and Max Delta Temperature is set, the simulator will use the Max Delta Temperature. This prevents coolers from producing a downstream temperature higher than the upstream temperature.

Since a cooler also produces hydraulic changes (flow and pressure), you can also specify a flowing resistance coefficient or a fixed discharge pressure setpoint.

A cooler is connected to other items in the configuration by an upstream node and a downstream node.

Flow through a cooler from the upstream node towards the downstream node is reported as positive; flow in the other direction is reported as negative.

Resistant Element

The simulator models only certain types of pipeline equipment devices and pipe. To model the hydraulic effects that result from other pipeline elements not supported by the simulator, a resistance element is provided that will generate a pressure drop as the flow through the device changes. A resistance element is not a controlled device, but will vary its pressure drop as the flow through the device changes.

Case Study 1

Scenario

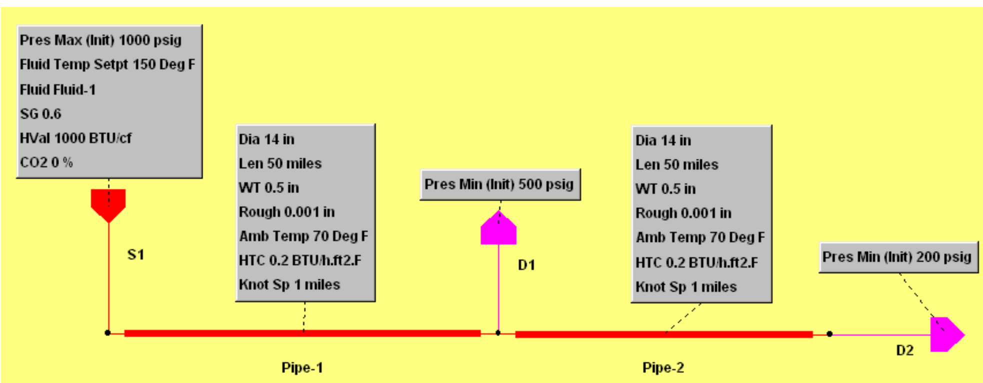
The system consists of one supply and two deliveries initially. The supply and delivery 1 are 50 miles apart and the delivery 1 and delivery 2 are another 50 miles apart. The pipeline and fluid details are as given below.

- Inside diameter = 14 in
- Length= 50 miles
- Wall thickness = 0.5 in
- Roughness= 0.001 in
- Fluid SG=0.6, Hval=1000 btu/cf CO_2=0%
- Gas temperature = 150F
- Ambient temperature = 70F

Formatted: Subscript

Supply and delivery constraints

- Supply S1 pressure = 1000 psig
- Delivery D1 Pressure (min) = 500 psig
- Delivery D2 Pressure (min)= 200 psig



Case1

Instructions

1) Configure the model (Pipes, supplies, deliveries)
2) Turn 'On' temperature tracking
3) Run the model for steady state
4) Determine the steady state flows at the supply and the deliveries

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

1) What is the maximum flow obtained in delivery D1 and delivery D2 ?

D1 =	mmscfd
D2 =	mmscfd

2) Delivery 2 needs to receive only 50 mmscfd of gas. How do you achieve this?

3) A new delivery D3 is to be located 50 miles from supply S1 (same node of delivery D1) wants to receive 40 mmscfd of gas at a pressure of 200 psig. Since there is a contractual obligation, pressure at this new delivery cannot exceed 200 psig. Propose a mechanism to achieve this

4) As per the existing contracts all deliveries have to receive gas at a temperature not less than 60°F. Find out the delivery temperatures after running the simulation and propose mechanisms in case the temperatures are less than 60°F

Deleted: 60F

Deleted: 60F

Delivery temperature D1 =	F
Delivery temperature D2 =	F
Delivery temperature D3 =	F

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

1) What is the maximum flow obtained in delivery D1 and delivery D2 ?

D1 = 60.38 mmscfd D2 = 70.90 mmscfd

Deleted: 60.3401
Deleted: 70.8527

2) Delivery 2 needs to receive only 50 mmscfd of gas. How do you achieve this?

A flow regulator is to be configured before D2 and its maximum flow must be set to 50 mmscfd

3) A new delivery D3 is to be located 50 miles from supply S1 (same node of delivery D1) wants to receive 40 mmscfd of gas at a pressure of 200 psig. Since there is a contractual obligation, pressure at this new delivery cannot exceed 200 psig. Propose a mechanism to achieve this

Configure a regulator and set its maximum downstream pressure to 200 psig

4) As per the existing contracts all deliveries have to receive gas at a temperature not less than 60°F. Find out the delivery temperatures after running the simulation and propose mechanisms in case the temperatures are less than 60°F

Deleted: 60F
Deleted: 60F

Delivery temperature D1 = 83.03 F Delivery temperature D2 = 58.09 F Delivery temperature D3 = 64.79 F Install a cooler at D1 and D3 and set its downstream temperature to 60°F Install heaters at D3 and set their downstream temperatures to 60°F

Deleted: 60F
Deleted: 60F

Case Study 1A (Equipment)

Deleted: Equipments

Purpose

The simple network will contain certain equipment that reproduce behaviors requires during the operation.

Deleted: equipments

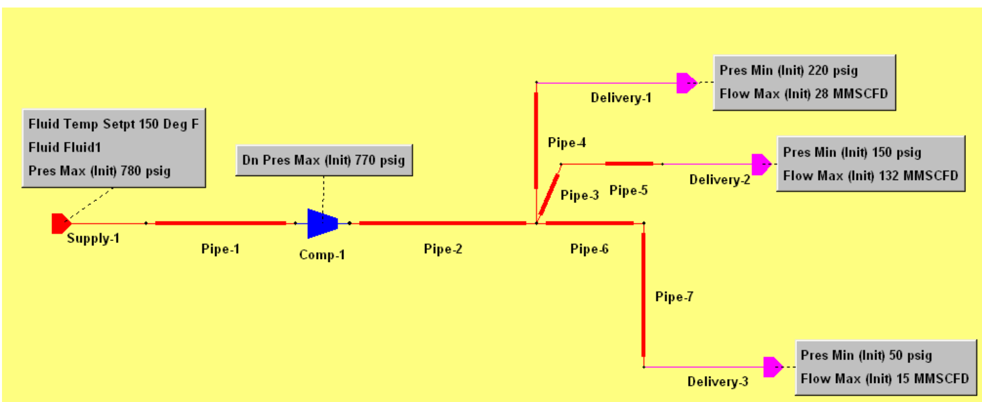
Important Elements

***pipeline**studio* Gas permits the inclusion of several equipment such as compressors, resistant elements, coolers, heaters and regulators that can properly simulate as close as possible a real pipeline network..

