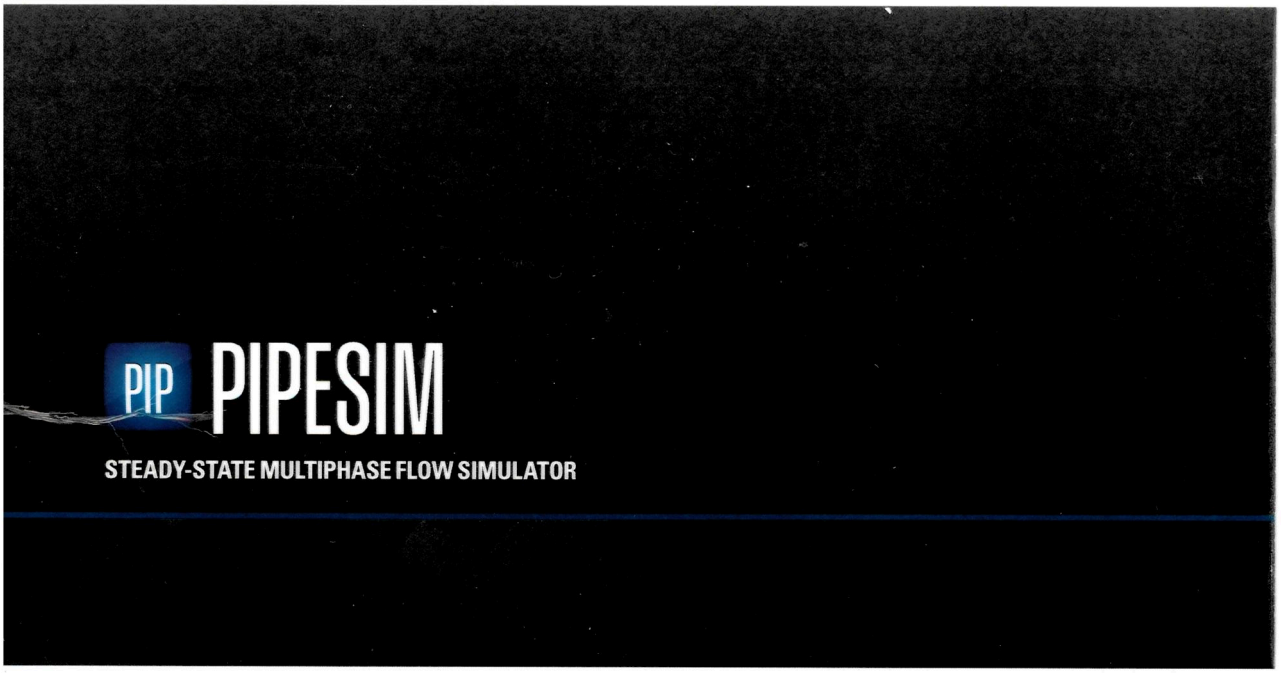
Fundamentals

Training and Exercise Guide



NExT

A Schlumberger Company

PIPESIM Fundamentals

Workflow/Solutions Training

Version 2014

Schlumberger Information Solutions
April 22, 2015

NExT
A Schlumberger Company

Schlumberger

Copyright Notice

Copyright © 2015 Schlumberger. All rights reserved.

This work contains the confidential and proprietary trade secrets of Schlumberger and may not be copied or stored in an information retrieval system, transferred, used, distributed, translated or retransmitted in any form or by any means, electronic or mechanical, in whole or in part, without the express written permission of the copyright owner.

Trademarks & service marks

Schlumberger, the Schlumberger logotype, and other words or symbols used to identify the products and services described herein are either trademarks, trade names or service marks of Schlumberger and its licensors, or are the property of their respective owners. These marks may not be copied, imitated or used, in whole or in part, without the express prior written permission of Schlumberger. In addition, covers, page headers, custom graphics, icons, and other design elements may be service marks, trademarks, and/or trade dress of Schlumberger, and may not be copied, imitated, or used, in whole or in part, without the express prior written permission of Schlumberger. Other company, product, and service names are the properties of their respective owners.

An asterisk (*) is used throughout this document to designate a mark of Schlumberger.

Security notice

The software described herein is configured to operate with at least the minimum specifications set out by Schlumberger. You are advised that such minimum specifications are merely recommendations and not intended to be limiting to configurations that may be used to operate the software. Similarly, you are advised that the software should be operated in a secure environment whether such software is operated across a network, on a single system and/or on a plurality of systems. It is up to you to configure and maintain your networks and/or system(s) in a secure manner. If you have further questions as to recommendations regarding recommended specifications or security, please feel free to contact your local Schlumberger representative.

Table of Contents

About this manual

Learning objectives 1
What you need 1
What to expect 2
Course conventions 3
Icons 4
Summary 5

Module 1: PIPESIM introduction

Learning objectives 7
Lesson 1: Introduction 7
Lesson 2: Tour of the user interface 11
Lesson 3: Tasks 15
Network simulation 16
Pressure/Temperature profile 17
Nodal analysis 17
System analysis 18
VFP tables 18
ESP design 19
Review question 19
Summary 19

Module 2: Simple pipeline tutorials

Learning objectives 21
Lesson 1: Single-phase flow calculations 21
Exercise 1: Model a water pipeline with hand calculations 23
Exercise 2: Build the water pipeline model with PIPESIM 27
Exercise 3: Run the model 33
Exercise 4: Analyze multiple scenarios with sensitivities 38
Exercise 5: Model a single-phase gas pipeline 40
Exercise 6: Calculate gas pipeline flow capacity 44
Lesson 2: Multiphase flow calculations 46
Exercise 1: Model a multiphase pipeline 49
Review questions 53
Summary 53

Module 3: Oil well performance analysis

Learning objectives . . . 55
Lesson 1: Nodal analysis . . . 55
Exercise 1: Build the well model . . . 57
Exercise 2: Perform a nodal analysis . . . 64
Lesson 2: Pressure/temperature profile . . . 67
Exercise 1: Generate a pressure/temperature profile . . . 67
Lesson 3: Black oil fluid calibration . . . 68
Single point calibration . . . 69
Save fluids and wells as templates . . . 69
Exercise 1: Calibrate PVT data . . . 70
Lesson 4: Inflow performance matching . . . 72
Exercise 1: Sensitize on the well PI to match performance . . . 73
Lesson 5: Well performance analysis . . . 75
Water-cut sensitivity analysis . . . 75
Exercise 1: Run a system analysis task . . . 75
Exercise 2: Run a nodal analysis task . . . 77
Lesson 6: Artificial lift . . . 79
Exercise 1: Evaluate gas lift performance . . . 80
Exercise 2: Evaluate ESP performance . . . 84
Exercise 3: Model multiple completions . . . 91
Lesson 7: Downhole chokes . . . 98
Exercise 1: Model a downhole choke . . . 98
Review questions . . . 100
Summary . . . 101

Module 4: Gas well performance

Learning objectives . . . 103
Lesson 1: Compositional fluid modeling . . . 103
Multiflash in the compositional fluid mode (native) . . . 104
Multiflash MFL files . . . 104
Equations of State . . . 105
Exercise 1: Create a compositional fluid model . . . 110
Lesson 2: Gas well deliverability . . . 115
Exercise 1: Calculate gas well deliverability . . . 117
Exercise 2: Calibrate the inflow model using multipoint test data . . . 118
Lesson 3: Erosion prediction . . . 120
Exercise 1: Select a tubing size . . . 121
Lesson 4: Choke modeling . . . 123
Exercise 1: Model a flowline and choke . . . 125
Exercise 2: Predict future production rates . . . 129

Lesson 5: Liquid loading 130
Turner droplet model 130
Exercise 1: Determine a critical gas rate to prevent well loading 132
Review questions 136
Summary 136

Module 5: Horizontal well design

Learning objectives 139
Lesson 1: Inflow performance for horizontal completions 139
Exercise 1: Horizontal well performance 142
Review questions 147
Summary 147

Module 6: Subsea tieback design

Learning objectives 149
Lesson 1: Flow assurance considerations 150
Exercise 1: Size the subsea tieback and riser 151
Lesson 2: Hydrates 162
Hydrate mitigation strategies in PIPESIM 163
Exercise 1: Select tieback insulation thickness 164
Exercise 2: Determine the methanol requirement 167
Lesson 3: Severe riser slugging 171
Exercise 1: Screen for severe riser slugging 173
Lesson 4: Slug catcher sizing 174
Hydrodynamic slugging 174
Pigging 175
Ramp-up 176
Evaluation of each scenario 177
Exercise 1: Size a slug catcher 177
Review questions 180
Summary 181

Module 7: Looped gas gathering network

Learning objectives 183
Lesson 1: Model a gathering network 183
Boundary conditions 184
Solution criteria 184
Exercise 1: Model a pipeline network 185
Exercise 2: Screen the network for erosion issues 196
Review questions 197
Summary 198

Module 8: Simple network model on the GIS map

Learning objectives 202
Lesson 1: Pipeline modeling 202
Exercise 1: Build the network model on a map 202
Review question 217
Summary 217

Appendix A: PIPESIM 2014 Fundamentals: Answer key

Module 2: Simple pipeline tutorial 219
Lesson 1: Single-phase flow calculations 219
Module 3: Oil well performance analysis 220
Lesson 1: Nodal analysis 220
Lesson 2: Pressure/temperature profile 220
Lesson 3: Black oil fluid calibration 220
Lesson 4: Inflow performance matching 221
Lesson 5: Well performance analysis 221
Lesson 6: Artificial lift 221
Lesson 7: Downhole chokes 222
Module 4: Gas well performance 223
Lesson 2: Gas well deliverability 223
Lesson 3: Erosion prediction 224
Lesson 4: Choke modeling 224
Lesson 5: Liquid loading 225
Module 5: Horizontal well design 225
Lesson 1: Inflow performance for horizontal completions 225
Module 6: Subsea tieback design 226
Lesson 1: Flow assurance considerations 226
Lesson 2: Hydrates 226
Lesson 3: Severe riser slugging 227
Lesson 4: Slug catcher sizing 227
Module 7: Looped gas gathering network 228
Lesson 1: Model a gathering network 228
Module 8: Simple network model on the GIS map 228
Lesson 1: Pipeline modeling 228

About this manual

This training provides an introduction to the **PIPESIM*** software application. **PIPESIM** is a production engineering technology that covers a wide range of applications relevant to the oil and gas industry.

Workflows featured in this training manual include well performance, fluid modeling, flow assurance, and network simulation.

Learning objectives



After completing this training, you will know how to:

- build a well or pipeline model
- define a black oil or compositional fluid model
- build a network model
- perform simulation operations and analyze the results

What you need



In this training, you need the following documents, hardware, and software:

- **PIPESIM 2014.1**
- A computer that meets the hardware/software requirements defined in the installation guide.
- Appropriate licenses for the **PIPESIM** modules to be used

What to expect

In each module in this training material, you encounter the following:

- Overview of the module
- Prerequisites to the module (if necessary)
- Learning objectives
- A workflow component (if applicable)
- Lessons, that explain a subject or an activity in the workflow
- Procedures, that show the steps needed to perform a task
- Exercises, that allow you to practice a task by using the steps in the procedure with a dataset
- Scenario-based exercises
- Questions about the module
- Summary of the module.

You also encounter notes, tips and best practices.

Course conventions

Content in this manual uses the following conventions.

Characters in **Bold**	Names of windows, dialog boxes, menus and tabs, such as "the **Open Asset Model** dialog box." Keyboard commands and action buttons, such as "click **OK**." Names of tools and icons, such as "**Query** tool" Schlumberger software application names, such as **Petrel***, **Techlog***, or **ECLIPSE***
Characters in *italics*	Names of files, directories and drive navigation paths, such as "the *sample.dat* file." Names of sections and areas in windows and dialog boxes, such as "*Experiments* area" First use of important terms or concepts, such as *compositional* simulation or "*Safe mode* is used when..."
Characters in `fixed-width`	Code, data, and other literal text the user sees or enters, such as "Enter `0.7323`."
Characters in *Times Italic*	Names of options, functions, processes, lists, columns, check boxes and option buttons. For example, choose a calculation method from the *Calc Method* list.

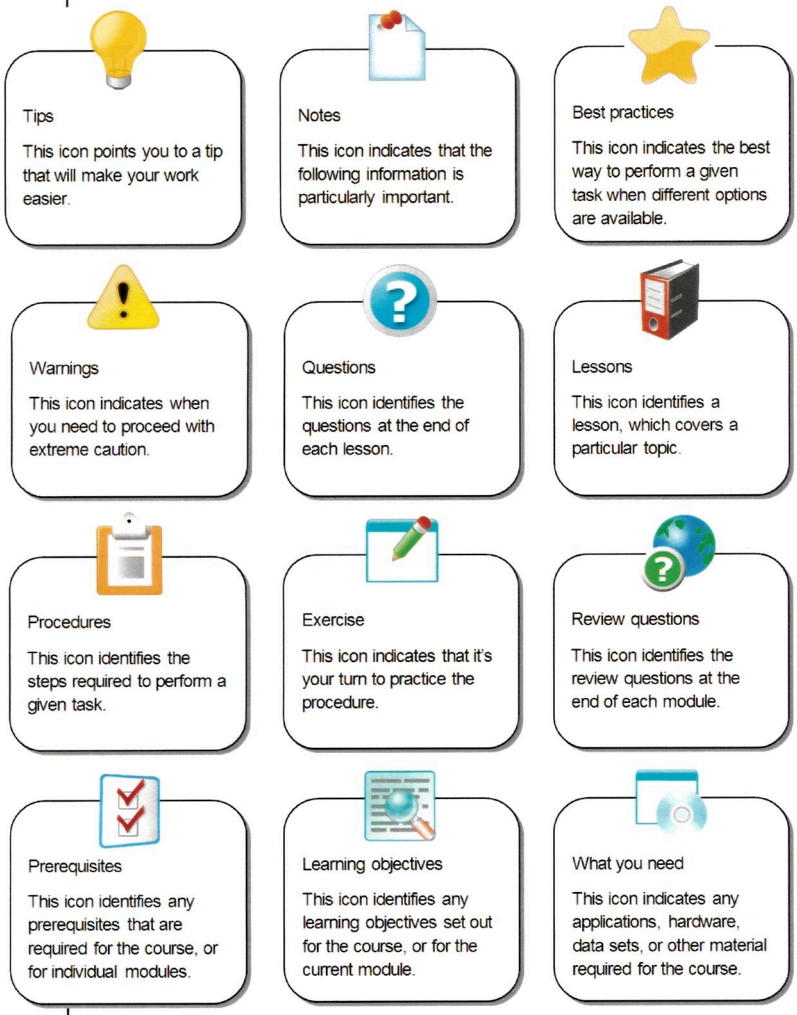
NOTE: Text you must enter is indicated in a fixed-width font or a fixed-width font inside triangle brackets. Do not include the brackets when you enter the required information.



Instructions to make menu selections are written using bold text and arrows indicating the selection sequence, such as "Select **File > Save**."

Icons

Throughout this manual, you find icons in the margin representing various kinds of information. These icons serve as at-a-glance reminders of their associated text. See below for descriptions of what each icon means.



Tips

This icon points you to a tip that will make your work easier.

Notes

This icon indicates that the following information is particularly important.

Best practices

This icon indicates the best way to perform a given task when different options are available.

Warnings

This icon indicates when you need to proceed with extreme caution.

Questions

This icon identifies the questions at the end of each lesson.

Lessons

This icon identifies a lesson, which covers a particular topic.

Procedures

This icon identifies the steps required to perform a given task.

Exercise

This icon indicates that it's your turn to practice the procedure.

Review questions

This icon identifies the review questions at the end of each module.

Prerequisites

This icon identifies any prerequisites that are required for the course, or for individual modules.

Learning objectives

This icon identifies any learning objectives set out for the course, or for the current module.

What you need

This icon indicates any applications, hardware, data sets, or other material required for the course.

Summary

In this introduction, we:

- defined the learning objectives
- outlined what tools you need for this training
- discussed course conventions that you encounter in this material

NOTES

Module 1 PIPESIM introduction

This module introduces **PIPESIM 2014** and describes the graphical user interface (GUI) to familiarize you with the application environment.

Learning objectives



After completing this module, you will be familiar with:

- separately licensed modules available in **PIPESIM**
- how to navigate the user interface
- the simulation tasks you can perform using **PIPESIM**

Lesson 1 Introduction



PIPESIM is a steady-state multiphase flow simulator used for the design and analysis of oil and gas production systems. With its rigorous simulation algorithms, **PIPESIM** helps you optimize your production and injection operations. As shown in Figure 1, **PIPESIM** models multiphase flow from the reservoir through to the surface facilities to enable comprehensive production (and injection) system analysis.

PIPESIM is most often used by reservoir, production, and facilities engineers. **PIPESIM** can be used to model well performance, conduct nodal (systems) analysis, design artificial lift systems, model pipeline networks and facilities, analyze field development plans, and optimize production.



NOTE: Steady-state flow simulation implies that the mass flow rate is conserved throughout the system. There is no accumulation of mass in any component in the system.

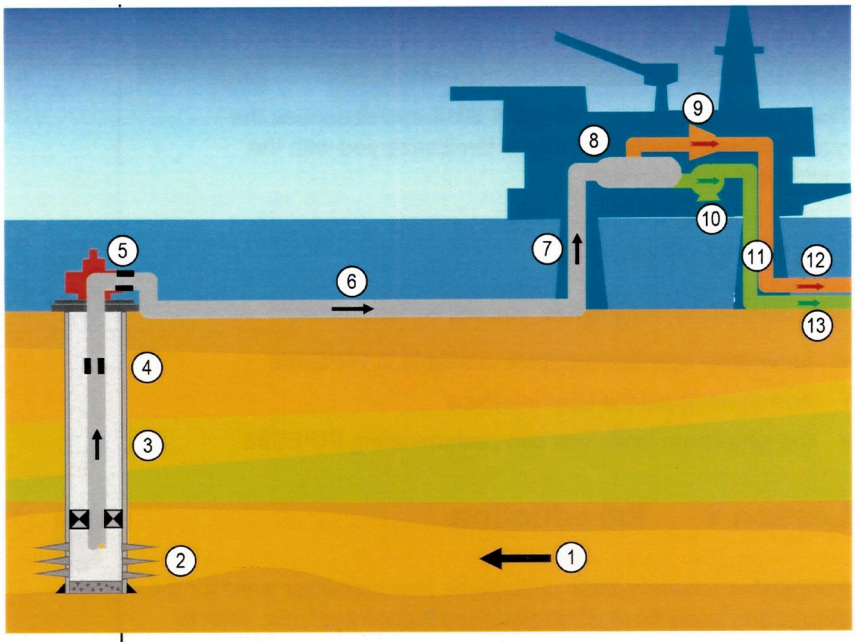


Figure 1 Total production system

1	Reservoir	**8**	Separator
2	Completion	**9**	Compressor
3	Tubing	**10**	Pump
4	Safety valve	**11**	Export lines
5	Choke	**12**	Gas
6	Flowline	**13**	Oil
7	Riser		

PIPESIM modules are available and licensed individually, depending on your needs:

Base System	Production system analysis core package for well modeling, NODAL analysis, artificial lift design, single-branch pipeline/ process facilities modeling
Network Analysis (NET)	Optional add-on to **PIPESIM** to model pipeline networks and conduct field development studies.
ECLIPSE 300 Flash Package	Optional add-on to **PIPESIM** to model compositional fluids using the **ECLIPSE 300** flash package
GERG Flash Package	Optional add-on to **PIPESIM**. The GERG-2008 Equation of State is an advanced and highly accurate method used to model gas compositions, particularly compositions rich in CO_2.
Multiflash Package	Optional add-on to **PIPESIM** for compositional fluid modeling using Multiflash (a third-party Flash package by KBC). This package enables fluid modeling and advanced flow assurance analyses.
Multiflash Hydrates	Optional add-on to **Multiflash** package to enable the calculation of hydrate dissociation conditions
Multiflash CSMA EOS	Optional add-on to **Multiflash** package to expose the CSMA Equation of State
Multiflash Package Linux Engine	Optional add-on to **Multiflash** package for Linux operating systems
Linux Computation Engines	Used only with **Avocet IAM** when **ECLIPSE Parallel** is run on a Linux cluster
PIPESIM OLGAS Steady State Flow Model: 2-Phase	Steady-state version of the 2-phase mechanistic Multiphase flow model used with the **OLGA** transient simulator

PIPESIM OLGAS Steady State Flow Model: 3-Phase	Steady-state version of 3-phase mechanistic Multiphase flow model used with the **OLGA** transient simulator (includes 2-phase option).
PIPESIM LEDA Flow Model: 2-Phase	Steady-state version of 2-phase mechanistic Multiphase Flow model from Kongsberg (third-party), developers of the LedaFlow transient simulator
PIPESIM LEDA Flow Model: 3-Phase	Steady-state version of 3-phase mechanistic Multiphase Flow Model from Kongsberg (third-party), developers of the LedaFlow transient simulator (includes 2-phase option)

Lesson 2 Tour of the user interface

The **PIPESIM** user interface allows you to construct well and network models in a single environment.