Deleted: equipments

The Scenario

A simple network will be configured and the basic network will be changing step by step adding a resistant element to reproduce a pressure drop upstream of the compressor, a cooler to modify the discharge temperature of the compressor, a heater to increase the temperature upstream of a delivery point and a regulator to reproduce a temperature drop by changing the downstream pressure control.



Case1A

Case Study 1A (continued)

Use the following data to configure the network:

Supply and Delivery Summary

XREG Name	Flow (MMSCFD	Pressure (PSIG)
Supply-1	?	780
Delivery-1	28	220
Delivery-2	132	150
Delivery-3	15	50

Pipeline Segment Data

Pipe Name	Length (Miles)	ID (Inches)	Roughness (Inches)
Pipe-1	18.3	15.5	0.0025
Pipe-2	17	14.876	0.0025
Pipe-3	12	16.876	0.0025
Pipe-4	30	10.02	0.00115
Pipe-5	17	16.876	0.0005
Pipe-6	29	17.35	0.0025
Pipe-7	20	14.876	0.0025

Pipe Name	Wall Thickness (inches)	Ambient Temperature (Deg F)	Heat Transfer Coefficient (BTU/h.ft^2.F)
Pipe-1	0.25	70	1
Pipe-2	0.25	70	1
Pipe-3	0.25	70	1
Pipe-4	0.25	66	1
Pipe-5	0.25	70	1
Pipe-6	0.25	70	1
Pipe-7	0.25	70.1	1

Formatted: Font: Arial Bold, Superscript

Gas Properties:

Fluid Name	SG	HV (BTU/CF)	CO_2 (%)
Fluid1	0.615	1050	1.4

Generic Compressor Comp-1 details:

Adiabatic Efficiency	Max Down Pressure
100 %	770 psig

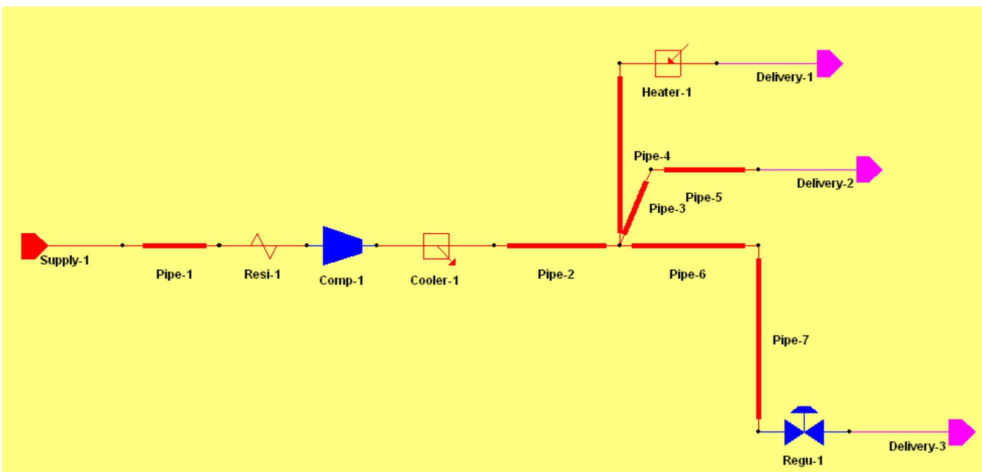
Use the following additional data:

Gas Temperature = 150 ^{o}F
Knot spacing = 1.0 miles

Case Study 1A (continued)

INSTRUCTIONS

1) Create a new case based on the schematic shown and the information given for CASE1A
2) Turn "On" the temperature tracking
3) Run steady state and record the details indicate in the observations and questions (see next page)
4) Add one by one each network element to create the new case name CASE1B
5) Make the proper adjustment for each network element and run steady state to verify the appropriate results and answer the questions (see next page)
6) Follow the instructions and answer the questions



Case1B

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at suction of the compressor Comp-1, and deliveries Delivery-1, Delivery-2 and Delivery-3?

Suction at compressor Comp-1= Delivery-1 = Delivery-2 = Delivery-3 =

2) What are the steady state temperatures at discharge of the compressor Comp-1, and deliveries Delivery-1, Delivery-2 and Delivery-3?

Discharge at compressor Comp-1= Delivery-1 = Delivery-2 = Delivery-3 =

3) Save the CASE1A as CASE1B and add the following:

A) Resistant Element upstream of the compressor Comp-1 to generate 10 psig pressure drop

B) Cooler downstream of the compressor Comp-1 to generate 10 Deg F temperature drop

C) Heater upstream of the delivery Delivery-1 to increase the temperature 10 Deg F

D) Regulator upstream of the delivery Delivery-3 controlling the downstream pressure that generates 10 Deg F temperature drop. Start controlling downstream 200 psig

Case Study 1A (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

After entering and executing the model described above, answer the following questions:

1) What are the steady state pressures at suction of the compressor Comp-1, and deliveries Delivery-1, Delivery-2 and Delivery-3?

Suction at compressor Comp-1= 513.954 psig
Delivery-1 = 299.367 psig
Delivery-2 = 201.39 psig
Delivery-3 = 446.963 psig

Deleted: 513.475
Deleted: 298.061
Deleted: 199.263
Deleted: 446.272

2) What are the steady state temperatures at discharge of the compressor C1, and deliveries Delivery-1, Delivery-2 and Delivery-3?

Discharge at compressor Comp-1= 126.74 Deg F
Delivery-1 = 65.3249 Deg F
Delivery-2 = 68.93 Deg F
Delivery-3 = 70.0895 Deg F

Deleted: 126.851
Deleted: 68.9912

3) Save the CASE1A as CASE1B and add the following:

A) Resistant Element upstream of the compressor Comp-1 to generate 10 psig pressure drop

Resistance Coefficient = 0.34 psi^2/(MMSCFD)^2

Formatted: Superscript
Formatted: Superscript

B) Cooler downstream of the compressor Comp-1 to generate 10 Deg F temperature drop

Mode Resistance Coefficient = 0.1 psi^2/(MMSCFD)^2 and Max Down Temp = 120 Deg F

C) Heater upstream of the delivery Delivery-1 to increase the temperature 10 Deg F

Mode Resistance Coefficient = 0.1 psi^2/(MMSCFD)^2 and Max Down Temp = 75 Deg F

D) Regulator (CV = 1000, size 12 inches) upstream of the delivery Delivery-3 controlling the downstream pressure that generates 10 Deg F temperature drop. Start controlling downstream pressure = 200 psig

Downstream Pressure = 300 psig

GAS NETWORK TRAINING

MODULE : LEAK

PipelineStudio Training: Module Leak I
The Module I
The User I
The Training I
Case Study 1 1

PipelineStudio Training: Module Leak

The Module

The case will illustrate the usage of a Leak Delivery that can help to estimate the amount of product that will exit the pipeline under certain conditions.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

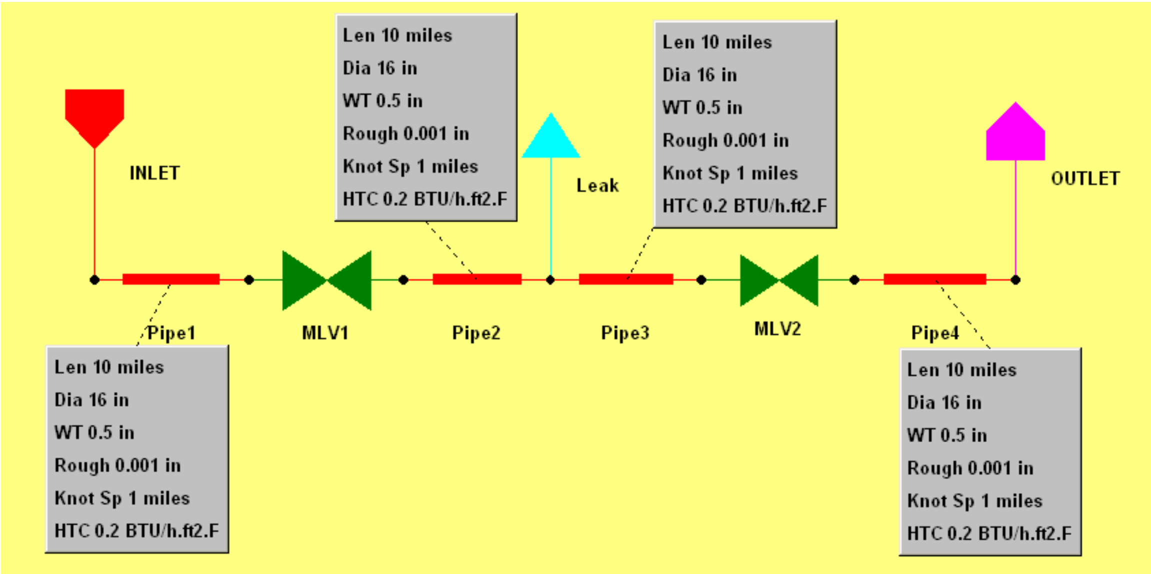
pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs or modules to allow the user to build on skills obtained from the previous module or lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1

Purpose

One of the more interesting pipeline transients possible is a leak. Learning how to recognize and efficiently handle a pipeline leak is an important part of controller operation although it does not happen very often.



Case 1

Pipeline & Simulation Details:

- Pipes 1 - 4 have an ID = 16 inch, Length = 10 miles
- Inlet - Max Pressure = 1000 psig
- Block Valves have ID=16 inch, Cv = 20000
- Fluid = 0.6 SG, 1000 BTU/ft^3, 0% CO_2, Temp. = 100 ^oF
- Outlet Flow = 200 mmscfd
- Leak Ambient Pressure = 14.696 psia
- Leak begins at time 3 hours and opens to a diameter of 2 inches after 0.1 hours
- Temperature tracking on
- Select the leak calculation method "Implicit"
- All other program defaults assumed

INSTRUCTIONS

1) Build a case called CASE1 with the above information

2) Enable the simulator volume accumulator and trend the necessary data to answer the following questions (see next page)

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

1. What is the peak rate of gas exiting from the leak?

2. How much gas is lost after 7 hours (see the hour 10)?

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

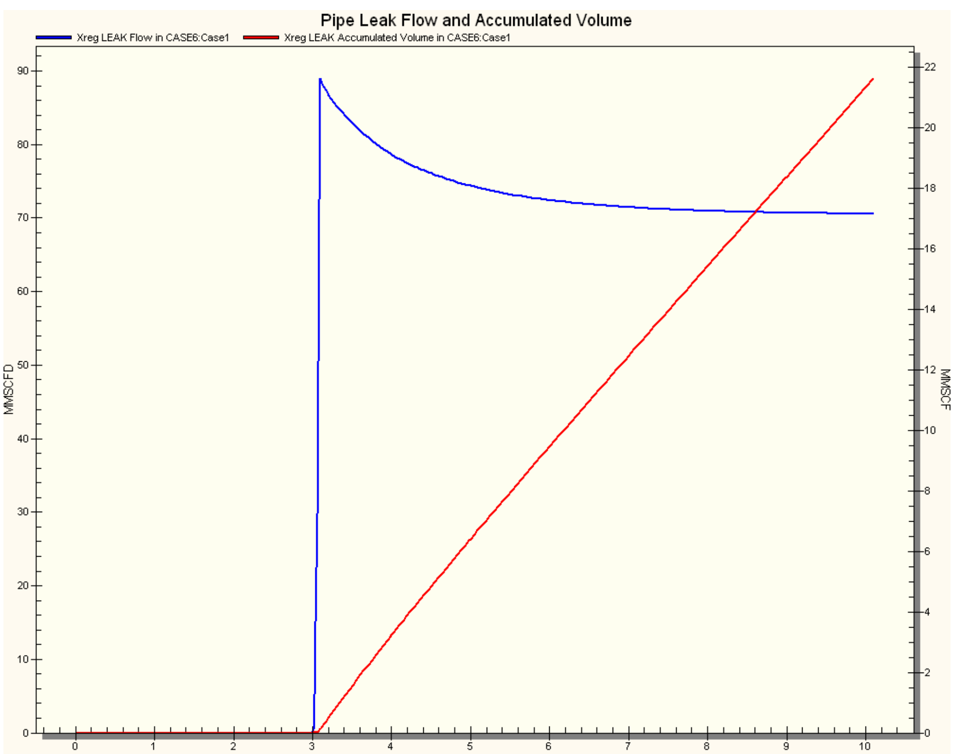
After entering and executing the model described above, answer the following questions:

1. What is the peak rate of gas exiting from the leak?

89.0 mmscfd

2. How much gas is lost after 7 hours (see the hour 10)?

21.61 mmscf



pipelinestudio®

GAS NETWORK TRAINING

MODULE : BLOWDOWN

energysolutions®

PipelineStudio Training: Module Blowdown I
The Module I
The User I
The Training I
Case Study 1 1
Blow Down 4
Case1A Blow Down 5
Start Up 10
Case1B Start Up 11

PipelineStudio Training: Module Blowdown

The Module

The module contains a series of cases; starting with a basic case that will be used to generate additional cases.