Launch **PIPESIM** from the **Start** menu by selecting **All Programs > Schlumberger > PIPESIM 2014.1 32-bit (or 64-bit) > PIPESIM 2014.1 32-bit (or 64-bit)**.

You can create a new network-centric workspace by selecting **New** in the *Network* area. A *workspace* is a concept introduced in **PIPESIM 2013**; it is the environment where all the modeling scenarios can be built and evaluated.

The **PIPESIM** interface (Figure 2), consists of a main window, where a model is constructed, surrounded by **Inputs**, **Tasks**, and **Status** (**Message center** and **Validation**) panes where most functionality is controlled using a typical ribbon toolbar.

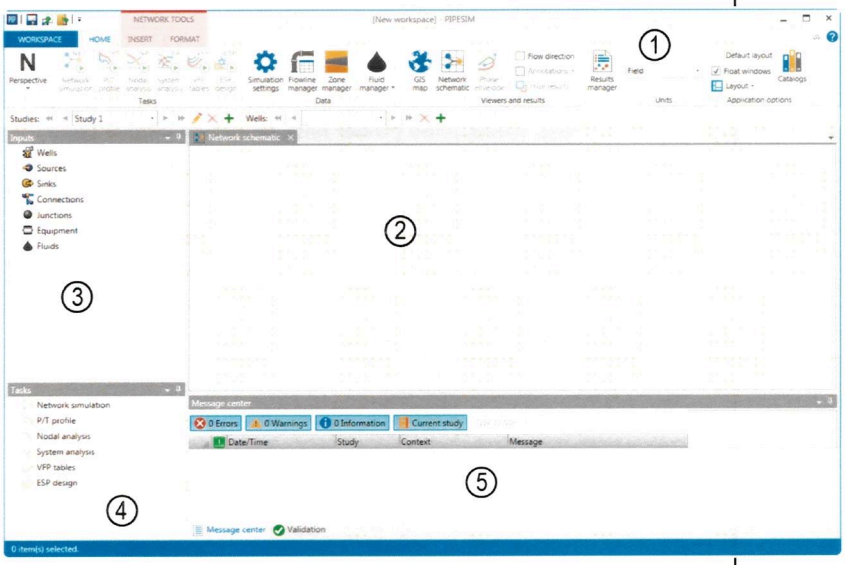


Figure 2 PIPESIM GUI

1 Ribbon toolbar
2 Main window
3 Inputs pane
4 Tasks pane
5 Message center pane

The layout of the panes can be controlled by choosing one of the predefined configurations from the **Layout** control on the **Home** tab (Figure 3).

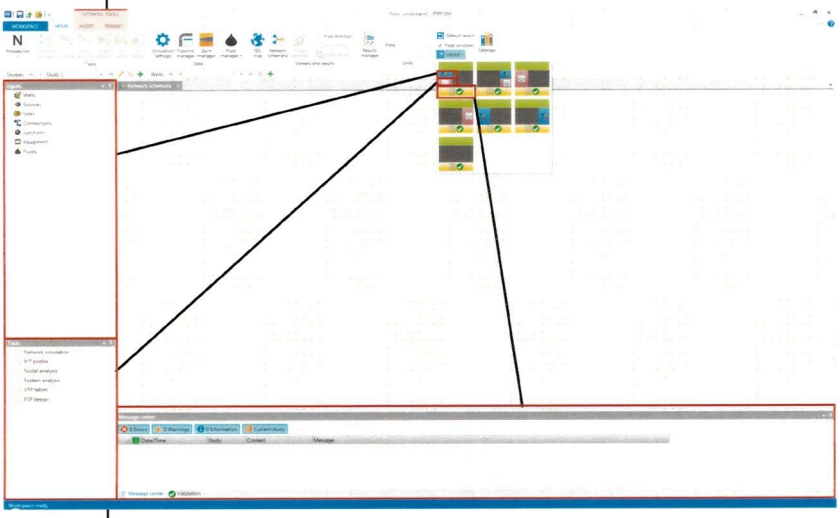


Figure 3 Predefined pane configurations: Inputs pane (upper left), Tasks pane (lower left), Message center pane (bottom)

To move panes manually, even outside the main **PIPESIM** window, drag the a pane or select the *Float* option (Figure 4).

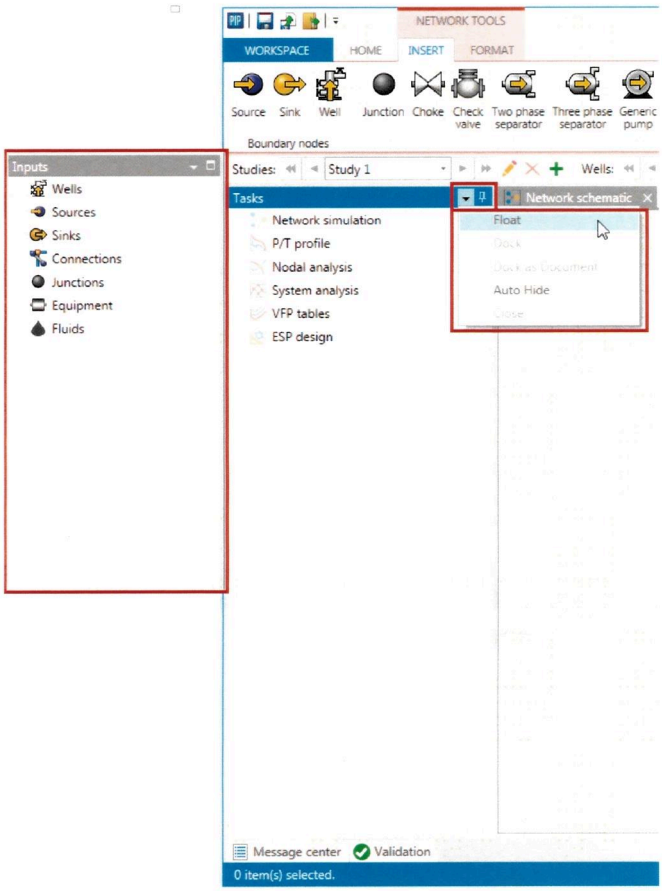


Figure 4 Inputs pane (left) using the Float view option

To reposition a pane anywhere in the **PIPESIM** window, drag the pane using the direction arrow prompts that appear (Figure 5).

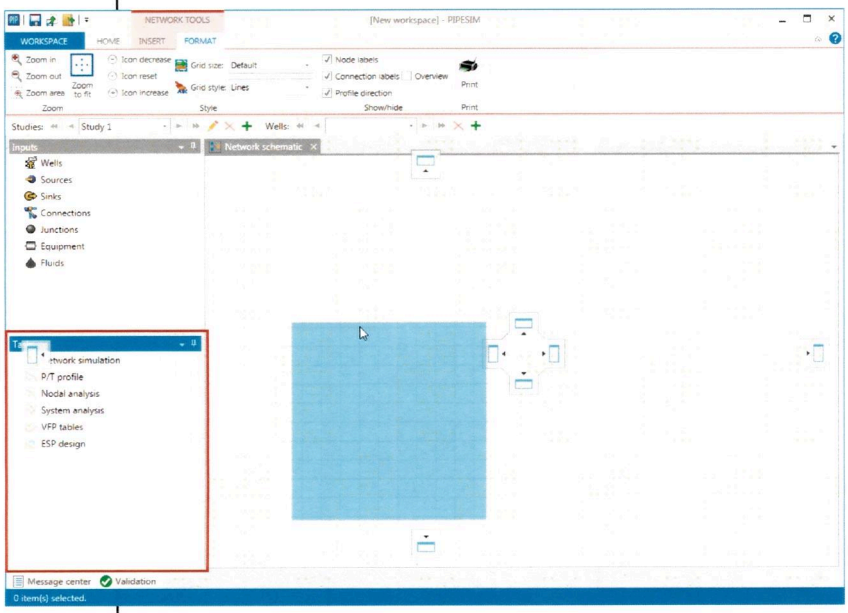


Figure 5 Position a pane

Most ribbon controls have tooltip descriptions that appear (Figure 6). Explore the options on all the tabs so you will become familiar with the breadth of functionality in **PIPESIM**. The training in this manual uses most of the controls.

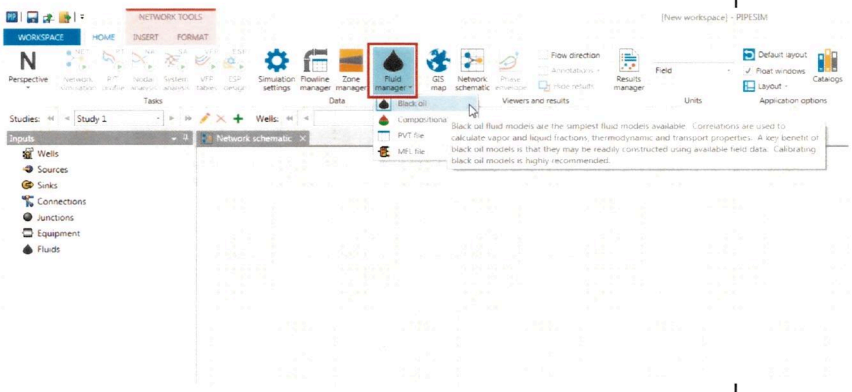


Figure 6 Tooltip example

Lesson 3 Tasks

There are many tasks you can launch from the **Tasks** pane: Network simulation, P/T profile, Nodal analysis, System analysis, VFP tables, and ESP design (Figure 7).

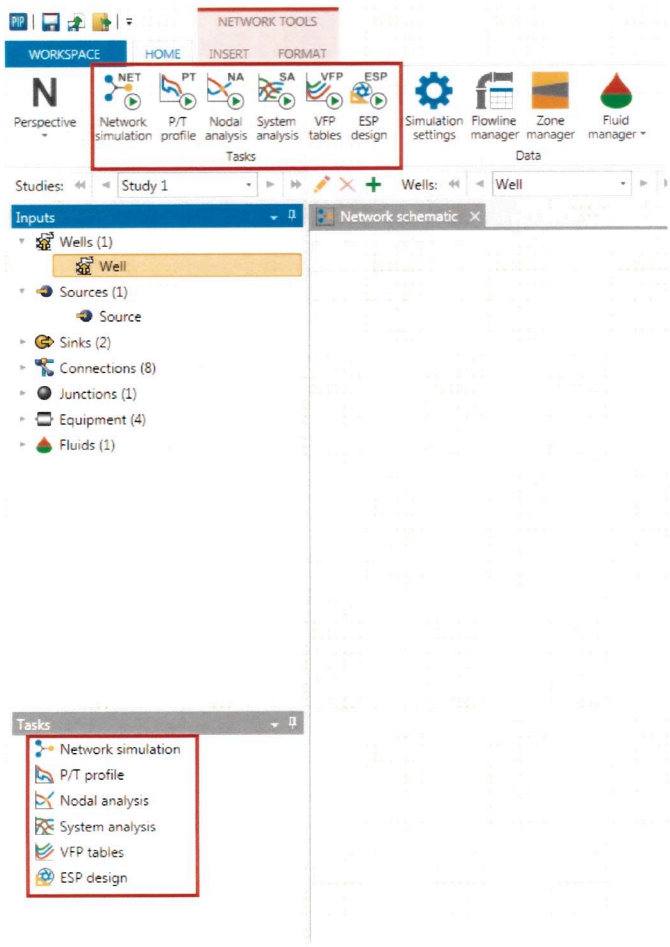


Figure 7 Task options

Network simulation

PIPESIM solves the source to sink (beginning to end) of a piping network for pressure, flow rate, and temperature.

Pressure/Temperature profile

PIPESIM generates pressure and temperature profiles of the system as a function of distance/elevation along the system. Both temperature and pressure profiles are generated on a node-by-node basis for the system.

Nodal analysis

A common way to analyze well performance is through a Nodal Analysis plot, which allows you to visually assess the impact of various system components.

This is done by splitting the system at the point of interest, known as the *Nodal analysis point*, and graphically representing the system response upstream (inflow) and downstream (outflow) of the nodal point.

The point at which the inflow and outflow curves intersect is the *operating* point for the given system, as shown in Figure 8.

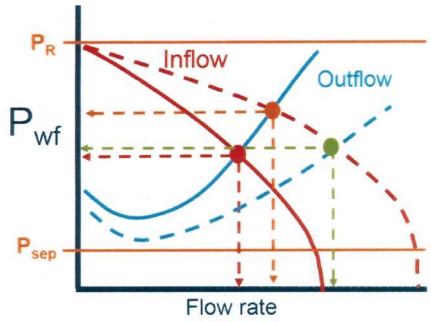


Figure 8 Nodal analysis Inflow/Outflow curves

System analysis

The system analysis task enables you to determine the performance of a given system for varying operating conditions on a case-by-case basis. The results of the system analysis appear in the form of plots of a dependent variable, such as outlet pressure, versus an independent variable, such as flow rate.

Families of X-Y curves can be generated by varying a single sensitivity variable (such as water cut) or by applying permutations and combinations of a group of sensitivity variables.

The ability to perform analysis by combining sensitivity variables in different ways makes the system analysis task a flexible tool for plotting data on a case-by-case basis. A simplified plot from a system analysis task is shown in Figure 9.

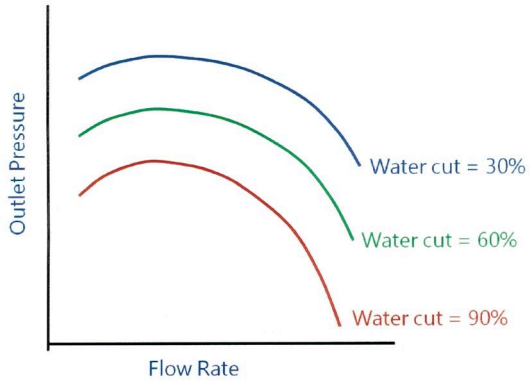


Figure 9 Typical system analysis plot

VFP tables

For the purposes of reservoir simulation, it is often necessary to generate vertical flow performance (VFP) curves for input to a reservoir simulator. The VFP curves allow the reservoir simulator to determine bottomhole flowing pressures as a function of tubing head pressure, flow rate, GOR, water cut, and various artificial lift variables.

The VFP tables task allows you to write tabular performance data to a file for input into a reservoir simulation model.

Currently, these reservoir simulators are supported:

- ECLIPSE
- PORES
- VIP
- COMP4
- MoReS (Shell Oil in-house reservoir simulator).

ESP design

This task allows you to specify target conditions for the ESP design, select the ESP from a catalog of pumps that are ranked based on efficiency at the specified design conditions, design the pump, and install it in the well. The task also allows you to design and install multiple (tandem) ESPs in a single well.

Review question



What simulation tasks are available in **PIPESIM**?

Summary

In this module, you learned about:

- individually licensed modules available in **PIPESIM**
- navigating the user interface
- the simulation tasks you can perform using **PIPESIM**

NOTES

Module 2 Simple pipeline tutorials

This module discusses how **PIPESIM** can model pipelines by performing a simple hand calculation to determine the pressure drop in a water pipeline, then constructing a simple pipeline model to validate pressure drop along a horizontal pipeline for a given inlet pressure and flow rate.

You also run sensitivity studies on the model.

Learning objectives



After completing this module, you will know how to:

- build the physical model
- create a fluid model
- choose flow correlations
- perform operations
- view and analyze results

Lesson 1 Single-phase flow calculations



Consider the case of a pipeline transporting water (Figure 10).

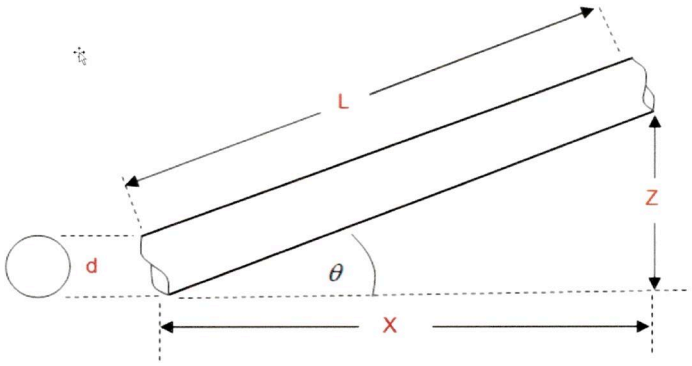


Figure 10 Pipeline transporting water

The pressure change per distance *L* for single phase flow is given by Bernoulli's equation:

$$\left(\frac{dp}{dL}\right)_{total} = \left(\frac{dp}{dL}\right)_{frictional} + \left(\frac{dp}{dL}\right)_{elevational} + \left(\frac{dp}{dL}\right)_{accelerational}$$

The *accelerational* term is normally negligible except for low pressure and high velocity gas flow, although **PIPESIM** always calculates it.

Assuming the accelerational term to be zero for your hand calculation, the pressure gradient equation becomes:

$$\left(\frac{dP}{dL}\right)_{total} = \frac{f\rho v^2}{2gd}_{frictional} - \rho g \sin\theta_{elevational}$$

where:

ρ = fluid density (lbm/ft^3)

g = gravitational constant

f = Moody friction factor

v = fluid velocity (ft/s)

d = pipe inside diameter (ft).

Exercise 1 Model a water pipeline with hand calculations



Use the data in the table and assume the flow is isothermal. Perform hand calculations to determine the delivery pressure of the pipeline using single-phase flow theory.



NOTE: You must have a hand calculator or spreadsheet to complete this exercise.

Pipeline data				
Diameter	d	3.068	in	(= 0.26 ft)
Length	L	20,025	ft	
Elevation Change	Z	1,000	ft	
Horizontal distance	X	20,000	ft	
Ambient temperature	T_{amb}	60	degF	
Inclination angle	q	2.862	deg	(=.04996 radians)
Roughness	e	0.0018	in	
Relative roughness	*e/d*	0.0005867	in	
Fluid data				
Water viscosity	m_w	1.2	cp	(= 8.06e-4 lb/ft-s)
Water density	r_w	63.7	lbm/ft^3	
Operating data				
Source temperature	T_{inlet}	60	degF	
Inlet pressure	P_{in}	1,200	psia	
Water flow rate	Q_w	6,000	BPD	(= 0.39 ft^3/s)
Constants				
Gravitational	g	32.2	ft/s^2	

TIP: To ensure unit consistency when performing the hand calculations, refer to the converted units in the far right column of the table.

1. Calculate the water velocity.

$$v = \frac{Q_w}{\left(\frac{\pi d^2}{4}\right)} = _____ \text{ ft/s}$$

2. Calculate the Reynold's number.

$$\text{Re} = \frac{\rho v d}{\mu} = _____$$

Is the flow laminar or turbulent? (See the Moody diagram in Figure 11.)