The cases will illustrate the usage of a Leak Delivery that can help to estimate the amount of product that will exit the pipeline under certain conditions and determine how long does it take to empty a pipeline.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

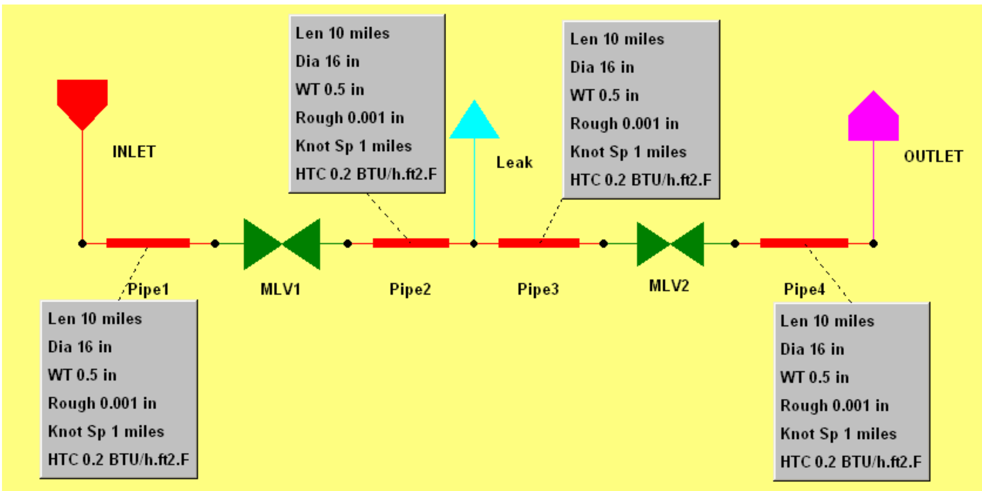
pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

Case Study 1

Purpose

One of the more interesting pipeline transients possible is a leak. Learning how to recognize and efficiently handle a pipeline leak is an important part of controller operation although it does not happen very often.



Case 1

Pipeline & Simulation Details:

- Pipes 1 - 4 have an ID = 16 inch, Length = 10 miles
- Inlet - Max Pressure = 1000 psig
- Block Valves have ID=16 inch, Cv = 20000
- Fluid = 0.6 SG, 1000 BTU/ft^3, 0% CO_2, Temp. = 100 ^oF
- Outlet Flow = 200 mmscfd
- Leak Ambient Pressure = 14.696 psia
- Leak begins at time 3 hours and opens to a diameter of 2 inches after 0.1 hours
- Temperature tracking on
- All other program defaults assumed

INSTRUCTIONS

1) Build a case called CASE1 with the above information

2) Enable the simulator volume accumulator and trend the necessary data to answer the following questions (see next page)

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

1. What is the peak rate of gas exiting from the leak?

2. How much gas is lost after 10 hours?

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

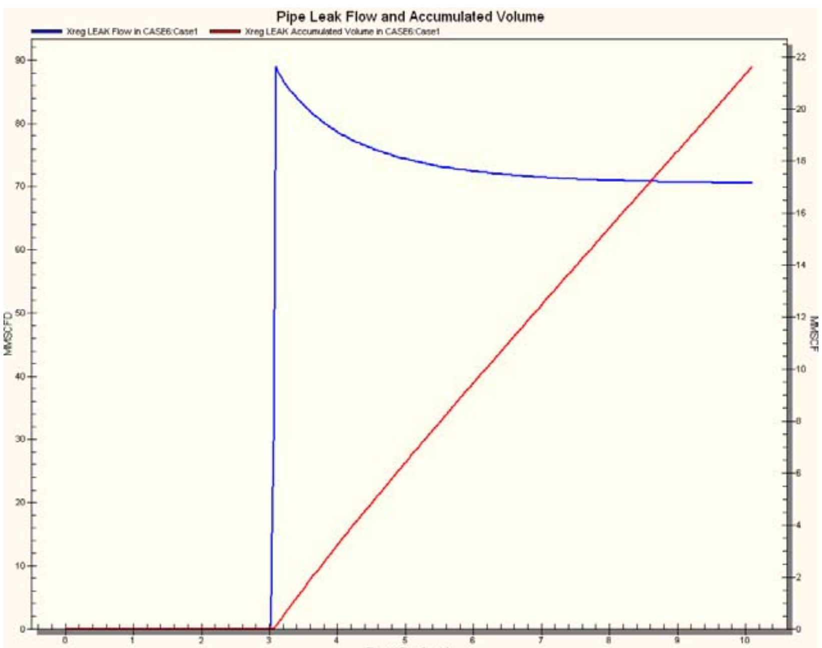
After entering and executing the model described above, answer the following questions:

1. What is the peak rate of gas exiting from the leak?

89.0 mmscfd

2. How much gas is lost after 10 hours?

21.34 mmscf



Blow Down

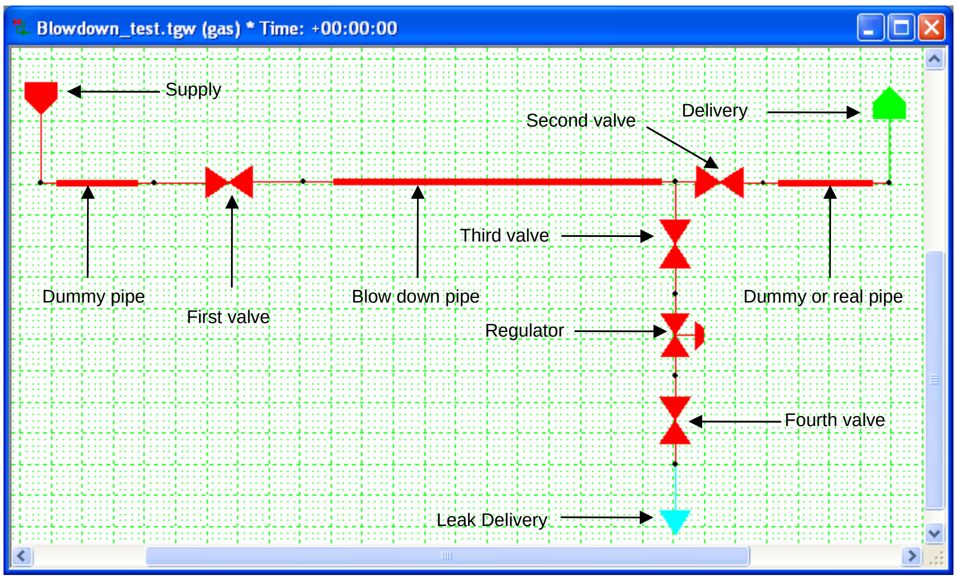
Purpose

Blow down is a very common activity during maintenance activities in the field, the first question that needs to be answer to do this type of operational planning is: How long does it take to empty the pipeline or section?

The question does not have an easy answer but Pipeline Studio permits the simulation of the scenario in order to have an idea or reference about the time require for that task.

It is important to mention that the deviation between the calculated values and field data has to be analysed to determine the variations and with special consideration of the gas quality (wet gas), initials pressures and the pressure control used to blow down the pipe or section.

To configure a section to blow down see the example:



This simple blow down network requires:
Supply: Required to reproduce the operational pressure coming from a compressor, regulator or any other source.
Delivery: The delivery is set to a specific flow to obtain the pressure drop expected in the blow down section or pipe in steady state conditions.
Pipe : The first pipe from left to right is a dummy section that connects the supply and the first valve. The second pipe is the blow down pipe. The third pipe could be a real pipe or dummy section to connect the delivery.
Valve: The first and second valve is used to isolate the blow down pipe from the supply and the delivery. The third and fourth valves connect the blow down pipe with the regulator and the leak delivery.
Regulator: The regulator is set with an estimate of the pressure value that is expected during the blow down.

Leak Delivery: The leak delivery represents the diameter available in the field that is used to blow down the pipe.

Once the blow down network is configured it is necessary to run steady state and made the adjustment by tuning as necessary.

The next step is to create the transient scenario, thus:

Blowdown_test.tgw (gas):2 * Time: +00:00:00

	Device Type	Name	Setpoint	Units	Initial..	1	1.1	5	10	15
1	Block Valve	V2	% open	percent	100	100	0	0	0	0
2	Leak Delivery	To_Atmosphere_2	Diameter	in	0	0	8	8	8	8
3	Delivery	OUT	Flow Maximum	MMSCFD	2	2	0	0	0	0
4	Block Valve	V1	% open	percent	100	100	0	0	0	0

The main purpose of using the transient scenario is to close valves at certain times to isolate the blow down pipe or section, change the flow rate to the delivery and to open the leak delivery to the diameter based on the field data.
The Pipeline Studio reports and graphical capabilities permit reviewing of the simulation results.

Case1A Blow Down

This case study is designed to determine the amount of time requires to blow down a pipe. The information obtained could be useful for maintenance or operational planning.
Use the following data to configure the network:

Fluid1 properties:

Specific Gravity	0.6	
Heating Value	1000	BTU/cf
Carbon Dioxide	0	percent

Supply **IN** details:

Fluid Temperature	80	Deg F
Maximum Pressure	765	psig

Delivery **OUT** details:

Maximum Flow	2	MMSCFD
Minimum Pressure	200	psig

Pipes details:

Pipe Name	Length in Miles	Diameter (inside) in inch	Wall Thickness in inch	Roughness in inch	Knot Spacing in Miles	Efficiency
Pipe_1	0.621	46.75	0.5	0.001	0.621	0.95
Blowdown_pipe	15.715	46.75	0.5	0.001	1	0.95
Pipe_2	6.213	46.75	0.5	0.001	0.6213	0.95

Additional details:

Ambient Temperature: 50 deg F
Heat Transfer Coefficient; 0.2 BTU/h.ft2.F

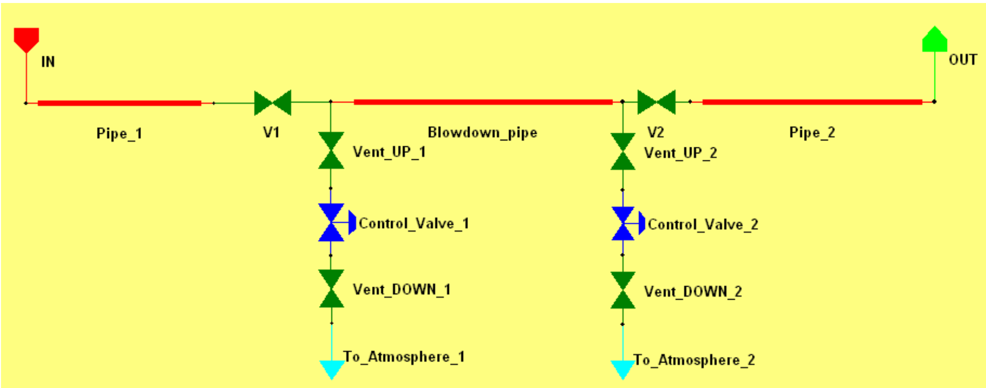
Valves details:

Valve Name	CV	Size in inches
V1	65000	48
V2	65000	48
Vent_UP_2	50000	8
Vent_DOWN_2	50000	8
Vent_UP_1	50000	8
Vent_DOWN_1	50000	8

Regulators details:

Regulator Name	CV	Size in inches	Downstream Press Max in Psig	Mode	Mode
Control_Valve_1	300	8	100	Max Down Pressure	Lock
Control_Valve_2	300	8	100	Max Down Pressure	Lock

Use the configuration layout as a reference to configure the network and use the configuration name: “Case1A”.



The following table shows the transient scenario:

Time (hours)	**Block Valve** V1 (%)	**Leak Delivery** To_Atmosphere_1 (inches)	**Leak Delivery** To_Atmosphere_2 (inches	**Delivery** OUT (MMSCFD)	**Block Valve** V2 (%)
1	100	0	0	2	100
1.1	0	8	8	0	0
2	0	8	8	0	0
3	0	8	8	0	0
4	0	8	8	0	0
5	0	8	8	0	0
6	0	8	8	0	0
7	0	8	8	0	0
8	0	8	8	0	0
9	0	8	8	0	0
10	0	8	8	0	0

Answer the questions (see next page)

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

1. Run the transient scenario to determine the blowdown time and establish the amount of hours requires to obtain the blowdown results.

2. Use Pipeline Studio graphical capabilities to show the results

Case Study 1A (continued)

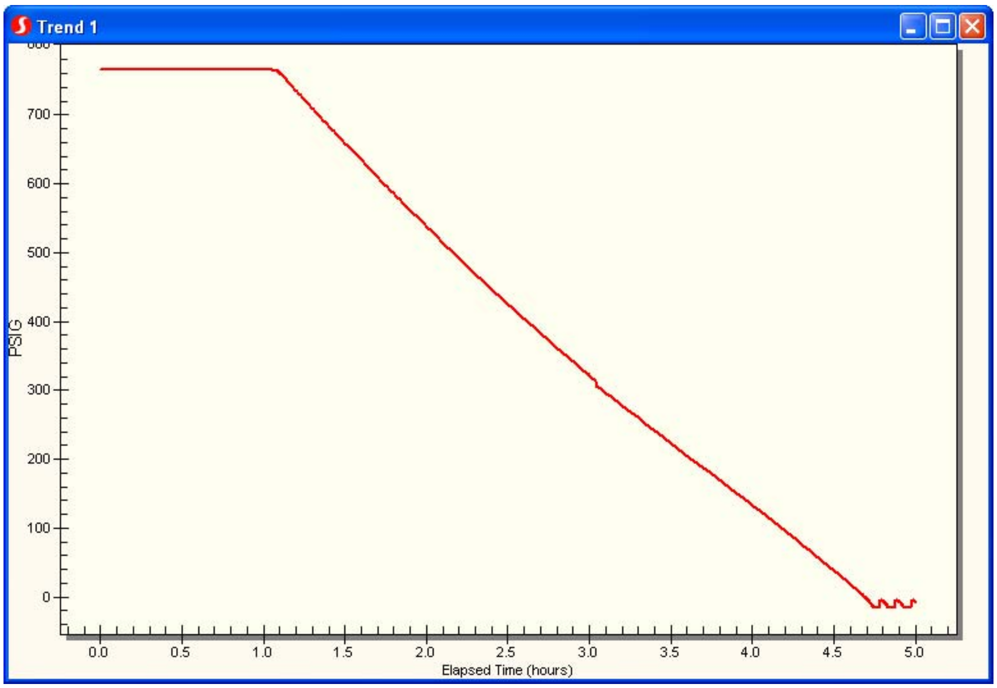
INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

1. Run the transient scenario to determine the blowdown time and establish the amount of hours requires to obtain the blowdown results.