3. Determine the friction factor using the Churchill equation for turbulent flow.

NOTE: Alternatively, you can look up the friction factor using the Moody diagram (Figure 11).

$$f = 8\left[\left(\frac{8}{Re}\right)^{12} + \frac{1}{(A+B)^{1.5}}\right]^{\frac{1}{12}}$$

$$A = \left[2.457\, ln\left(\frac{1}{\left(\frac{7}{Re}\right)^{.9} + 0.27(\varepsilon/d)}\right)\right]^{16}$$

$$B = \left[\frac{37530}{Re}\right]^{16}$$

f = _____

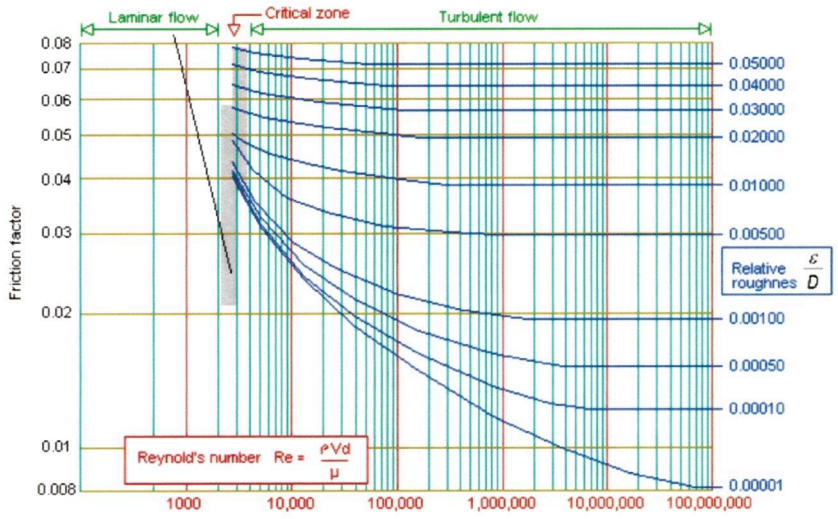


Figure 11 Moody diagram

4. Evaluate the frictional pressure term, $\frac{f\rho v^2}{2gd}$:

   $\left(\frac{dp}{dL}\right)_{friction}$ = _____ psf/ft

   Divide this by 144 to get_____ psi/ft

   Multiply by the given length of pipe, L, to get the total frictional pressure drop:

   $dp_{friction}$ = _____ psi

5. Evaluate the elevational pressure term $\rho \sin\theta$.

   NOTE: If you are using Excel, be sure the angle is expressed in *radians*.

   $\left(\frac{dp}{dL}\right)_{elevational}$ = _____ psf/ft

   Divide this by 144 to get_____ psi/ft

Multiply by the given length of pipe, L, to get the total elevational pressure drop

$dp_{elevation}$ = _____ psi

6. Add the frictional and elevational terms to determine the total pressure term:

$$\left(\frac{dP}{dL}\right)_{total} = \left(\frac{dP}{dL}\right)_{frictional} + \left(\frac{dP}{dL}\right)_{elevational}$$

$\left(\frac{dp}{dL}\right)_{total}$ = _____ psi/ft

Multiply by the given length of pipe, L, to get the total pressure drop

dp_{total} = _____ psi

7. Calculate the outlet pressure given the inlet pressure:

$P_{out} = P_{in} - dp_{total}$ = _____ psia



NOTE: There are two levels available in **PIPESIM** for units customization:
1) *Global Unit System*: Select the default global unit system under **Workspace** > **Options**. The units selected here are used for every new workspace created.

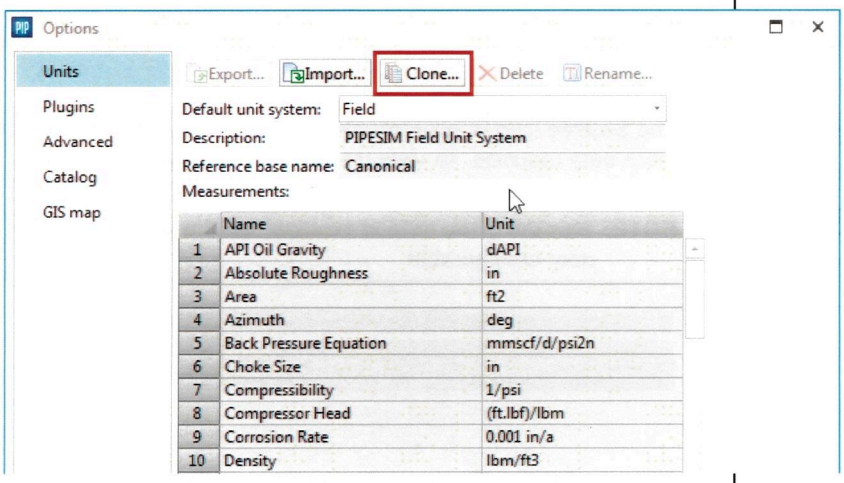
2) *Local Unit System*: For specific workspaces, you can use a local unit system that is different from the default global units. Select **Home > Units** and choose your preferred local unit system from the options list. These units are saved with the workspace and preserved each time you reopen the workspace.

Exercise 2 Build the water pipeline model with PIPESIM

In this exercise, you use **PIPESIM** to model the water pipeline that was hand calculated in the previous exercise. You define parameters for each component in the model, perform operations, view and analyze the results, and compare **PIPESIM** results to your hand calculations.

1. Start **PIPESIM**.
2. On the **Workspace** tab, choose *Options*.
3. Select Field for the Default unit system.
4. Click **Close** to close the window.

NOTE: None of the units supplied by the system are user configurable. Click **Clone** to begin creating a user-customizable unit system.

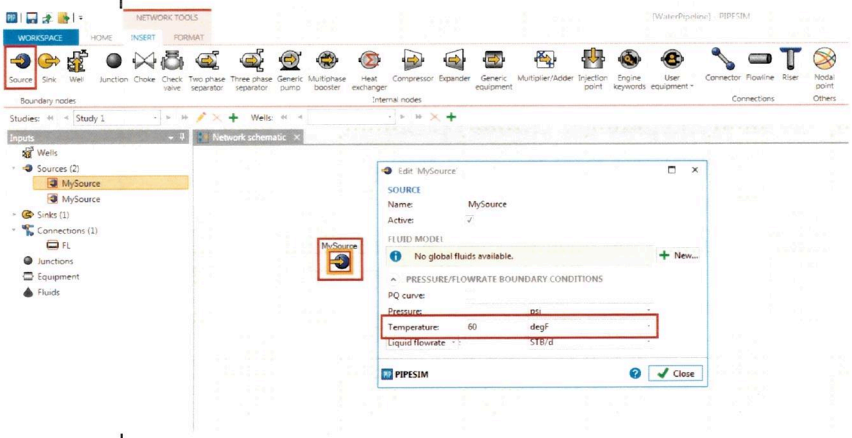


5. Return to the **Workspace** tab.
6. Under N*etwork,* select **New** to create a new, network-centric workspace.
7. On the **Insert** tab, click **Source**.
8. Click the network diagram to place it.

9. The red highlight around the icon indicates that some required setup data for the source is missing.

   Double-click the icon to open its edit window.

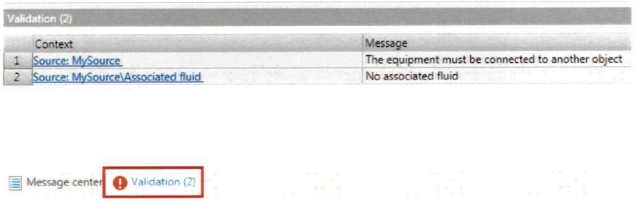
10. Rename the source `MySource`.
11. Enter a temperature of `60 degF` and click **Close**.



12. The **Source** icon is still highlighted in red, indicating there is more data required. To view the list of missing data, click the **Validation** tab at the bottom of the **PIPESIM** window.



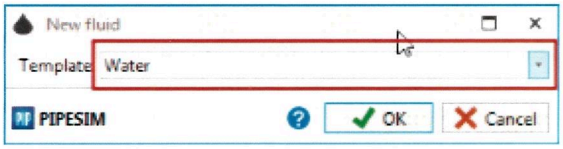
NOTE: This tab is the default location, but the position is user configurable.



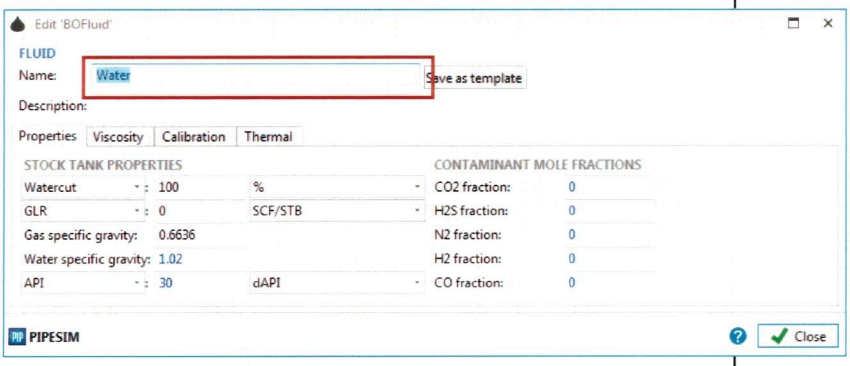
13. You can click the link in the second row to go to the location where a fluid can be created for this object.

NOTE: Alternatively, you can define the fluid model in the Source you created or from the **Fluid Manager** on the **Home** tab. The **Fluid Manager** is a single, central location where fluids can be created, viewed, and edited in the model.

14. From the **Home** tab, select **Black oil** from the *Fluid manager* options list. (Black oil is the default)..) This option launches the **Fluid manager** dialog box.
15. When the **Fluid manager** window appears:
    a. To add a row, click the green + sign.
    b. Select Water as the fluid Template by choosing it from the options list.
    c. Click **OK** to create the fluid.

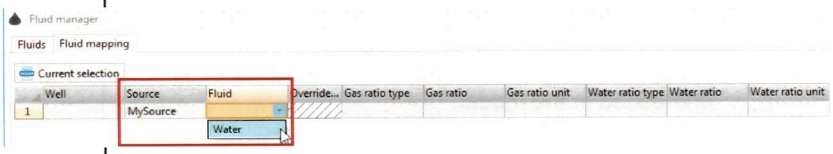


16. Double-click the new line to display the **Fluid editor** dialog box.
17. To match the entries in the figure, edit the fields.



18. Close the fluid editor.

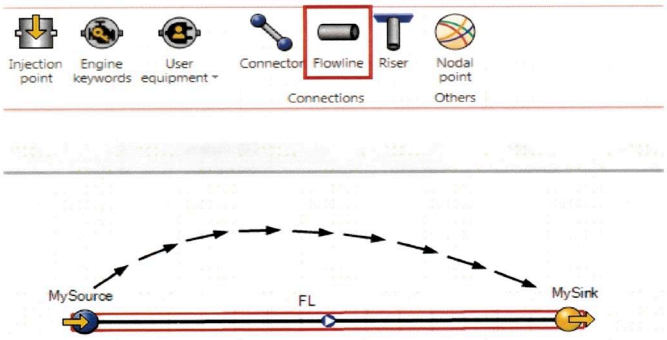
19. On the **Fluid mapping** tab of the **Fluid manager** window, associate the new Water fluid with the source that was created earlier.



20. Close the **Fluid manager**.
21. Click the **Insert** tab.
22. Add a Sink to the network diagram and rename it `MySink`.
23. Draw a flowline between the source and the sink.
    a. Click **Flowline** on the **Insert** tab ribbon.
    b. Move the cursor towards the source and drop it on the source only when the red X cursor changes to a black check mark.

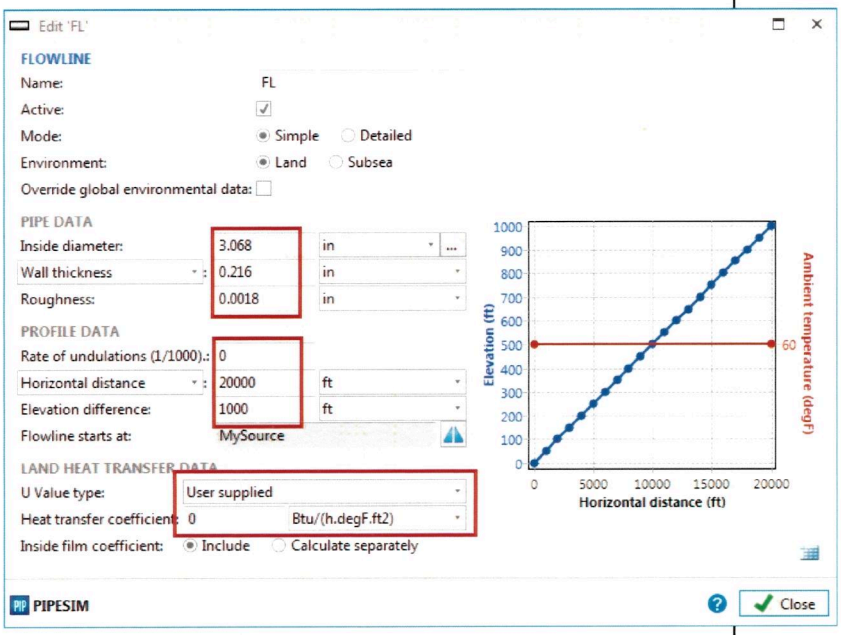
NOTE: The pointer has an accompanying small X in a red circle until it reaches the source, at which point the X turns to a check mark in a black circle. This is the signal that the pointer is positioned correctly to attach to this object. These signals apply at the starting object where the connector, flowline, or riser is anchored and at the terminal object to be connected.

The result should match the figure.



24. To launch the **Flowline editor**, double-click the flowline.
25. Enter the flowline data shown in the figure.

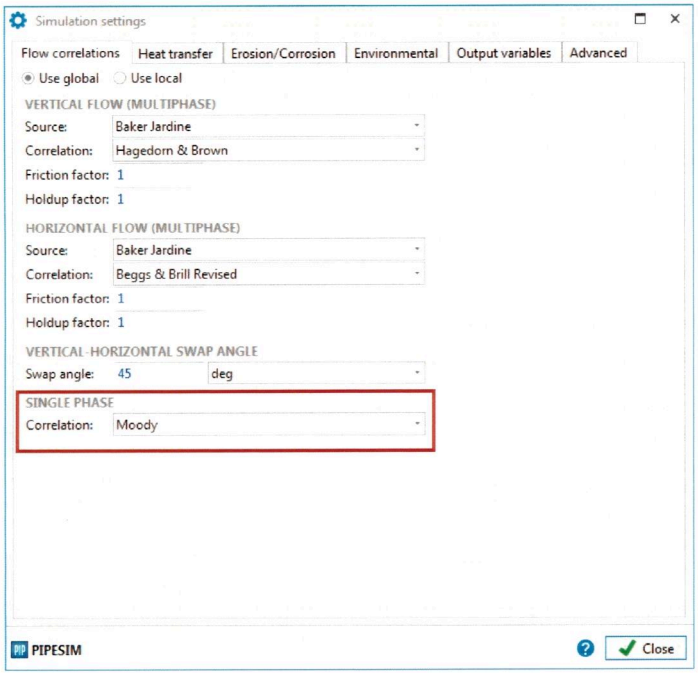
NOTE: An easy way to populate Pipe data is using the Flowline catalog. Click ... to select the desired flowline based on the ID, OD, WT, and roughness.



NOTE: The heat transfer coefficient is set to 0 to represent an insulated (isothermal) pipe and to ignore temperature changes for this simple example case. In practical situations, the heat transfer coefficient is always a positive value.

26. Click **Close** to exit the Flowline editor.
27. On the **Home** tab, select Simulation settings. The **Flow correlations** tab is now active.

28. For this model, an all water fluid, the only correlation to set is for Single phase. Select the Moody correlation.



29. Click **Close** to exit the **Simulation settings** dialog box.

NOTE: It is not critical for you to perform the steps in this exercise in the sequence shown. You could, for example, create the entire model before creating the fluid.

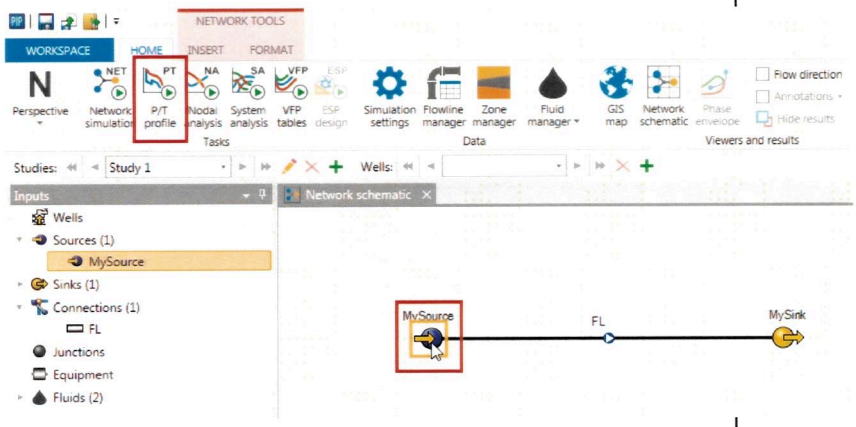
Exercise 3 Run the model

Now that you have created the model, you can perform the necessary calculations.

1. To activate all the possible tasks that can be run for the selected object, click MySource then click P/T profile on the **Home** tab.

TIP: You can select the P/T profile task from the **Tasks** pane.

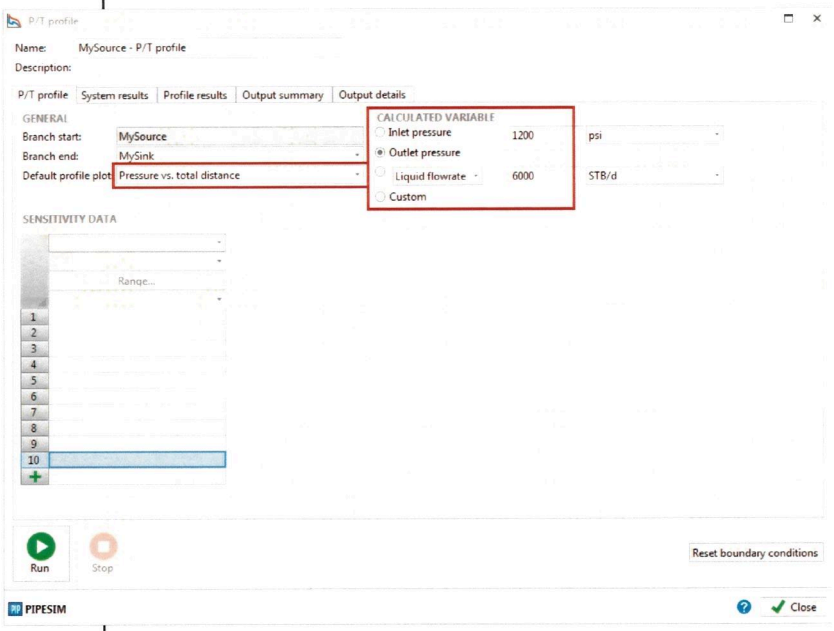
The Pressure/Temperature Profile task is used to model the distribution of pressure, temperature, and other parameters along the flow path.



NOTE: The Pressure Temperature Profile task requires you to designate a calculated variable and specify all other variables. Generally, two variable specifications are required out of the options (rate, inlet pressure, and outlet pressure), while the third is calculated. You can specify all three variables and a fourth variable can be calculated, for example, choke size.

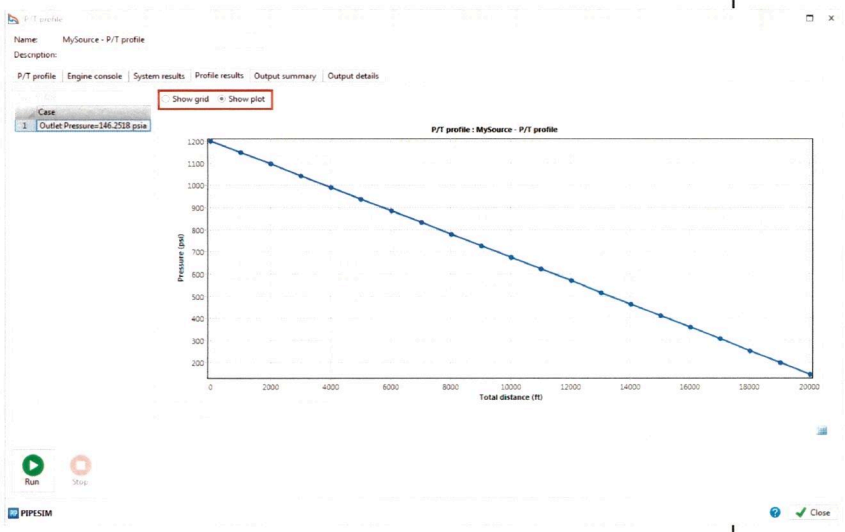
2. Designate Outlet pressure as the Calculated variable.
3. Enter values for the *Inlet pressure* and *Liquid flow rate*, as shown in the figure.

4. Set *Pressure vs. total distance* as the Default profile plot.



5. Click **Run** to start the simulation.

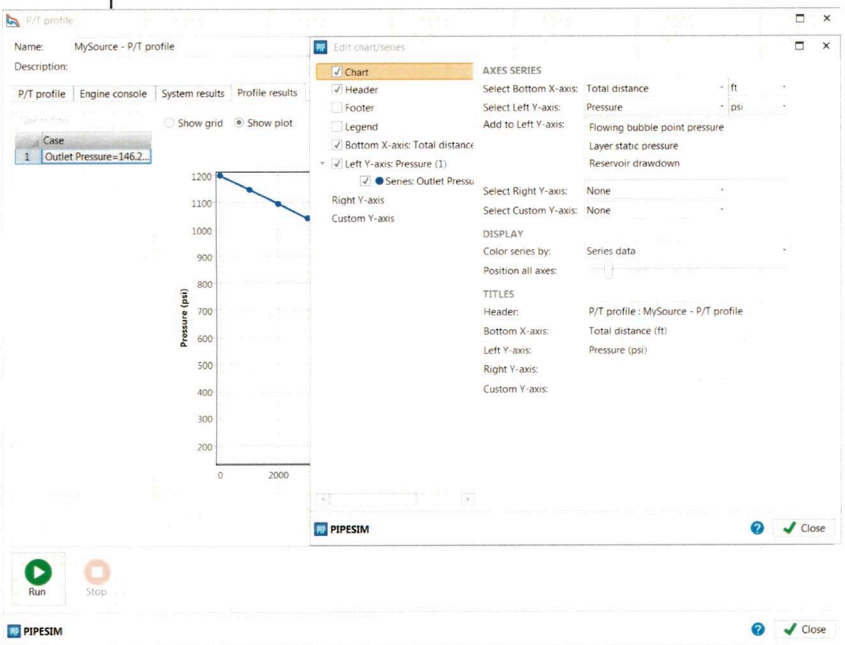
6. On the **System results** and **Profile results** tabs, switch between *Show grid* and *Show plot* to view the tabular and graphical results. All **PIPESIM** calculation results can be viewed here. `



NOTE: The **PIPESIM** output reports (Summary and Detailed) are not displayed by default as part of the simulation results. To display these reports, go to **Workspace > Options > Advanced**. Under **Result Display Options**, select the box *Show engine output files*. Re-run the simulation task. The reports display as additional tabs in the task window. To save the reports, right-click the report and select **Save as**.

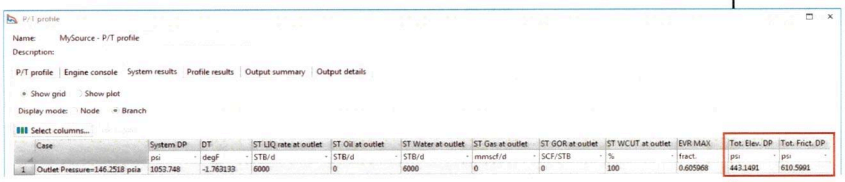
7. You can configure all aspects of the plot by double-clicking anywhere on the plot to display the dialog box shown in the figure.

TIP: To display the plot data in tabular format, click the icon in the lower right corner.



8. To display the various system and profile variables to complete the table that follows, configure the plot and grid views.
9. The liquid velocity and outlet pressure at the sink are relatively straightforward to retrieve from the plot or the grid view of the Profile results.
10. To display the pressure drops:.
    a. Click the **System Results** tab.
    b. Set the *Show Grid* option.
    c. Set the *Branch* display mode.
    d. Click the **Select Columns** button.
    e. Select *Total Elevation Pressure Drop*.

f. Select *Total Frictional Pressure Drop.*



NOTE: Profile results display as a function of distance along the flow path. To view minimum and maximum values or net changes in values for the overall system, it is much easier to view such results as System result variables.

11. Compare **PIPESIM** results to your hand calculations.

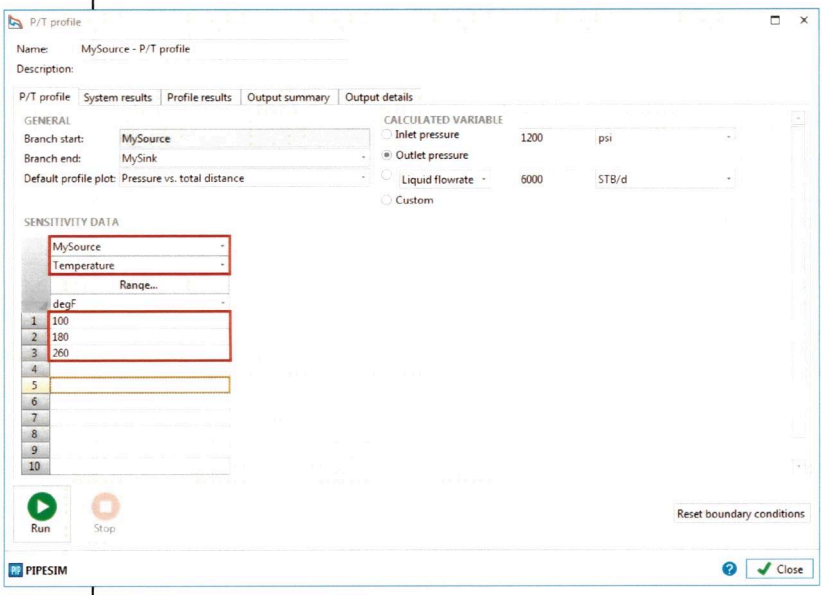
Results	Hand calculation	PIPESIM
Liquid Velocity (ft/s)		
$\Delta P_{frictional}$ (psi)		
$\Delta P_{elevational}$ (psi)		
ΔP_{total} (psi)		

Exercise 4 Analyze multiple scenarios with sensitivities

In this exercise, you use the model from the previous exercise to explore how it responds to different inlet temperatures.

1. In the **P/T profile** tab, modify the *Sensitivity data* section, as shown in the figure.

TIP: The **Range** button allows you to define a series of values for the selected sensitivity variable.

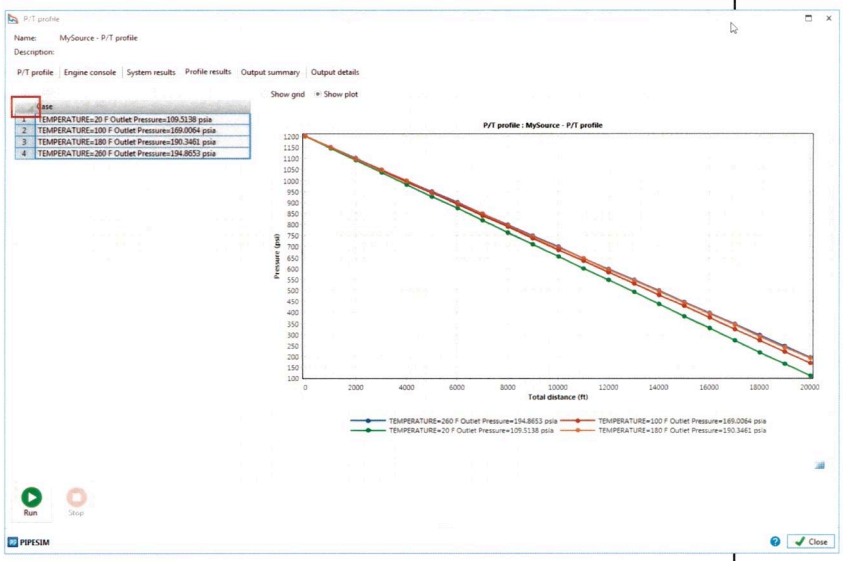


2. Click **Run** to launch the P/T profile task for all the sensitivity values.
3. Click the **Profile results** tab to view the plot.

   By default, all the sensitivity plots are displayed because all three rows in the *Case* section to the left of the plot are highlighted.

   - To display the cases, one at a time, click a single row.
   - To redisplay all three cases, click in the cell to the left of Case, as indicated in the figure.

NOTE: PIPESIM has a default setting to automatically display a maximum of 20 cases simultaneously for profile sensitivity plots. To change this setting, select **Workspace > Advanced > Result Display Options > Max. auto-selected profile results**.



Notice that the highest inlet temperature (260 degF) generates the lowest pressure drop. As the temperature increases, these parameters change:

- Viscosity decreases
- Reynold's number increases
- Corresponding friction factor decreases
- Frictional pressure gradient is lower

In other words,

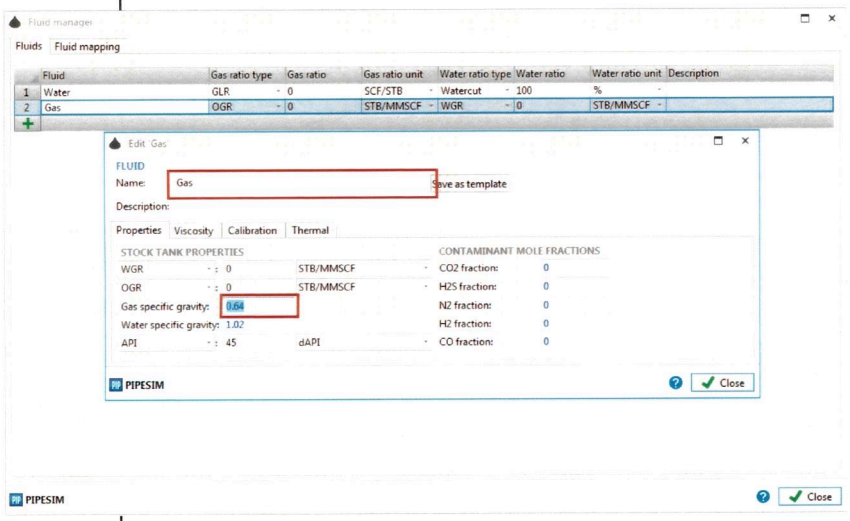
$$T\uparrow \Rightarrow \mu\downarrow \Rightarrow \mathrm{Re} = \frac{\rho v d}{\mu}\uparrow \Rightarrow f\downarrow \Rightarrow \left(\frac{dp}{dL}\right)_{friction}\downarrow$$

NOTE: For water, the effect of temperature on the density is negligible because water is essentially an incompressible fluid.

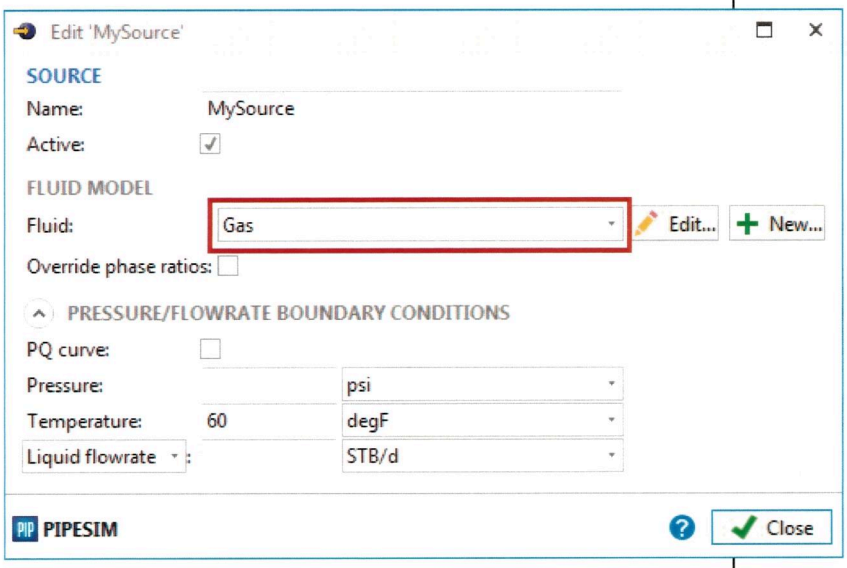
Exercise 5 Model a single-phase gas pipeline

In this exercise, you investigate the flow of a single phase gas without changing the physical components of the model from the previous exercise.

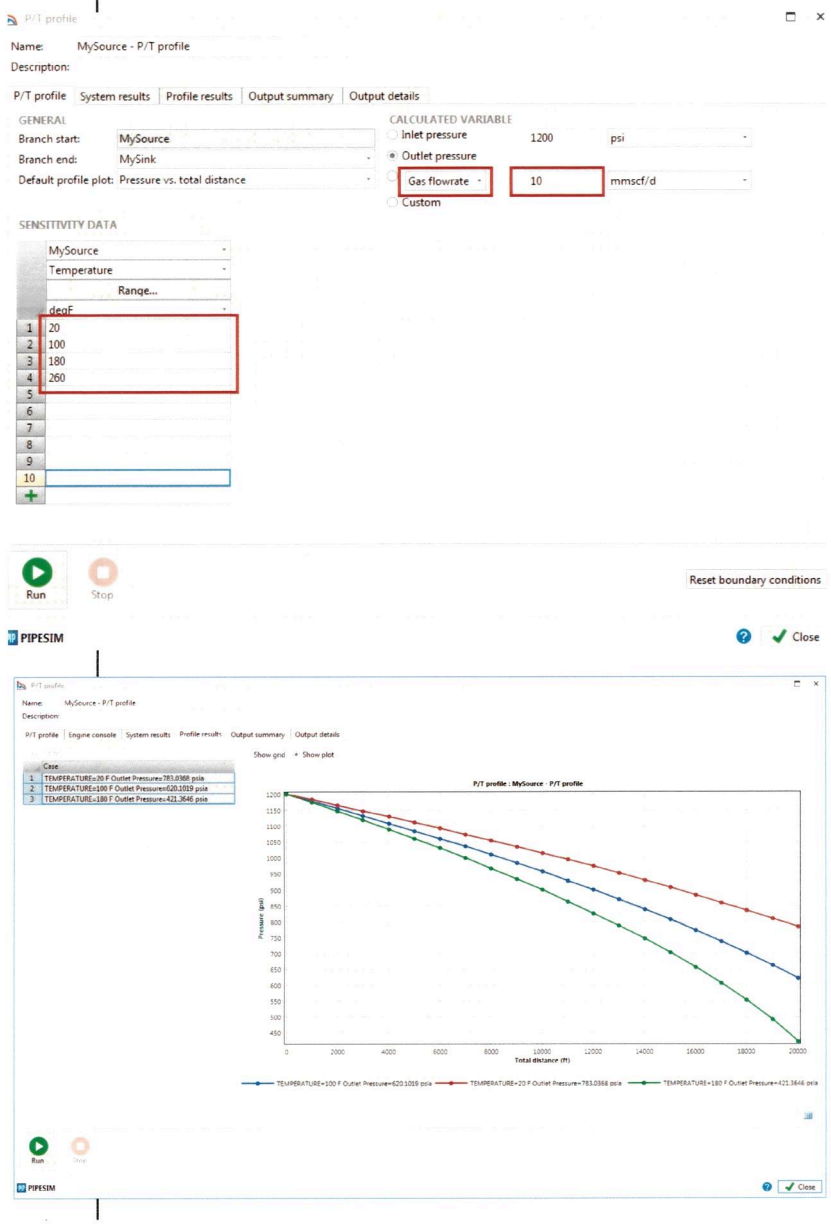
1. Launch the **Fluid manager** from the **Home** tab.
2. Create a new fluid using the Dry Gas template.
3. Edit the template to match the entries shown in the figure.



4. The created fluid can be mapped on the **Fluid mapping** tab of the **Fluid manager**. However, this time, associate the fluid with the source by using the **Edit** dialog box of the source.



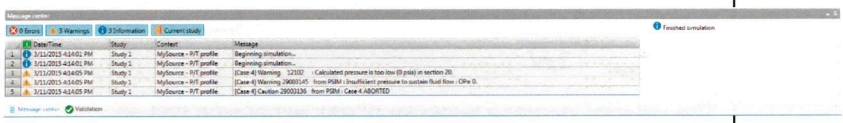
5. Re-run the P/T profile task for the updated temperatures, after reconfiguring it to use a Gas instead of Liquid flow rate, as shown in the figures.



6. Only three sensitivity cases are plotted. Which case is not plotted, and why not?

   Review the output reports and information in the **Message center** (usually located at the bottom of the **PIPESIM** window) to understand why.

   The 260 degF fails to run because the high temperature results in a high pressure drop, which ultimately results in the inability of the flow rate to be delivered to the sink because of insufficient pressure. The **Message center** shows you the status of the simulation and any errors or warnings.



In the example using water, the *lowest* temperature case generated the *highest* pressure loss. When the fluid was switched to gas in the model, the *highest* temperature case generated the *highest* pressure loss. In the water case, increased viscosity with lower temperature was the significant factor, while the density remained constant between the temperature cases. However, gas is a compressible fluid.

For example, gas density is described by the ideal gas law, rearranged into the expression:

$$\rho_g = \frac{pM}{zRT}$$

where:

r_g = gas density

p = pressure

M = molecular Weight

z = gas compressibility factor

R = Ideal gas constant

T = temperature.

In the *all gas* case, the highest inlet temperatures yield the highest pressure drop. This is because, as the temperature increases, the density decreases; this decrease results in a decrease in the Reynold's number.

Correspondingly, the friction factor increases and, as a result, the frictional pressure gradient is higher. In other words,

$$T\uparrow \gg r_g\downarrow \gg \mathrm{Re} = \frac{\rho v d}{\mu}\downarrow \gg f\uparrow \gg \mathrm{Re} = \frac{\rho v d}{\mu}\uparrow$$

and because

$$\left(\frac{dp}{dL}\right)_{friction} = \frac{f\rho v^2}{2gd}$$

The velocity increase because of gas expansion has an exponential effect on the frictional pressure term. This effect accounts for the increase in the frictional gradient along the flowline and the curvature in the pressure profile plot.



NOTE: The viscosity of the gas increases slightly with increasing temperature, but this effect is small and does little to offset the effects of decreasing density.



Exercise 6 Calculate gas pipeline flow capacity

Previously, the outlet pressure was calculated for a known inlet pressure and flow rate. In this exercise, you specify the inlet and outlet pressures and calculate the corresponding gas flow rate.