5 hours will be enough the blowdown time is 4.68 hours

2. Use Pipeline Studio graphical capabilities to show the results



Start Up

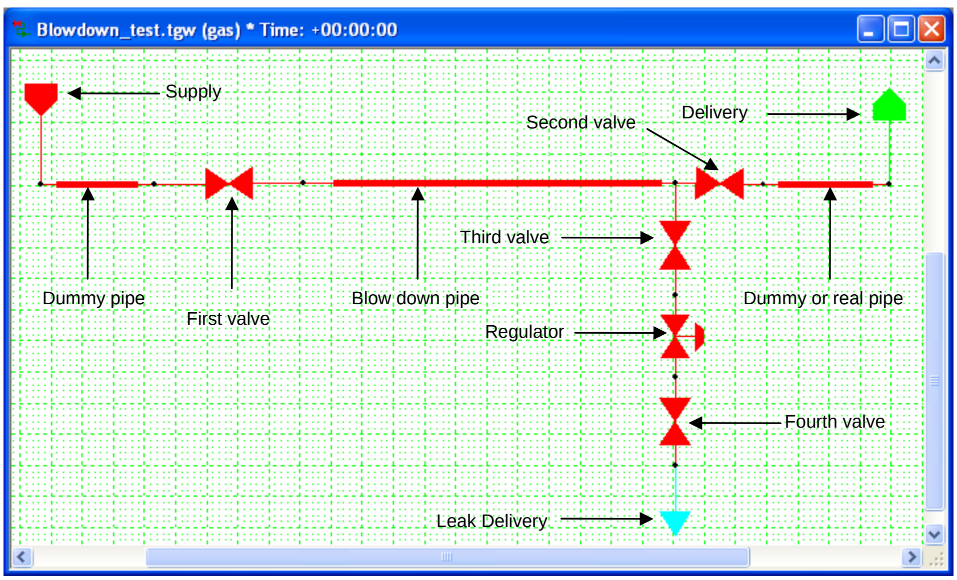
Purpose

It is a common practice to follow safety procedures. Examples are when a pipeline section has been replaced or a new section will be added to the existing network.
Before operation of a new pipeline section it is important to purge first the whole section, to avoid the risk of having the combination of air + natural gas.
Safety procedures establish the pressure value needed to purge the section and for how long to maintain the purge process to ensure that the air is not present inside the pipeline.
A practical rule may be used to calculate the reference pressure from the safety procedure to calculate the linepack or inventory of the section by assuming that the pressure at the beginning of the section is equal to the pressure at the end of the section.
Once the linepack is estimated the safety procedure indicates that the volume required to purge the pipeline of air is 1.5 times the inventory calculated.
The main question is to determine or calculate for how long the purge process needs to be maintained to ensure or avoid the risk of having the combination of air + natural gas knowing the parameters of the procedures and the physical data of the pipeline section.

Note: Pipeline Companies have their own references or criteria to purge pipeline sections; the intention of this training is to demonstrate that applying certain criteria is possible to reproduce the purge and start up process.

Pipeline Studio allows creating a configuration that could reproduce the purge and start up process.

Assuming that the same section was previously blow down now requires the start – up process
The configuration layout will looks as follow:



The configuration contains the devices needed to reproduce start-up of the pipeline or section process.
The procedure to calculate the purge time may be as follows:

- Using the pressures in the pipeline, establishing the initial conditions required for the purge process. Run steady state to obtain the information.
- Multiply the calculated linepack by the safety factor, for example 1.5, to obtain the purge volume
- Create the transient scenario and run the configuration to obtain the time requires for the purge process
- Use Pipeline Studio reports and graphical capabilities to obtain the calculated values.
- Once the purge process is completed, the next step is to increase the pressure to reach the operational level.

The ideal case is to calculate the linepack or inventory using the operational pressure level, modify the transient scenario and use the operational pressure, run the transient scenario and verify how long it will take to reach the nearest inventory value previously obtained.

Case1B Start Up

This case study is designed to estimate the amount of time requires to start up a pipe after a blow down procedure. The information obtained could be useful for maintenance or operational planning.
Use the following data to configure the network:

Fluid1 properties:

Specific Gravity	0.6	
Heating Value	1000	BTU/cf
Carbon Dioxide	0	percent

Supply **IN** details:

Fluid Temperature	80	Deg F
Maximum Pressure	50	psig

Leak Delivery **To_Atmosphere_2** details:

Diameter	0	Inches
Ambient Pressure	14.696	Psia
Coefficient	1	

Valves details:

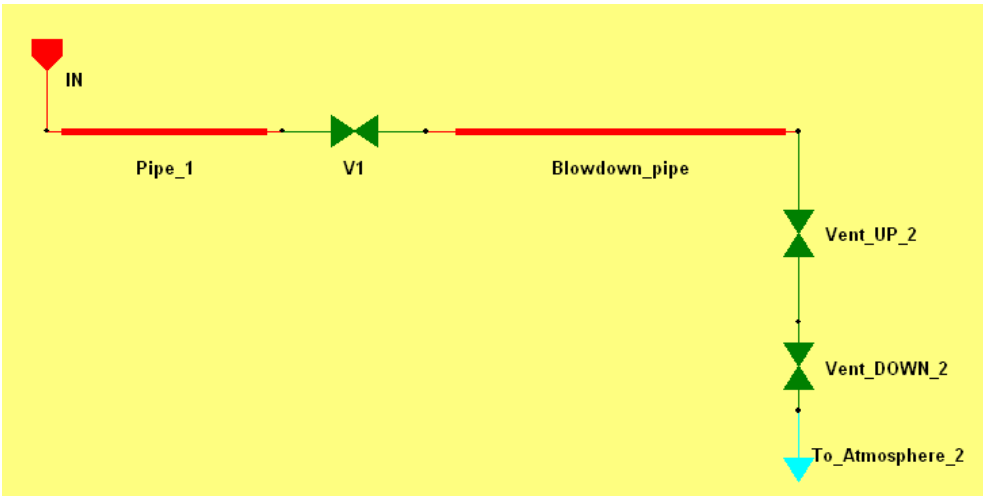
Valve Name	CV	Size in inches
V1	65000	48
Vent_UP_2	50000	8
Vent_DOWN_2	50000	8

Case Study 1B (continued)

Pipes details:

Pipe Name	Length in Miles	Diameter (inside) in inch	Wall Thickness in inch	Roughness in inch	Knot Spacing in Miles	Efficiency
Pipe_1	0.621	46.75	0.5	0.001	0.621	0.95
Blowdown_pipe	15.715	46.75	0.5	0.001	1	0.95

Use the configuration layout as a reference to configure the network and use the configuration name: "CASE1B".