There are three key variables involved in P/T profile, Nodal analysis, and System analysis tasks:

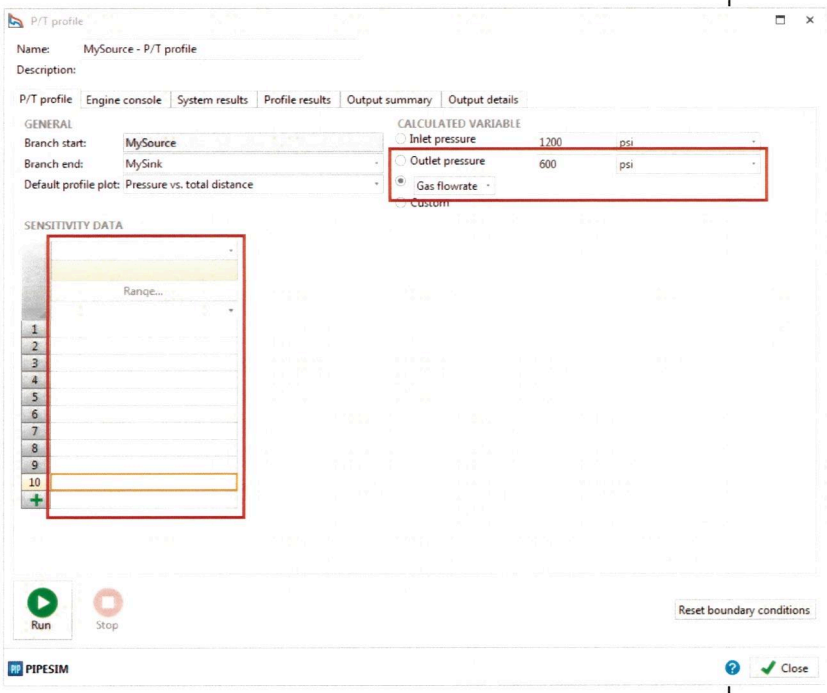
- Inlet pressure
- Outlet pressure
- Flow rate.

Two of these variables must be specified, and the third is calculated. Some tasks allow you to specify all three variables, in which case a matching variable, such as pump speed or choke setting, must be set as the calculated variable.

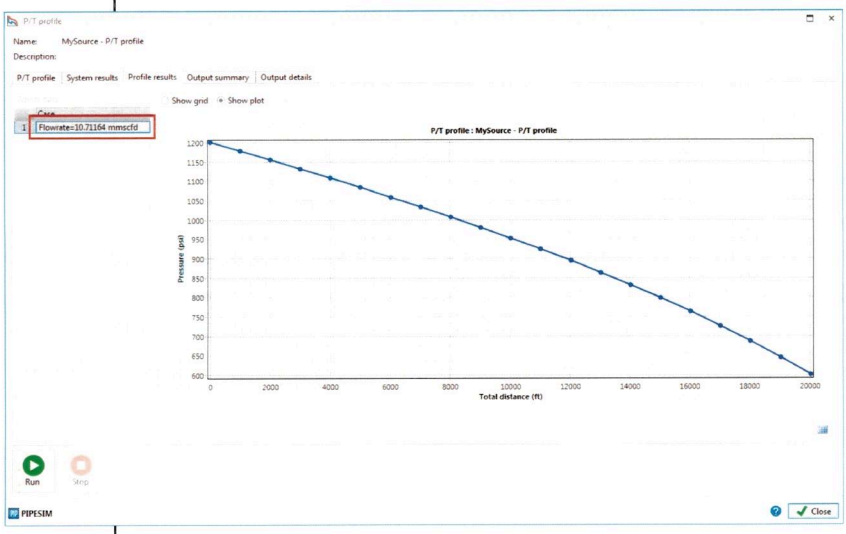
PIPESIM generally performs calculations in the direction of flow. When the outlet pressure is calculated, the solution is non-iterative in that the outlet pressure is calculated during the first and only pressure traverse calculation.

When outlet pressure is specified and inlet pressure or flow rate is calculated, the process becomes iterative and successive estimates of the calculated variable are supplied until the calculated outlet pressure agrees with the specified pressure.

1. Reconfigure the **P/T profile** dialog box. Set *Gas flow rate* as the Calculated variable with an Outlet pressure of 600 psia.
2. Highlight the cells in the *Sensitivity data* section and delete the values from the previous exercise.



3. Re-run the task and observe the flow rate that this pipeline can deliver under the specified conditions.



4. Calculated Gas flow rate_____ (mmscf/d)



Lesson 2 Multiphase flow calculations

While pressure losses for single-phase flow in pipes have long been accurately modeled with familiar expressions such as the Bernoulli equation, accurate predictions of pressure loss in two-phase flow have proved to be more challenging because of added complexities.

The lower density and viscosity of the gas phase causes it to flow at a higher velocity relative to the liquid phase, a characteristic known as *slippage*. This slippage results in frictional pressure losses associated with shear stresses encountered at the gas/liquid interface, as well as along the pipe wall. Additionally, the highly compressible gas phase expands as the pressure decreases along the flow path.

Further complicating matters are the variety of physical phase distributions, termed flow *regimes* or flow *patterns* (Figure 12 and Figure 13), that impact the pressure losses in the flow path.

The prevailing flow pattern for a specific set of conditions depends on the relative magnitude of the forces acting on the fluids.

Buoyancy, turbulence, inertia, and surface-tension forces are greatly affected by the relative flow rates, viscosities, and densities of the fluids, as well as the pipe diameter and inclination angle. The complex dynamics of the flow pattern govern slippage effects and, therefore, variations in liquid holdup and pressure gradient.

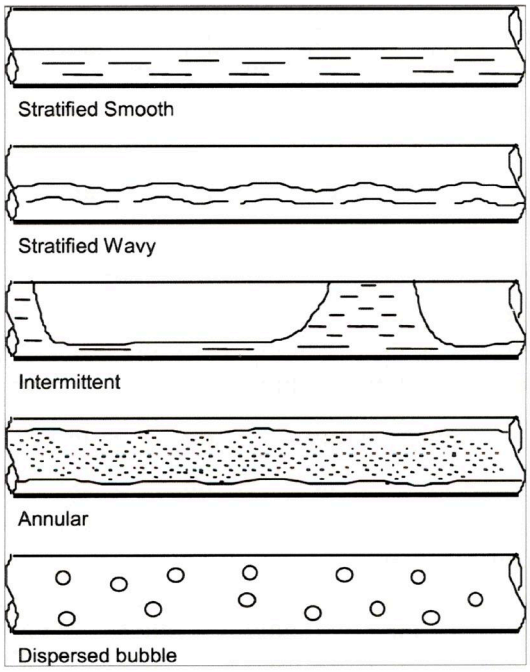


Figure 12 Multiphase flow regimes for horizontal flow

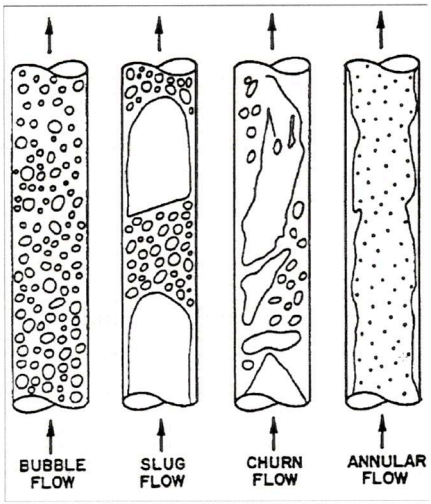


Figure 13 Multiphase flow regimes for vertical flow

Many empirical correlations and mechanistic models have been proposed to predict liquid holdup and pressure loss. (Refer to the **PIPESIM** help system for details.)

Some correlations and models are general, while others apply only to a narrow range of conditions. Many of these approaches begin with a prediction of the flow pattern, with each flow pattern having an associated method of predicting liquid holdup and frictional pressure loss.

In steady-state flow, the gas usually travels faster than the liquid, so it slips past the liquid. For the volume flow rate to remain constant, the area of the pipe occupied by the gas must shrink. This condition gives rise to a higher liquid volume fraction than if the gas traveled at the same velocity, resulting in *liquid holdup*, as illustrated in Figure 14.

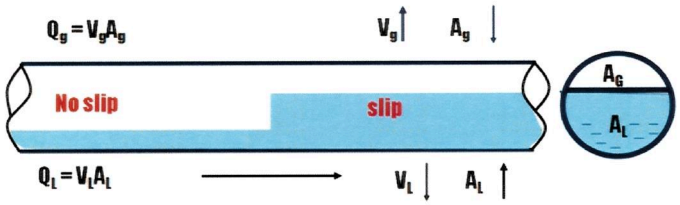


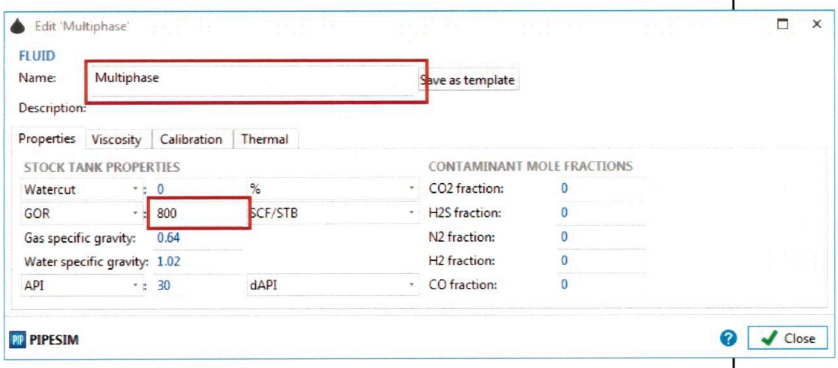
Figure 14 Liquid Holdup

Liquid holdup is generally the most important parameter in calculating pressure loss. Quantifying liquid holdup is necessary to predict hydrate formation and wax deposition and to estimate the liquid volume expelled during pigging operations for sizing slug catchers. The liquid holdup prediction is used to determine a two-phase friction factor from which a pressure gradient is calculated.

Exercise 1 Model a multiphase pipeline

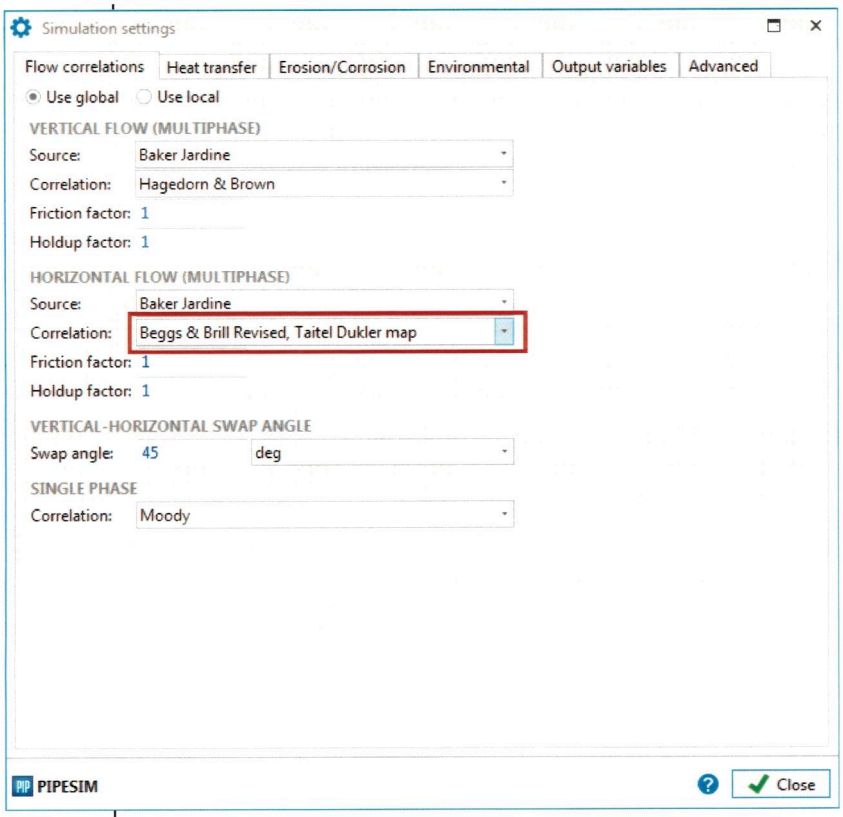
The previous exercises explored single-phase flow of water and gas through a pipeline. In this exercise, you modify the existing pipeline model and explore multiphase flow.

1. Use the **Fluid Manager** on the **Home** tab to create a multiphase fluid with the properties shown in the figure. (Do not use a fluid template for this case.)

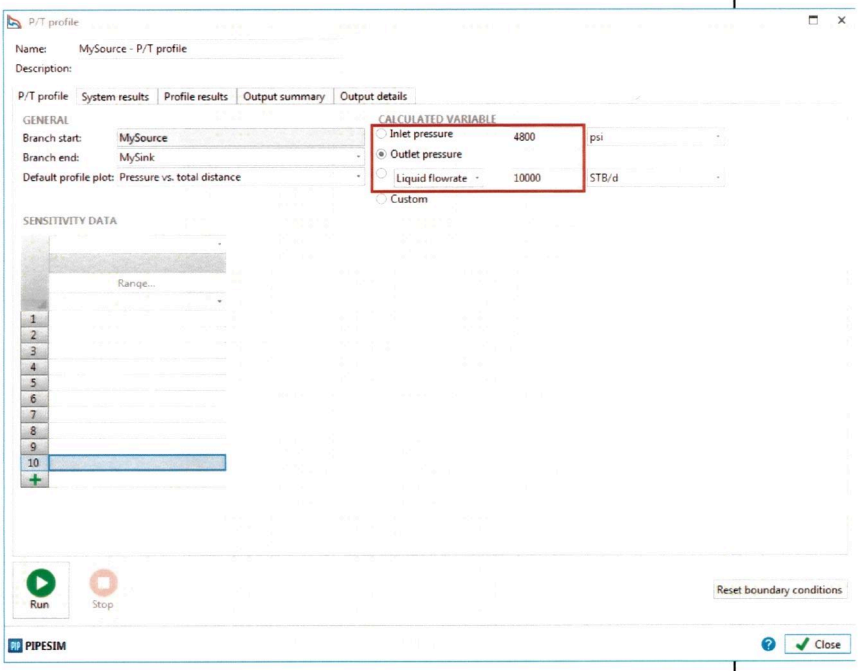


2. Click the **Fluid mapping** tab and map the new Multiphase fluid to the source.
3. From the **Home** tab, select **Simulation settings**.
4. Click the **Flow correlations** tab and configure the choices for vertical and horizontal flow, as shown in the figure.

NOTE: Observe that the Swap angle is set to 45 degrees. This angle corresponds to the switch between using the vertical and horizontal flow correlations. The swap angle in **PIPESIM** is referenced from the horizontal x-axis. In this example, the pipeline inclination angle is about 3 degrees (that means only the horizontal flow correlation is used).


Simulation settings
Flow correlations
Heat transfer
Erosion/Corrosion
Environmental
Output variables
Advanced
Use global
Use local
VERTICAL FLOW (MULTIPHASE)
Source: Baker Jardine
Correlation: Hagedorn & Brown
Friction factor: 1
Holdup factor: 1
HORIZONTAL FLOW (MULTIPHASE)
Source: Baker Jardine
Correlation: Beggs & Brill Revised, Taitel Dukler map
Friction factor: 1
Holdup factor: 1
VERTICAL-HORIZONTAL SWAP ANGLE
Swap angle: 45 deg
SINGLE PHASE
Correlation: Moody
PIPESIM
Close

5. Reconfigure the **P/T profile** dialog box, as shown in the figure.



6. Run the model.
7. From the **Profile Results** tab, click *Show Grid*.
8. Look at the *Liquid Holdup* or *Flow Pattern* column. Notice that the flow is initially single-phase liquid until the pressure falls below the bubblepoint, at which point a two-phase oil-gas mixture evolves.
9. The single-phase Moody correlation is used in the first part of the pipe, and the Beggs and Brill multiphase correlation is used in the second part of the pipe after the pressure falls below the bubble point.

P/T profile | Engine console | System results | Profile results | Output summary | Output details

Case

1 Outlet Pressure=1237.482 psia

Show grid Show plot

Type filter: All

Select columns... Expand all

	Case	Equipment	Type	Total distance	Elevation	Pressure	Temperature	Fluid mean...	Liquid holdup	EVR	G-L Pattern	O-W Pattern
				ft	ft	psi	degF	ft/s	%			
1	Outlet Pressure=1237.482 psia	FL	Flowline	0	0	4800	60	16.70284	100	1.141815	Liquid	Undefined
2	Outlet Pressure=1237.482 psia		Flowline	1001.249	50	4657.942	61.05692	16.72333	100	1.142515	Liquid	Undefined
3	Outlet Pressure=1237.482 psia		Flowline	2002.498	100	4516.066	62.11313	16.74475	100	1.143246	Liquid	Undefined
4	Outlet Pressure=1237.482 psia		Flowline	3003.748	150	4374.358	63.16872	16.76719	100	1.144012	Liquid	Undefined
5	Outlet Pressure=1237.482 psia		Flowline	4004.997	200	4232.806	64.22381	16.79073	100	1.144815	Liquid	Undefined
6	Outlet Pressure=1237.482 psia		Flowline	5006.246	250	4091.396	65.27849	16.81551	100	1.145659	Liquid	Undefined
7	Outlet Pressure=1237.482 psia		Flowline	6007.495	300	3950.113	66.3329	16.84166	100	1.14655	Liquid	Undefined
8	Outlet Pressure=1237.482 psia		Flowline	7008.745	350	3808.941	67.38716	16.86932	100	1.147491	Liquid	Undefined
9	Outlet Pressure=1237.482 psia		Flowline	8009.994	400	3667.86	68.44142	16.8987	100	1.14849	Liquid	Undefined
10	Outlet Pressure=1237.482 psia		Flowline	9011.243	450	3526.852	69.49584	16.92998	100	1.149552	Liquid	Undefined
11	Outlet Pressure=1237.482 psia		Flowline	10012.49	500	3385.98	70.27883	16.97163	99.62114	1.150965	Liquid	Undefined
12	Outlet Pressure=1237.482 psia		Flowline	11013.74	550	3239.844	69.77631	17.0687	97.34495	1.154252	Dispersed Bubble	Undefined
13	Outlet Pressure=1237.482 psia		Flowline	12014.99	600	3083.936	69.29066	17.19805	94.90344	1.158618	Intermittent	Undefined
14	Outlet Pressure=1237.482 psia		Flowline	13016.24	650	2916.842	68.82136	17.37253	92.24226	1.16448	Intermittent	Undefined
15	Outlet Pressure=1237.482 psia		Flowline	14017.49	700	2735.989	68.365	17.6135	89.26818	1.173528	Intermittent	Undefined
16	Outlet Pressure=1237.482 psia		Flowline	15018.74	750	2537.458	67.91562	17.95819	85.83018	1.183946	Intermittent	Undefined
17	Outlet Pressure=1237.482 psia		Flowline	16019.99	800	2316.021	67.46481	18.47451	81.68996	1.200845	Intermittent	Undefined
18	Outlet Pressure=1237.482 psia		Flowline	17021.24	850	2080.189	67.01947	19.2362	76.8013	1.22535	Intermittent	Undefined
19	Outlet Pressure=1237.482 psia		Flowline	18022.49	900	1828.674	66.56608	20.39842	70.88205	1.261824	Intermittent	Undefined
20	Outlet Pressure=1237.482 psia		Flowline	19023.74	950	1553.275	66.07481	22.31203	63.37056	1.319684	Intermittent	Undefined
21	Outlet Pressure=1237.482 psia		Flowline	20024.98	1000	1237.482	65.48603	25.9284	53.22471	1.422618	Intermittent	Undefined

Review questions



- Which types of pressure drop does **PIPESIM** report?
- What is the default single-phase flow correlation in **PIPESIM**?
- How do you describe a Black Oil fluid model for water or dry gas?
- Did you observe any difference in pressure drop between your hand calculations and the **PIPESIM** reported results? If yes, why?

Summary

In this module, you learned about:

- building the physical model
- creating a fluid model
- choosing flow correlations
- performing operations
- viewing and analyzing results

NOTES

Module 3 Oil well performance analysis

This module examines a producing oil well located in the North Sea. You analyze the performance of this well using Nodal analysis, calibrate the black oil fluid using laboratory data, and match IPR data.

You also analyze the behavior of the well with increasing water cut and evaluate the options to implement gas lift or install an ESP at a later stage when the well is unable to flow naturally.

Learning objectives



After completing this module, you will know how to:

- perform Nodal analysis
- estimate bottomhole flowing conditions
- calibrate black oil pressure, volume, and temperature (PVT) data
- perform inflow performance relationship (IPR) matching
- conduct water cut sensitivity analysis
- evaluate gas lift performance
- design and install an ESP
- model multiple completions
- install a downhole choke

Lesson 1 Nodal analysis



Nodal analysis is used to evaluate the performance of a production or injection well. It involves specifying a nodal point (solution node), usually at the bottomhole or wellhead, that divides the producing system into two parts: the inflow and the outflow. This relationship is represented graphically in Figure 15.

The *solution node* is defined as the location where the pressure differential upstream (inflow) and downstream (outflow) of the node is zero.

Solution nodes can be judiciously selected to isolate the effect of certain variables.

For example, if the node is taken at the bottomhole, factors that affect the inflow performance of a production well, such as skin factor, can be analyzed independently of variables that affect the outflow (for example, tubing diameter or separator pressure).

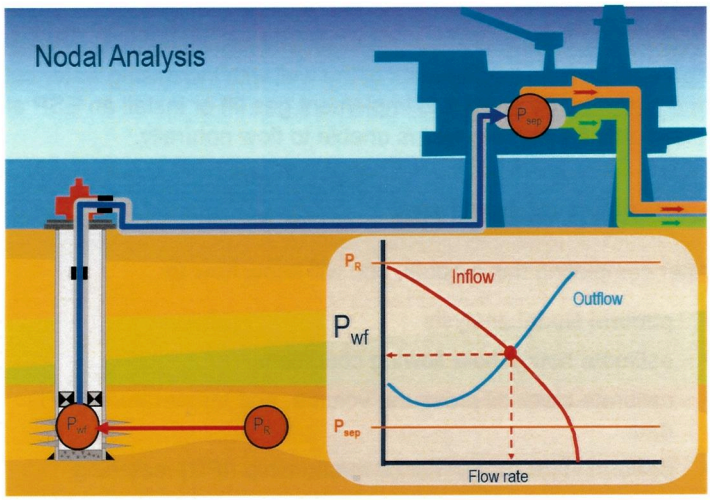


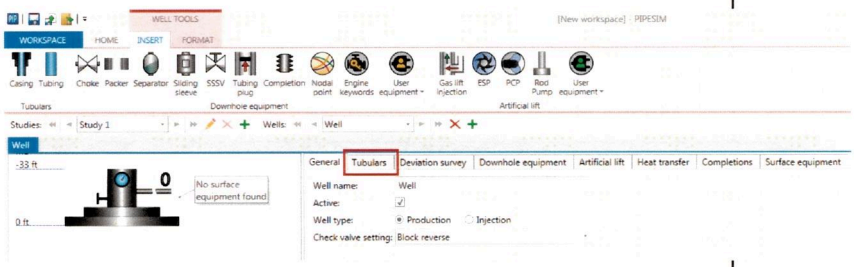
Figure 15 Intersection points of the inflow and outflow performance curves

Exercise 1 Build the well model

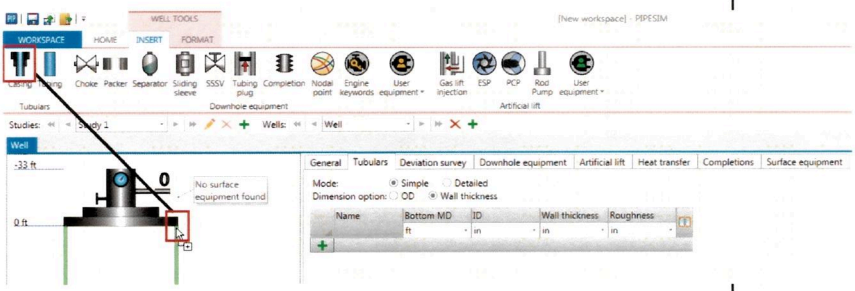
In this exercise, you build a model of an oil production well.

This exercise could be done using the Network-centric workspace of the previous exercises. However, in this instance, the exercise uses the well-centric mode instead.

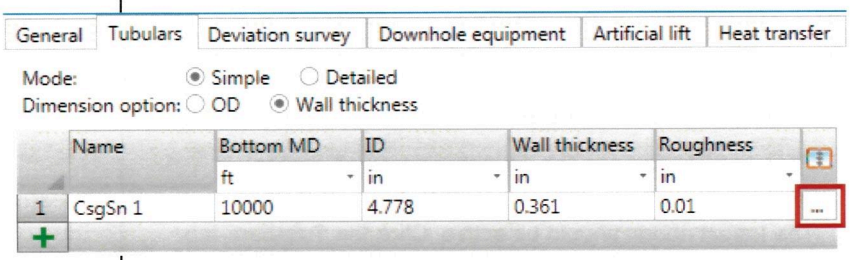
1. From the **Workspace** tab, launch a new well-centric workspace.
2. The **Insert** ribbon should be active. Click the **Tubulars** tab and leave the default options: *Simple Mode* and *Wall thickness Dimension.*.



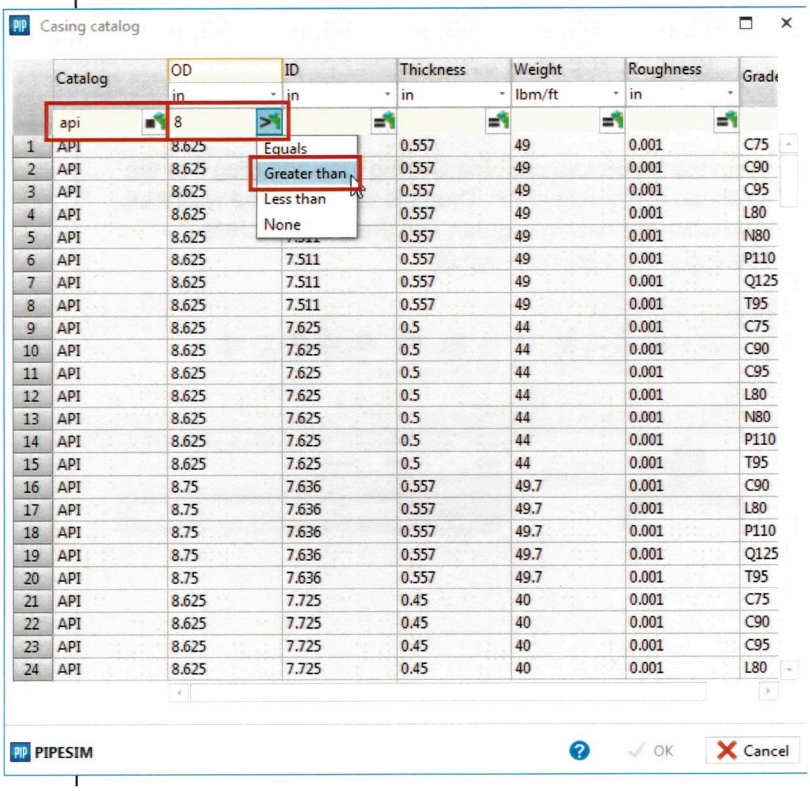
3. From the **Insert** tab, click the **Casing** icon and drag it to the wellhead in the schematic. Drop the casing on the wellhead, only when the casing is green and the green circle is flashing, as shown in the figure.



4. On the **Tubulars** tab of the well editor, click the ... button on the far right to launch the Casing Catalog.



5. In the *Catalog* column, enter `API` to filter the catalog to display only API-type casings.
6. Use the filter on the *OD* (Outside Diameter) column to filter the list to display only casings with an internal diameter greater than 8 inches.

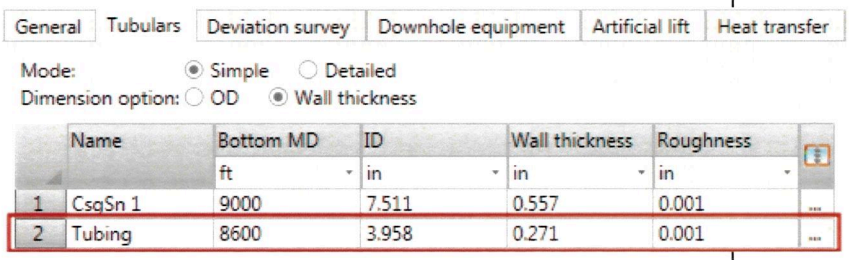


7. Select the first L80 grade of casing in the filtered list (8.625" OD, 7.511" ID, Weight = 49 lbm/ft) by selecting the corresponding row, then click **OK**.
8. Set the MD of the casing to 9,000 ft.
9. Add a tubing string by clicking the (**+**) sign in the Tubulars table.

NOTE: You can add tubing by dragging from the ribbon toolbar as well.



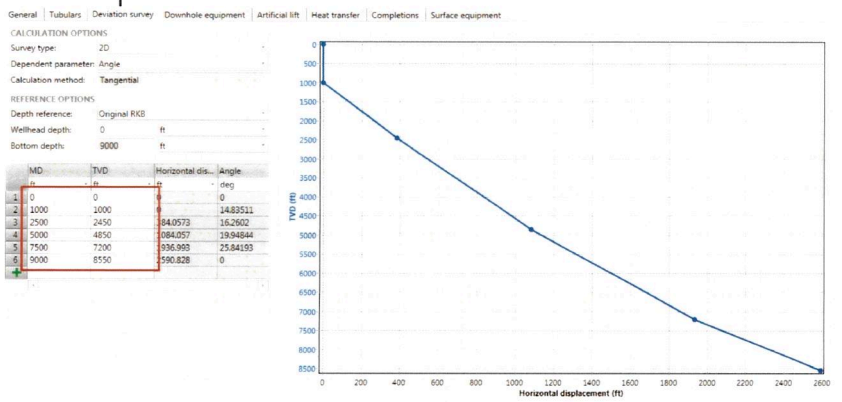
10. Specify the parameters shown in the figure. Instead of using the catalog, enter the numbers directly.



11. On the **Deviation survey** tab, change the Survey type to *2D* and leave the default Dependent parameter as *Angle*.

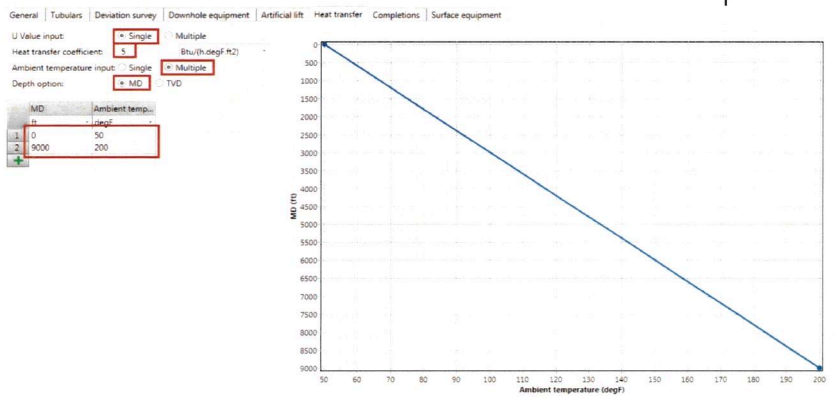
12. Enter the *MD*/*TVD* columns for a 2D survey, as shown in the figure. The preview plot automatically updates to display the deviation survey profile.

NOTE: The white background indicates editable fields. The shaded background indicate read-only fields that are automatically calculated. For the deviation survey grid, the read-only fields are calculated based on the selected Dependent parameter.



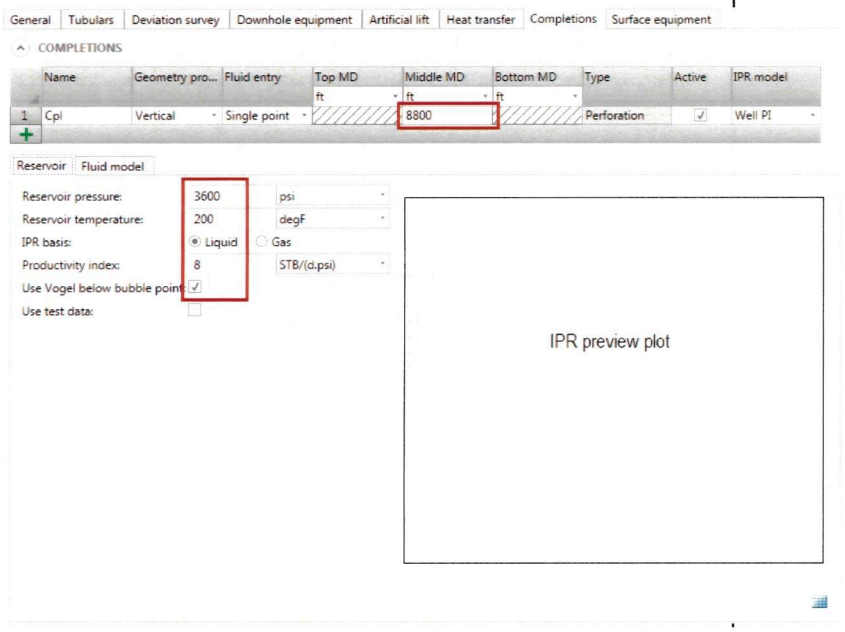
NOTE: To copy any tabular data (such as deviation survey data) from Excel or Word documents and paste them in a **PIPESIM** grid, copy the data from the external source, click in the first cell in the first column of the **PIPESIM** grid, and press Ctrl+V.

13. Configure the parameters in the **Heat transfer** tab, as shown in the figure.



14. On the **Completions** tab, add a completion to the well and configure, as shown.

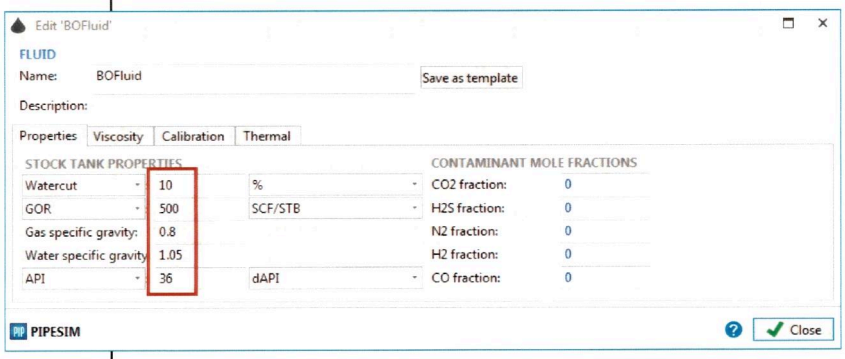
NOTE: The IPR preview plot in the **Completions** tab of the **Well editor** is blank if no fluid is defined.



15. On the **Completions** tab, click the **Fluid model** tab and click **New** to create a new fluid for the completion.

WARNING: Do not use a fluid template and leave the defaults for all the other tabs.

After you create the fluid and return to the **Reservoir** tab of the main **Completions** tab, the IPR preview plot appears.

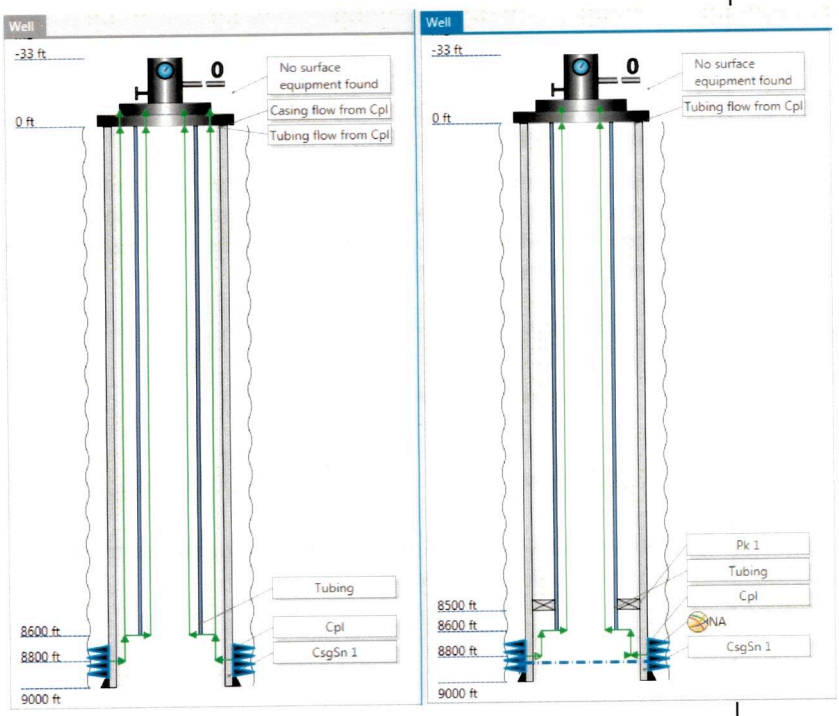


16. The dynamic well schematic diagram on the left represents the wellbore configuration up to this point. The green lines represent fluid flow paths. The current well configuration (on the left) indicates a dual flow path that supports the simultaneous flow of fluid up the tubing and annulus.

    The fluid flow must be restricted to the tubing only, by adding a packer as in the well schematic on the right.

17. On the **Downhole equipment** tab, add a Packer at 8,500 ft to prevent flow up the annulus between the tubing and casing.

    There is no packer shown in the flow path on the left; a single flow path is shown with a packer.

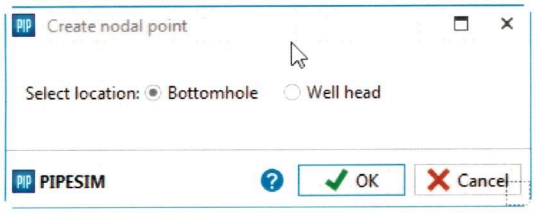


18. Make sure that the Hagedorn-Brown correlation is selected for vertical flow and the Beggs-Brill Revised correlation is selected for horizontal flow under **Home > Simulation settings > Flow correlations**.

Exercise 2 Perform a nodal analysis

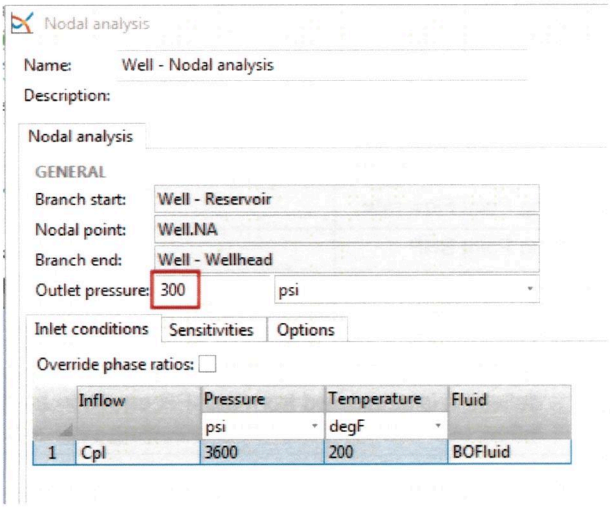
In this exercise, you perform a Nodal analysis task for a given outlet (wellhead) pressure to determine the operating point (intersection of the inflow and outflow curves) and the absolute open flow potential (AOFP) of the well.

1. On the **Home** tab, click the **Nodal analysis** button. A dialog box appears, prompting you to choose the location of the Nodal Analysis point. The Nodal analysis point divides the system into *inflow* and *outflow*.
2. Select *Bottomhole*. The *inflow* in this case describes the pressure-rate (P-Q) relationship from the reservoir to the bottomhole, while the *outflow* represents the P-Q relationship from the bottomhole to the wellhead.