Answer the questions (see next page)

Case Study 1B (continued)

OBSERVATIONS AND QUESTIONS

Use Pipeline Studio graphical capabilities to support the answers

1. Run steady state using 50 psig and obtain the inventory based on that current pressure level.

2. Obtain 1.5 times the inventory value.

3. Create a transient scenario to purge from the pipe 1.5 times the inventory value obtained after run steady state. The time obtained will be the amount of time required to purge the pipe.

Note: To build the transient scenario use the diameter 8 inches for leak delivery also let the transient run the first hour using steady state or initial values.

4. Run steady state using 765 psig and obtain the inventory based on that current pressure level.

5. Assuming that the 75 % of the inventory value obtained in steady state is the minimum required before to operate the pipeline; modify the transient scenario to determine how long does it take to reach the closest value to the 75 % of the steady state inventory value?

Case Study 1B (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

Use Pipeline Studio graphical capabilities to support the answers

1. Run steady state using 50 psig and obtain the inventory based on that current pressure level.

Inventory = 4.361 MMSCF

Deleted: 4.358

2. Obtain 1.5 times the inventory value.

Inventory = 4.361 MMSCF X 1.5 = 6.541 MMSCF

Deleted: 4.358

Deleted: 6.537

3. Create a transient scenario to purge from the pipe 1.5 times the inventory value obtained after run steady state. The time obtained will be the amount of time required to purge the pipe.

Note: To build the transient scenario use the diameter 8 inches for leak delivery also let the transient run the first hour using steady state or initial values.

After 1.74 hours the amount of gas is = 6.963 slightly higher than 6.537 MMSCF

Deleted: 6.957

4. Run steady state using 765 psig and obtain the inventory based on that current pressure level.

Inventory = 62.605 MMSCF 75 % = 46.953 MMSCF

Deleted: 62.560

Deleted: 46.687

5. Assuming that the 75 % of the inventory value obtained in steady state is the minimum required before to operate the pipeline; modify the transient scenario to determine how long does it take to reach the closest value to the 75 % of the steady state inventory value?

The value obtained for the Inventory is 46.700 MMSCF after 0.66 Hours (3.41-2.74)

pipelinestudio®

GAS NETWORK TRAINING

MODULE : RELIEF VALVE

energysolutions®

PipelineStudio Training: Module Relief Valve I
The Module I
The User I
The Training I
MAOP 1
Case1 (MAOP) 1
RELIEF VALVE 6
Case1A (Relief Valve) 6

PipelineStudio Training: Module Relief Valve

The Module

The module contains a series of cases; starting with a case that will be used as starting point to generate additional cases.

The cases will illustrate the importance of having a device that can release products and avoid over pressure in the pipeline.

The User

This ***pipelinestudio*** Training Class is intended to the users of the product or someone who wants to become more productive using ***pipelinestudio***.

The goal of the ***pipelinestudio*** training class is to provide a pipeline engineer, technician or professional with all of the skills and familiarity with the product needed to allow them to begin making productive and efficient use of the product.

The Training

pipelinestudio Training is being provided in a laboratory-type environment with the intent to provide the trainee with an interactive and high-ratio of hands on experience. The purpose of this is to keep things interesting and provide better overall retention by the student.

pipelinestudio training is divided into ordered and methodical labs to allow the user to build on skills obtained from the previous lab and maximize the retention of knowledge associated with this training. Lecture and discussion will be incorporated into the labs themselves.

MAOP

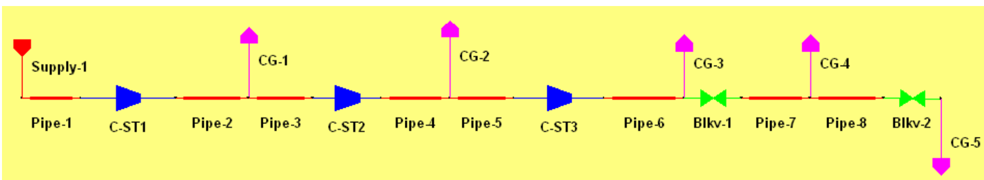
Purpose

This case demonstrate the hydraulic behavior when a delivery suddenly change from the normal operation to "0" flow. The pressure upstream of the delivery will increase to the level that goes above of the MAOP established.

Case1 (MAOP)

Instructions

Use the configuration layout as a reference to configure the network and use the configuration name: "Case1"



Fluid **Fluid1** properties:

Specific Gravity	0.6	
Heating Value	1000	BTU/cf
Carbon Dioxide	0	percent

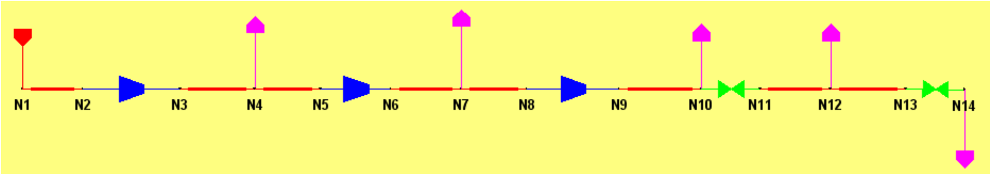
Supply **Supply-1** details:

Fluid Temperature	90	Deg F
Maximum Pressure	700	Psig

Pipes details:

Pipe Name	Length in Miles	Diameter (inside) in inch	Wall Thickness in inch	Roughness in inch	Knot Spacing in Miles	Efficiency
Pipe-1	29.204	12.126	0.312	0.001	1	1
Pipe-2	15.534	10.02	0.312	0.001	1	1
Pipe-3	17.398	10.02	0.312	0.001	1	1
Pipe-4	6.213	12.126	0.312	0.001	1	1
Pipe-5	9.32	12.126	0.312	0.001	1	1
Pipe-6	12.427	12.126	0.312	0.001	1	1
Pipe-7	12.427	12.126	0.312	0.001	1	1
Pipe-8	52.816	12.126	0.312	0.001	1	1

Nodes **Location**:



Nodes **Elevation** details:

Node Name	Elevation in ft	Node Name	Elevation in ft
N1	820.21	**N8**	6561.68
N2	2460.63	**N9**	6561.68
N3	2460.63	**N10**	4921.26
N4	3169.29	**N11**	4921.26
N5	3937.01	**N12**	3280.84
N6	3937.01	**N13**	328.084
N7	5045.93	**N14**	328.084