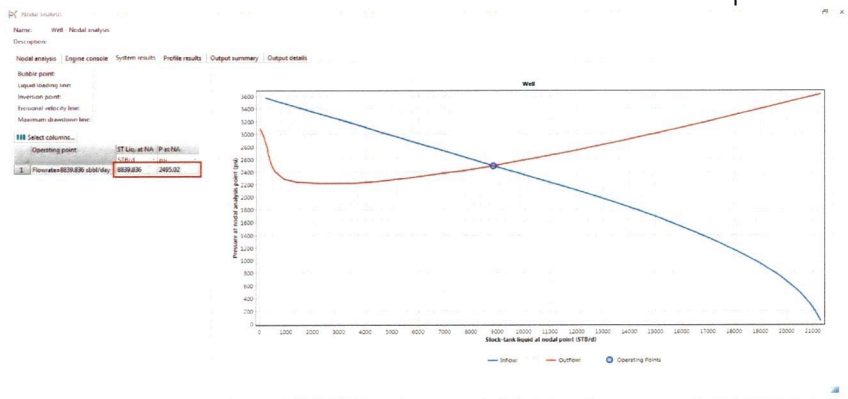


3. Set the Outlet pressure as 300 psi and rename the case to `Oil Well Nodal`, if desired.

4. Click the **Run** button. If the **Run** button is inactive, check the **Message center** to see if there are any validation issues and resolve them.



5. The Nodal Analysis system plot appears.



6. The inflow performance relationship (IPR) curve represents the P-Q relationship for the reservoir/completion, modeled with the Well PI equation in this case. The outflow represents the P-Q relationship for multiphase flow through the 4-12 inch tubing, modeled using the Hagedorn & Brown correlation in this case.

7. The graphical intersection of both curves is known as the *Solution/Operating point*. This point represents the expected rate[1] and flowing bottomhole pressure from a well, completed with the tubing described by the outflow and in the reservoir/completion described by the inflow. These options are discussed in more detail later.
8. Review the **System results** and complete the Results table.

Results	
Operating Point Flow rate	
Operating Point BHP	
AOFP	



NOTE: The Absolute Open Hole Flow Potential (AOFP) is the theoretical maximum a reservoir/completion can deliver. It is calculated from the IPR equation as the inflow rate that corresponds to a flowing BHP = 0.

9. To extract the AOFP, click the icon at the bottom right corner of the System results plot (or in the IPR preview plot in the **Completions** tab of the well editor).
10. To observe the Pressure profile corresponding to the operating point, click the **Profile Results** tab.

1. This predicted rate is an estimate that has inherent uncertainty, based on limitations with the data quality and the models used, particularly the multiphase flow correlations. The better the data and model quality, the better the predicted rates and flowing pressures. **PIPESIM** provides the option to tune PVT correlations to match lab measured data, as well as the option to calibrate multiphase flow correlations with field measurements.

Lesson 2 Pressure/temperature profile



While the Nodal analysis operation produces profile plots for each operating point, you also can generate these plots using the Pressure/Temperature (P/T) Profile task. One advantage of the P/T Profile task is that you can calculate boundary pressures by supplying a rate.

Exercise 1 Generate a pressure/temperature profile



Inlet and outlet pressure always reference the boundaries of the system. In this case, the inlet pressure is the reservoir pressure, while the outlet pressure corresponds to the wellhead pressure. The inlet pressure is specified at the completion or source level, while the outlet pressure is always specified manually in the task.

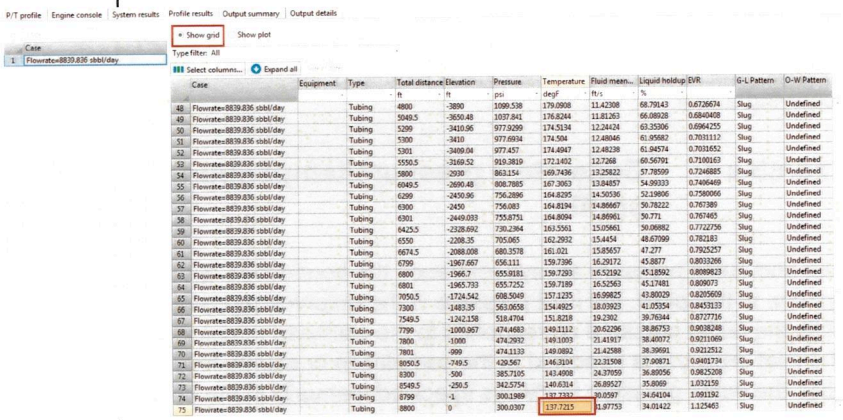
1. Launch the P/T profile task from the **Home** tab and specify *Liquid flow rate* as the Calculated variable.
2. Enter the Outlet (Tubing head) Pressure of `300` psi.
3. Leave Sensitivity data empty and leave the Default profile plot set to *Elevation vs. Pressure*.
4. Run the model.
5. Review the **Profile results** tab and complete the Results table. They should exactly match the results from the previous exercise because the boundary conditions are identical.



NOTE: To see profile results at smaller, discrete intervals, go to **Home > Simulation settings > Advanced** and select the *Print computation segment result.* Scenario: Determine a more accurate depth or distance at which the bubble point is reached.

Results	
Wellhead pressure	300 psia
Production Rate	
Flowing BHP	
Flowing WHT	
Depth at which gas appears	

6. (Optional) Select *Show grid* to display the results of all variables calculated during the task, for example wellhead temperature (WHT).



	Case	Equipment	Type	Total distance	Elevation	Pressure	Temperature	Fluid mean...	Liquid holdup	EVR	G-L Pattern	O-W Pattern
				ft	ft	psi	degF	ft/s	%			
48	Flowrate=8839.836 sbbl/day		Tubing	4800	-3890	1099.538	179.0908	11.42308	68.79143	0.6726674	Slug	Undefined
49	Flowrate=8839.836 sbbl/day		Tubing	5049.5	-3650.48	1037.841	176.8244	11.81263	66.08928	0.6840408	Slug	Undefined
50	Flowrate=8839.836 sbbl/day		Tubing	5299	-3410.96	977.9299	174.5134	12.24424	63.35306	0.6964255	Slug	Undefined
51	Flowrate=8839.836 sbbl/day		Tubing	5300	-3410	977.8934	174.504	12.48046	61.95682	0.7031112	Slug	Undefined
52	Flowrate=8839.836 sbbl/day		Tubing	5301	-3409.04	977.457	174.4947	12.48238	61.94574	0.7031652	Slug	Undefined
52	Flowrate=8839.836 sbbl/day		Tubing	5550.5	-3169.52	910.3819	172.1402	12.7268	60.56791	0.7160163	Slug	Undefined
54	Flowrate=8839.836 sbbl/day		Tubing	5800	-2930	863.154	169.7436	13.25822	57.78599	0.7246885	Slug	Undefined
55	Flowrate=8839.836 sbbl/day		Tubing	6049.5	-2690.48	808.7885	167.3063	13.84857	54.99333	0.7406469	Slug	Undefined
56	Flowrate=8839.836 sbbl/day		Tubing	6299	-2450.96	756.2896	164.8295	14.50536	52.19806	0.7580066	Slug	Undefined
57	Flowrate=8839.836 sbbl/day		Tubing	6300	-2450	756.083	164.8194	14.88667	50.78222	0.767389	Slug	Undefined
58	Flowrate=8839.836 sbbl/day		Tubing	6301	-2449.033	755.8751	164.8094	14.86961	50.771	0.767465	Slug	Undefined
59	Flowrate=8839.836 sbbl/day		Tubing	6425.5	-2328.692	730.2964	163.5561	15.05661	50.06882	0.7722756	Slug	Undefined
60	Flowrate=8839.836 sbbl/day		Tubing	6550	-2208.35	705.065	162.2932	15.4454	48.67099	0.782183	Slug	Undefined
61	Flowrate=8839.836 sbbl/day		Tubing	6674.5	-2088.008	680.3578	161.021	15.85657	47.277	0.7925257	Slug	Undefined
62	Flowrate=8839.836 sbbl/day		Tubing	6799	-1967.667	656.111	159.7396	16.29172	45.8877	0.8033266	Slug	Undefined
63	Flowrate=8839.836 sbbl/day		Tubing	6800	-1966.7	655.9181	159.7293	16.52192	45.18592	0.8089823	Slug	Undefined
64	Flowrate=8839.836 sbbl/day		Tubing	6801	-1965.733	655.7252	159.7189	16.52563	45.17481	0.809073	Slug	Undefined
65	Flowrate=8839.836 sbbl/day		Tubing	7050.5	-1724.542	608.5049	157.1235	16.99825	43.80029	0.8205609	Slug	Undefined
66	Flowrate=8839.836 sbbl/day		Tubing	7300	-1483.35	563.0658	154.4925	18.03923	41.05354	0.8453133	Slug	Undefined
67	Flowrate=8839.836 sbbl/day		Tubing	7549.5	-1242.158	518.4704	151.8218	19.2302	39.76344	0.8727716	Slug	Undefined
68	Flowrate=8839.836 sbbl/day		Tubing	7799	-1000.967	474.4683	149.1112	20.62296	38.86753	0.9038248	Slug	Undefined
69	Flowrate=8839.836 sbbl/day		Tubing	7800	-1000	474.2932	149.1003	21.41917	38.40072	0.9211069	Slug	Undefined
70	Flowrate=8839.836 sbbl/day		Tubing	7801	-999	474.1133	149.0892	21.42588	38.39691	0.9212512	Slug	Undefined
71	Flowrate=8839.836 sbbl/day		Tubing	8050.5	-749.5	429.567	146.3104	22.31508	37.90871	0.9401734	Slug	Undefined
72	Flowrate=8839.836 sbbl/day		Tubing	8300	-500	385.7195	143.4908	24.37059	36.80056	0.9825208	Slug	Undefined
73	Flowrate=8839.836 sbbl/day		Tubing	8549.5	-250.5	342.5754	140.6314	26.89527	35.8069	1.032159	Slug	Undefined
74	Flowrate=8839.836 sbbl/day		Tubing	8799	-1	300.1989	-137.7332	30.0597	34.64104	1.091192	Slug	Undefined
75	Flowrate=8839.836 sbbl/day		Tubing	8800	0	300.0307	137.7215	31.87753	34.01422	1.125463	Slug	Undefined

NOTE: The icon flips any plot to display a grid of only the data used for the plot.

Lesson 3 Black oil fluid calibration

Black oil fluid properties (also known as PVT properties) are predicted by correlations developed by fitting experimental fluid data with mathematical models. Various correlations have been developed over the years based on experimental datasets covering a range of fluid properties.

The **PIPESIM** help system describes the range of fluid properties used to develop each correlation, which helps you select the most appropriate correlation for the fluid you want to model. The default correlations in **PIPESIM** are based on the overall accuracy of the correlations as applied to a broad range of fluids.

PIPESIM provides functionality to match PVT fluid properties with laboratory data to increase the accuracy of fluid property calculations and improve overall model predictability, particularly over the range of pressures and temperatures, for the system being modeled.

For example, calibration of the bubble point pressure can result in the initial appearance of gas at a depth of 1,000 feet higher or lower than an uncalibrated model. This results in a significantly different mixture fluid density and, thus, a vastly different elevational pressure gradient.

Likewise, calibration of the fluid viscosity can drastically improve the calculation of the frictional pressure gradient, especia!ly in heavy oils and emulsions.

If the calibration data is omitted, **PIPESIM** calibrates with only oil and gas gravity. After the calibration is performed, a calibration factor is calculated as the ratio of measured value to the value calculated by the selected correlation.

The calibration method available in **PIPESIM 2014.1** is Single Point Calibration.

Single point calibration

In many cases, actual measured values for some properties show a slight variance from calculated values. When this variance occurs, it is useful to calibrate the property using the measured point. **PIPESIM** can use the known data for the property to calculate a calibration constant Kc:

Kc = Measured Property @(P,T)/Calculated Property @(P,T)

This calibration constant is used to modify all subsequent calculations of the property in question:

Calibrated value = Kc * (Predicted value)

Save fluids and wells as templates

PIPESIM lets you save fluids as templates that can be easily reused for other wells and sources. Click *Save as template* next to the fluid name in the fluid editor, or right-click the fluid in the **Inputs** pane and select *Save as template.*

The fluid is saved in the fluid templates catalog, accessible under *Home* > *Catalogs*. All the fluid information, including the calibration data, is saved to the template. The fluid template can be edited and renamed in the catalog, and it is available for reuse in the *Template* options list when you create a new fluid.

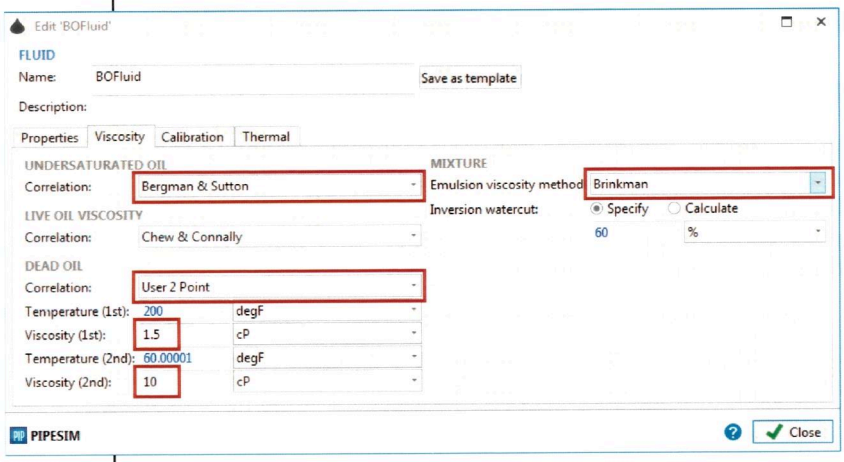
Similarly, wells can be saved as templates in the Catalog. All the well information, excluding the fluid information mapped to the well, is saved to the template. This is useful if you have *typical/ type* wells; they can be saved as templates and reused in large networks, resulting in significant time savings.



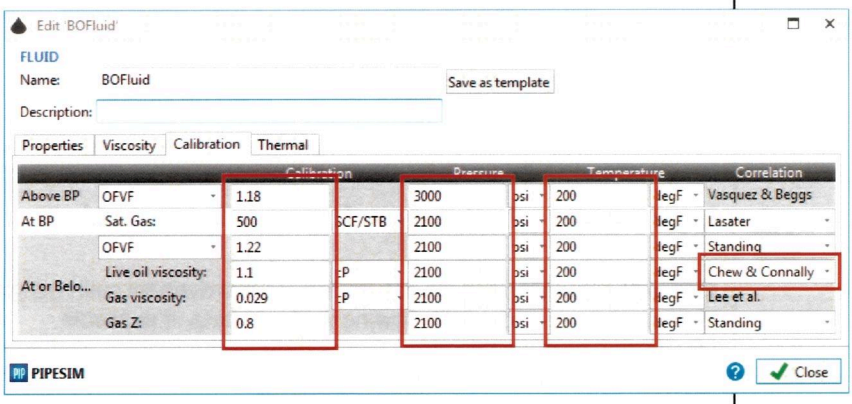
Exercise 1 Calibrate PVT data

Continue with the model from the previous exercise.

1. Launch the fluid editor from the **Completions** tab of the Well editor or from the **Fluid Manager** on the **Home** tab.
2. Click the **Viscosity** tab and configure the **PIPESIM** viscosity model settings, as shown in the figure.



3. Click the **Calibration** tab and enter the measured data shown in the figure to calibrate the PVT model. After you enter the data, click **Close** to exit the fluid editor. The fluid is now calibrated.



4. It is useful to see if the fluid calibration has an impact on the depth at which the bubble point is reached. Re-run the P/T profile task with the same Outlet pressure of 300 psi from the previous exercise to observe the difference in results between the uncalibrated and calibrated models.
5. Determine the flowing bottomhole pressure, flowing wellhead temperature, AOFP and production rate for the given wellhead pressure and enter them in the Results table.
6. Compare your answers to the uncalibrated model results.

Results		
Wellhead pressure = 300 psi	**Uncalibrated**	**Calibrated**
Production Rate, STB/d		
Flowing BHP, psi		
Flowing WHT, deg.F		
AOFP, STB/d		
Depth where gas appears		

NOTE: **PIPESIM** provides the option to calibrate Multiphase flow correlations and the Overall Heat Transfer Coefficient (U-value) by tuning them to match field measurements (flowing pressures, rates, and temperatures). This is useful for improving model accuracy and predictability.
Currently, this data matching task is available only in the **PIPESIM** Classic version. It will be introduced in the new **PIPESIM**.

Lesson 4 Inflow performance matching

PIPESIM simulation results can be further improved by calibrating the well IPR equations with flowing well test data, if available. Ideally, you perform this step after calibrating the multiphase flow correlation for the Outflow curve, by tuning it to match measured flowing rate and pressure data. Even if the Outflow curve cannot be calibrated because there is a lack of measurements, the IPR can still be calibrated if well performance data is available.

Most of the **PIPESIM** IPR equations can be calibrated with well test data by selecting the *Use test data* box on the **Completions** tab of the **Well editor** and entering the data. (Do not do this if there is no well test data available.)

If the rate, static reservoir pressure, and flowing wellhead pressure are available, equations can still be matched even without multipoint or isochronal well test data. Match the equations by tuning the uncertain parameters of the IPR equation, such as Skin for the Darcy equation or Production Index for the PI equation.

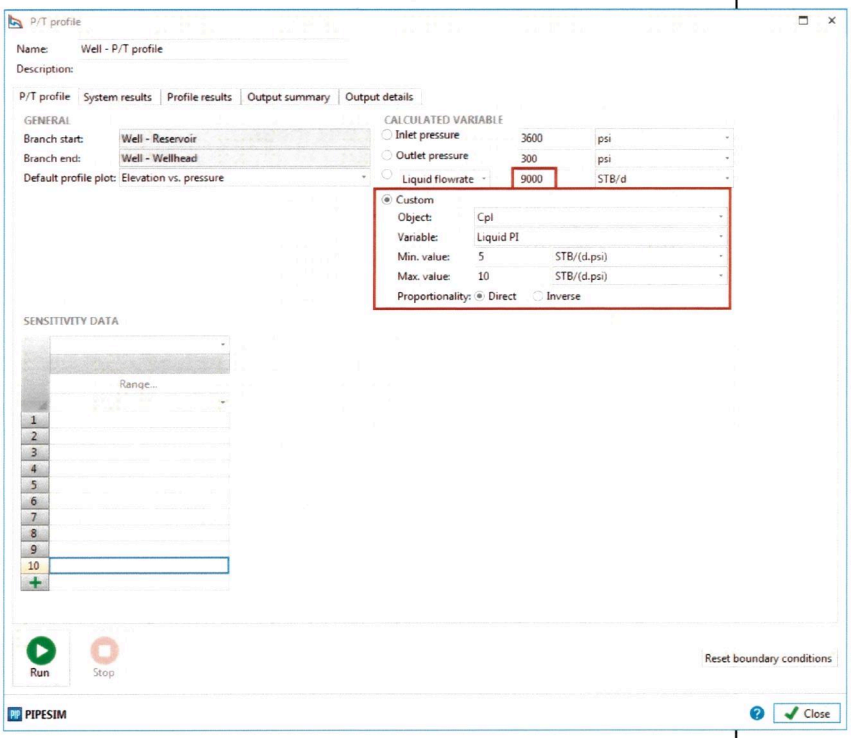
For the oil well in this case, a Liquid PI value of 8 STB/d psi was initially specified. It is a more accurate value based on the available well performance information.