Delivery **CG-1** details:

Location: Node N4

Maximum Flow	18	MMSCFD
Mode	Max Flow	

Delivery **CG-2** details:
Location: Node N7

Maximum Flow	15	MMSCFD
Mode	Max Flow	

Delivery **CG-3** details:
Location: Node N10

Maximum Flow	14	MMSCFD
Mode	Max Flow	

Delivery **CG-4** details:
Location: Node N12

Maximum Flow	1	MMSCFD
Mode	Max Flow	

Delivery **CG-5** details:
Location: Node N14

Maximum Flow	36	MMSCFD
Mode	Max Flow	

Block Valves details:

Valve Name	CV	Size in inches	Node UP	Node DN
Blkv-1	50000	12	N10	N11
Blkv-2	50000	12	N13	N14

Generic Compressor **C-ST1** details:

Location: Node **N2** UP **N3** DN

Adiabatic Efficiency	100	%
Mechanical Efficiency	100	%
Max Down Pressure	1100	
Mode	Max Down Pressure	

Generic Compressor **C-ST2** details:

Location: Node **N5** UP **N6** DN

Adiabatic Efficiency	100	%
Mechanical Efficiency	100	%
Max Down Pressure	1100	
Mode	Max Down Pressure	

Generic Compressor **C-ST3** details:
Location: Node **N8** UP **N9** DN

Adiabatic Efficiency	100	%
Mechanical Efficiency	100	%
Max Down Pressure	1100	
Mode	Max Down Pressure	

Case Study 1 (continued)

OBSERVATIONS AND QUESTIONS

After the network has been configured, add the trends accordingly and continue with the following instructions:

A) Turn “On” the temperature tracking
B) Enable the Alarms, select the block valve Blkv-2 and input 1250 psig for HiHi alarm
C) Using the Network View Properties color the devices by “Status”
D) Add data blocks to verify pressures and flows
E) Run steady state
F) Select pressure profile, disable the Autoscales and use as a upper value 1600 psig
G) Split the Network View and the profile window (Tile Horizontally)
H) Activate the Interactive Transient simulation
I) Select the delivery "City_Gate-5" and change the flow to "0"
J) Start the Interactive Transient simulation
K) Verify the time when the pressure reach 1250 psig (MAOP = 1250 psig)
L) Run the simulation enough time to review the trends and the alarm window

Case Study 1 (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

After the network has been configured, add the trends accordingly and continue with the following instructions:

A) Turn "On" the temperature tracking

B) Enable the Alarms, select the block valve Blkv-2 and input 1250 psig for HiHi alarm

C) Using the Network View Properties color the devices by "Status"

D) Add data blocks to verify pressures and flows

E) Run steady state

F) Select pressure profile, disable the Autoscales and use as a upper value 1600 psig

G) Split the Network View and the profile window (Tile Horizontally)

H) Activate the Interactive Transient simulation

I) Select the delivery "City_Gate-5" and change the flow to "0"

J) Start the Interactive Transient simulation

K) Verify the time when the pressure reach 1250 psig (MAOP = 1250 psig)

L) Run the simulation enough time to review the trends and the alarm window

After 32 minutes the pressure reached is 1250 psig

Deleted: 25

RELIEF VALVE

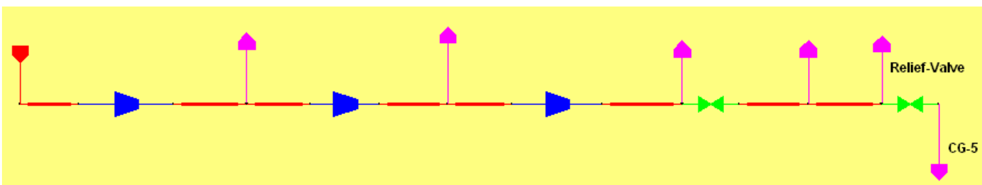
Purpose

This case demonstrate the hydraulic behavior when a delivery suddenly change from the normal operation to "0" flow. The pressure upstream of the delivery will increase to the level that is necessary to use a relief valve to avoid violation to the MAOP. This case study is designed to help identify the requirements for a relief valve in order to protect any specific location in the network.

Case1A (Relief Valve)

Instructions

Open the Case1 and use the configuration layout as a reference to configure the location of the relief valve and use the configuration name: "Case1A"



Since the Maximum Allowable Operating Pressure (MAOP) has been established = 1250 psig the setpoint for the relief valve is 1245 psig.

Delivery **Relief-Valve** details:
Location: Node N13

Maximum Pressure	1245	psig
Maximum Pressure	1245	psig
Check Valve	Yes	
Mode	Check	

After the network has been configured add the trends accordingly; continue with the instructions (see next page):

Case Study 1A (continued)

OBSERVATIONS AND QUESTIONS

After the network has been configured, add the trends accordingly and continue with the following instructions:

A) Turn “On” the temperature tracking

B) Select the block valve Blkv-2 and input 1244 psig for HiHi alarm

C) Select the delivery “Relief-Valve” and enable the trends Accumulate Volume, pressure and flow

D) Run steady state

E) Select pressure profile, disable the Autoscales and use as a upper value 1600 psig

F) Split the Network View and the profile window (Tile Horizontally)

G) Activate the Interactive Transient simulation

H) Select the delivery "City_Gate-5" and change the flow to "0"

I) Start the Interactive Transient simulation

J) Verify the time when the pressure reach 1245 psig (MAOP = 1250 psig)

K) Run the simulation enough time to review the trends and the alarm window

Verify the amount of product that needs to exit the pipeline to keep the pressure around 1245 psig?

Case Study 1A (continued)

INSTRUCTOR'S SOLUTION

OBSERVATIONS AND QUESTIONS

After the network has been configured, add the trends accordingly and continue with the following instructions:

A) Turn "On" the temperature tracking

B) Select the block valve Blkv-2 and input 1244 psig for HiHi alarm

C) Select the delivery "Relief-Valve" and enable the trends Accumulate Volume, pressure and flow

D) Run steady state

E) Select pressure profile, disable the Autoscales and use as a upper value 1600 psig

F) Split the Network View and the profile window (Tile Horizontally)

G) Activate the Interactive Transient simulation

H) Select the delivery "City_Gate-5" and change the flow to "0"

I) Start the Interactive Transient simulation

J) Verify the time when the pressure reach 1245 psig (MAOP = 1250 psig)

K) Run the simulation enough time to review the trends and the alarm window

After 21 minutes the pressure reached is 1244 psig

Deleted: 19

Verify the amount of product that needs to exit the pipeline to keep the pressure around 1245 psig?

Accumulated Volume = 1 MMSCF after 1hour43mins

Deleted: 0.99

Deleted: 1.55

Deleted: hours