Exercise 1 Sensitize on the well PI to match performance

This exercise continues with the previously built model, with the objective to determine the productivity index based on a well test. A 4-hour well/production test was run on the well, and a rate of 9,000 STB/d was measured at a flowing wellhead pressure of 300 psi.

You use a pseudo iterative method in **PIPESIM** to calculate the Productivity Index value that matches the known flow rate and boundary conditions.

1. Launch the P/T profile task used in the previous exercise.
2. Reconfigure the task as shown in the figure, so that the Liquid PI of the completion is the calculated variable based on the known pressure and flow rate data.



NOTE: The *proportionality* refers to the relationship between the custom variable and the outlet pressure. A *Direct* proportionality indicates an expectation that the outlet pressure increases when the custom variable increases, for example the choke bean diameter.

An *Inverse* proportionality implies an expectation that the outlet pressure decreases when the custom variable increases, such as water cut of a black oil fluid in a production well. Refer to the **PIPESIM** Help (F1) for additional details.

3. Run the task.
4. To see the Productivity index (LPI) value that matches the actual data, inspect the P/T task Profile results.
5. Update the IPR model in the **Completions** tab of the well editor with the matched productivity index value.
6. To determine the new AOFP of the well, re-run the Nodal analysis task.
7. Compare the new AOFP to the value from the earlier exercise with an uncalibrated IPR model.

Results	
Matched PI	
New AOFP	
Previous AOFP	

Lesson 5 Well performance analysis



After building an accurate base case model for the well, several simulation operations can be performed to evaluate a variety of operating scenarios.

Water-cut sensitivity analysis

After an initial design is made, it is important to evaluate how the system responds to changing operating conditions. For example, increasing water production is inevitable for most oil and gas fields. As a result, water production must be considered in well design.

At some point in the producing life of the well, it is possible that the reservoir pressure is insufficient to lift the fluid to the surface when the water cut reaches a specific limit. This situation occurs because water has a higher density than oil and, as the water cut increases, the hydrostatic pressure losses also increase.

Using the wellhead pressure, reservoir pressure, and matched PI from the previous exercise, you now determine the water cut limit for this well to be able to produce.

Either of the **PIPESIM** tasks, System analysis or Nodal analysis, can be used to model this problem. Both task options are presented in the exercise.

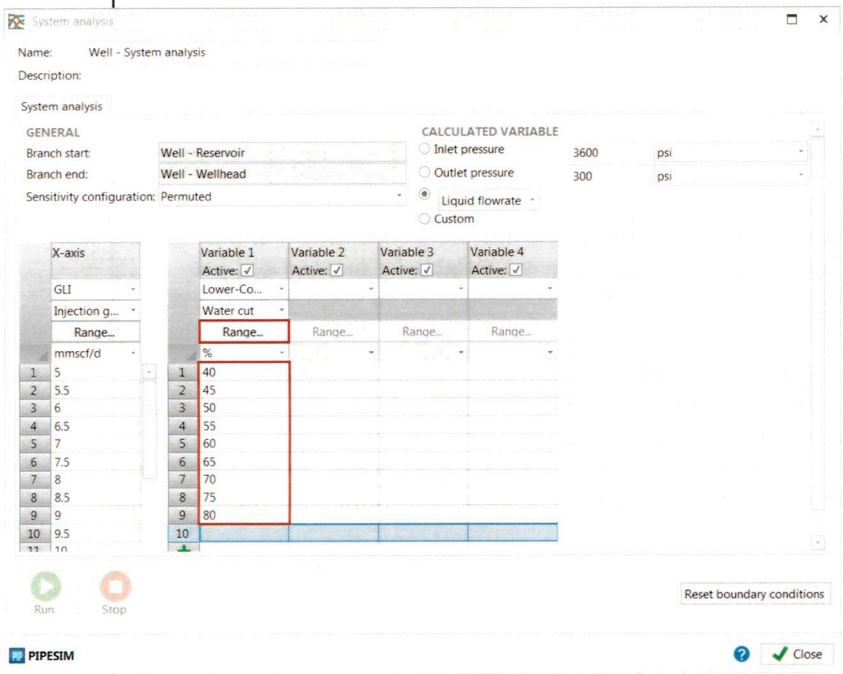
Exercise 1 Run a system analysis task



This exercise shows you how to set up and run the system analysis task for this problem.

1. Ensure that you enter your matched PI value of `9.375` STB/d/psi in the IPR model under the **Completions** tab of the well editor.
2. Launch the System analysis task from the **Home** tab.
3. Select *Liquid flow rate* as the Calculated variable.
4. Enter the Outlet Pressure as `300` psi (the required minimum wellhead pressure).
5. For the X-axis value, select *Cpl* (*Completion*), then select *Water Cut*.

6. Click the **Range** button to configure water-cut values of 40% to 80%, in increments of 5%.

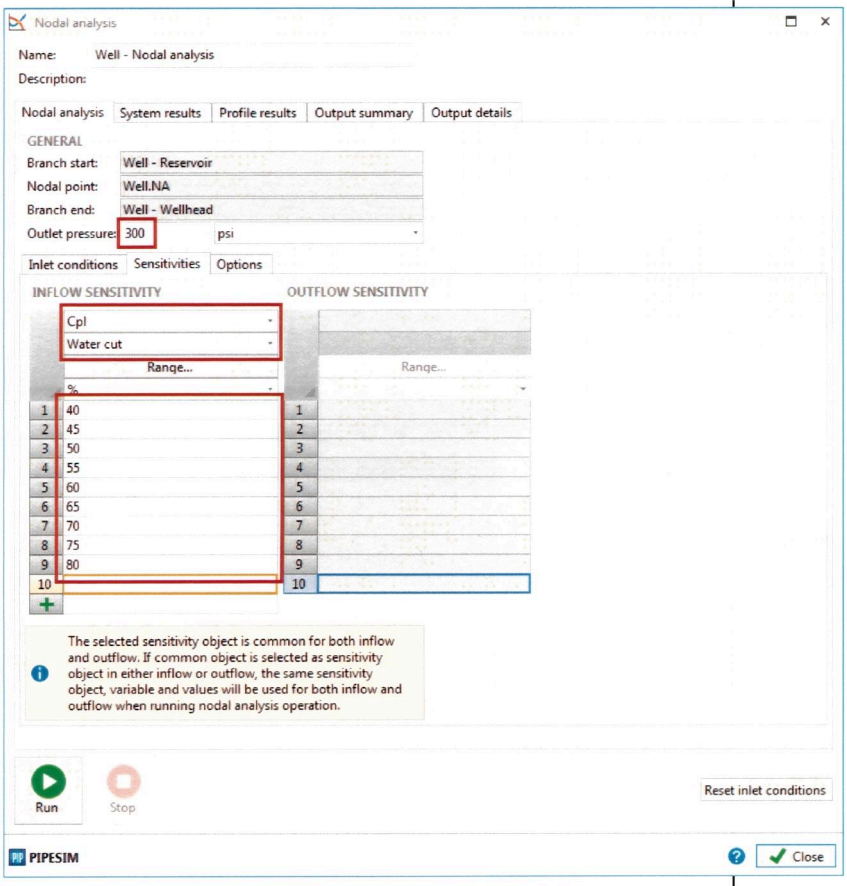


7. Run the model to generate a plot of calculated liquid rate vs. water cut. You observe in the plot that there are no results for some of the requested water-cut values. This is because there was no solution for those high water cut values, because the well is dead at those conditions.
8. Re-run the System analysis using more closely spaced sensitivity values to determine the critical water cut, such as the water cut value above which the well stops flowing.
9. Record the answer.

Exercise 2 Run a nodal analysis task

This exercise shows you how to set up and run the Nodal analysis task for this problem.

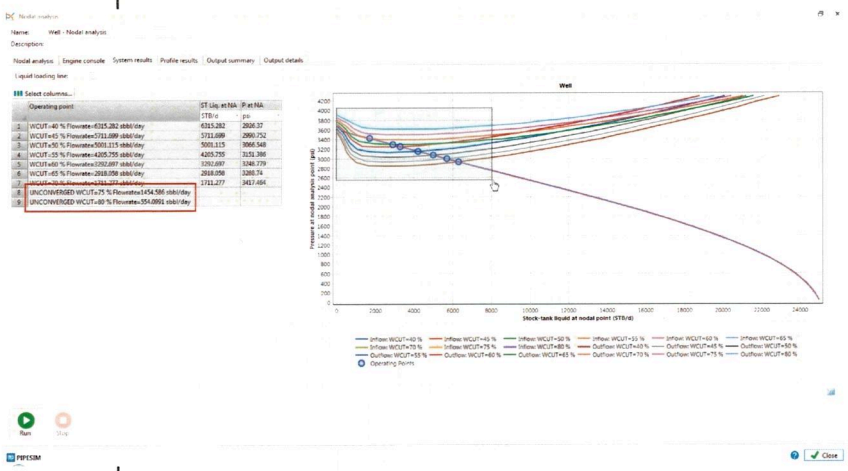
1. Launch the Nodal Analysis task from the **Home** tab.
2. Click the **Sensitivities** tab and configure Water cut as the Inflow Sensitivity variable.



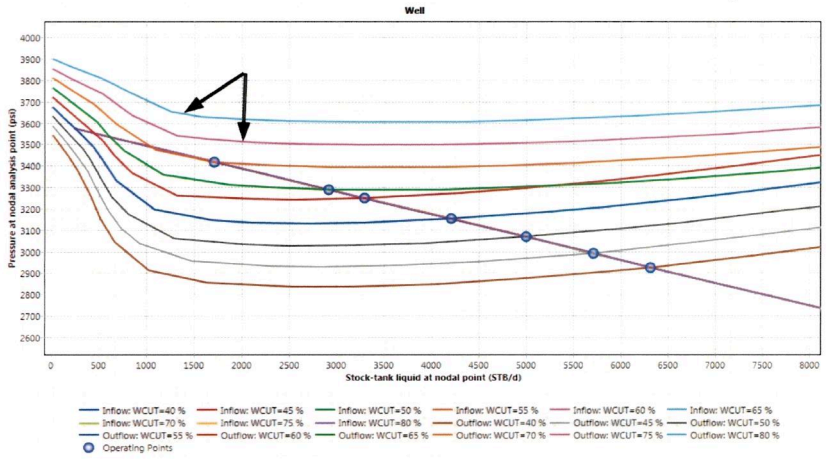
3. Run the model to generate the Nodal analysis plot and determine the critical water cut, such as the water cut value above which the well stops flowing.

NOTE: No solution is shown for water cut values of 75% and 80%.

4. Change the display.
   - Zoom a section of the plot. Draw a rectangle from top left to bottom right.
   - Un-zoom a section of the plot. Draw a rectangle from bottom right to top left.



Arrows in the figure identify the water cut values at which the well is dead.



Results	
Critical Water Cut	

Lesson 6 Artificial lift

The primary purpose of installing an artificial lift system is to maintain a reduced bottomhole pressure to enable the desired reservoir fluids to be produced at an acceptable rate.

There are several artificial lift options that operate in two primary ways: by decreasing hydrostatic head such as Gas lift, or by mechanically lifting the fluids, such as electric submersible pumps (ESPs). **PIPESIM** is able to model several artificial lift technologies including Gas lift, ESPs, Rod pumps, progressive cavity pumps (PCPs) and Multiphase boosters.

In the following exercises, you model the performance of the oil well with Gas lift installed. Then you design and install an ESP, and evaluate the well performance.

You also determine the better artificial lift option (purely from a performance perspective), taking into account a gas lift availability constraint of 3 mmscf/d and a design liquid production rate of 10,000 STB/d at 70% water cut.

Exercise 1 Evaluate gas lift performance

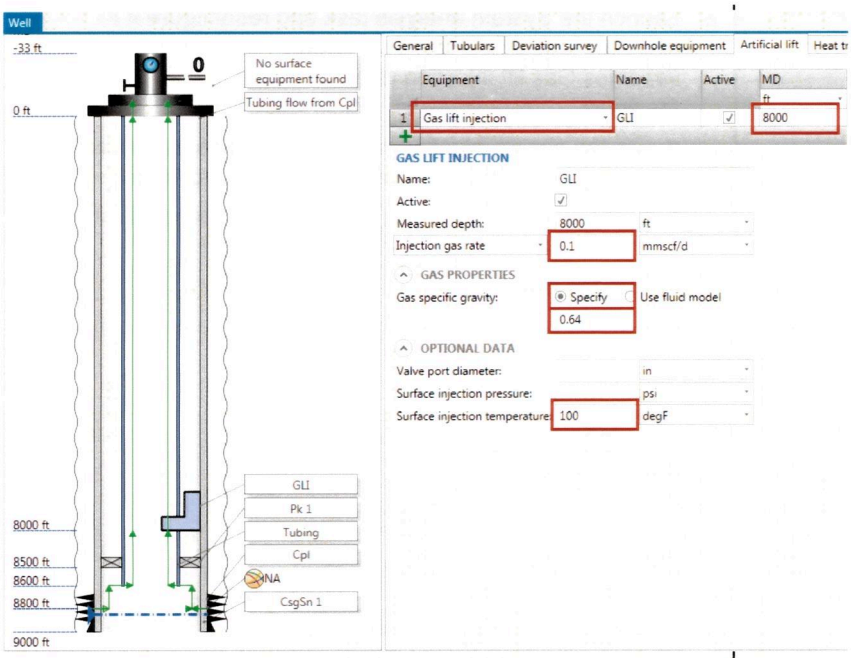
The basic principle behind gas lift injection in oil wells is to lower the density of the fluid in the tubing. This results in a reduction of the elevational/hydrostatic component of the pressure gradient above the point of injection and a lower bottomhole pressure. Lowering the bottomhole pressure increases reservoir drawdown and thus, production rate.

In this exercise, you examine how this well responds to gas lift by introducing a *Gas Lift Injection* point near the bottom of the tubing above the packer.

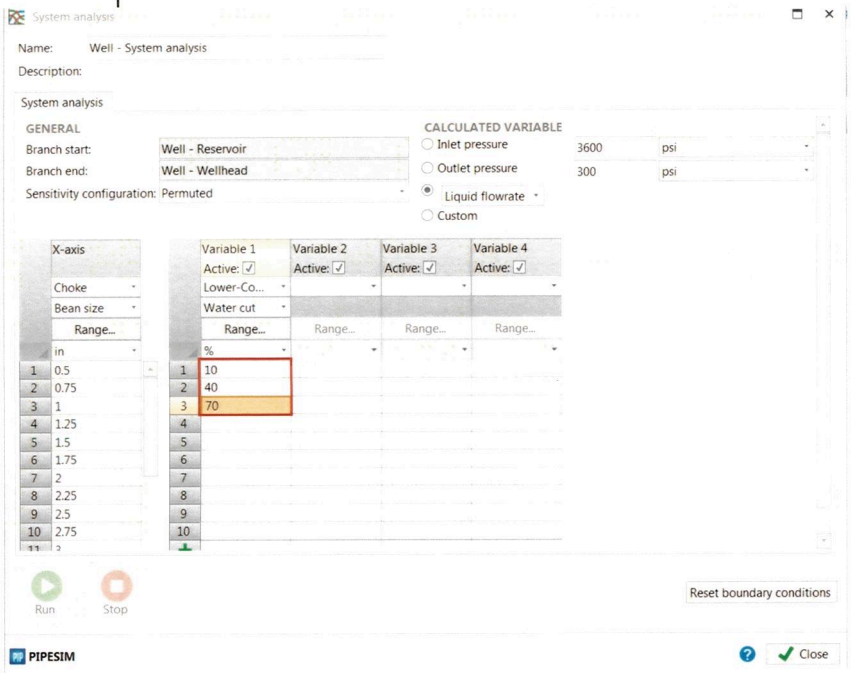
You also determine if the well can deliver the design liquid production rate of 10,000 STB/d at 70% water cut, with a maximum gas injection rate of 3 mmscf/d. The System analysis task is used to evaluate a range of cases with varying gas injection rates and produced fluid water-cut values.

Continue using the model from the previous exercise.

1. In the **Well editor** window under the **Artificial lift** tab, add a Gas lift injection point and enter the details, as shown.


Well
-33 ft
0 ft
8000 ft
8500 ft
8600 ft
8800 ft
9000 ft
No surface equipment found
Tubing flow from Cpl
GLI
Pk 1
Tubing
Cpl
INA
CsgSn 1
General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat tr
Equipment
Name
Active
MD
ft
1
Gas lift injection
GLI
8000
GAS LIFT INJECTION
Name:
GLI
Active:
Measured depth:
8000
ft
Injection gas rate
0.1
mmscf/d
GAS PROPERTIES
Gas specific gravity:
Specify
Use fluid model
0.64
OPTIONAL DATA
Valve port diameter:
in
Surface injection pressure:
psi
Surface injection temperature:
100
degF

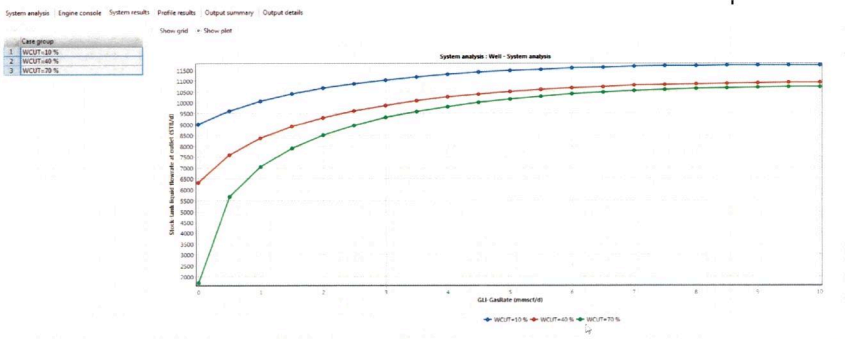
2. Launch the System analysis task and reconfigure it to calculate Liquid flow rate as a function of the permuted variables Gas lift injection rate and Water cut, as shown in the figure.
   a. Set the gas lift injection rate variable to be evaluated over a range from 0 to 10 mmscf/d in 0.5 mmscf/d increments.
   b. Enter `10%`, `40%`, and `70%` as the Water cut sensitivity values.
   c. Run the task.



3. Review the System results plot and determine the optimum gas injection rate for the worst-case water cut scenario of 70%.

TIP: Assume that every incremental gas injection rate of 0.5 mmscf/d must deliver a minimum incremental liquid rate of 100 STB/d to be economical.

Water cut, %	Optimum gas injection rate, mmscf/d	Liq. Prod. Rate, STB/d
70		



System analysis | Engine console | System results | Profile results | Output summary | Output details

Show grid • Show plot

	Case group
1	WCUT=10 %
2	WCUT=40 %
3	WCUT=70 %

	GLI-GasRate	Stock-tank liq...	GLI-GasRate	Stock-tank liq...	GLI-GasRate	Stock-tank liq...
	mmscf/d	STB/d	mmscf/d	STB/d	mmscf/d	STB/d
	WCUT=10 %	WCUT=10 %	WCUT=40 %	WCUT=40 %	WCUT=70 %	WCUT=70 %
1	0	9005.901	0	6315.285	0	1715.131
2	0.5	9608.837	0.5	7585.754	0.5	5681.408
3	1	10050.25	1	8357.594	1	7052.527
4	1.5	10384.74	1.5	8895.251	1.5	7899.575
5	2	10647.03	2	9296.872	2	8496.832
6	2.5	10857.71	2.5	9610.928	2.5	8952.723
7	3	11029.08	3	9861.737	3	9305.968
8	3.5	11169.7	3.5	10065.55	3.5	9587.438
9	4	11285.66	4	10231.94	4	9815.153
10	4.5	11381.33	4.5	10369.19	4.5	10001.24
11	5	11460.18	5	10482.51	5	10153.32
12	5.5	11523.65	5.5	10575.39	5.5	10278.52
13	6	11575.45	6	10651.5	6	10380.76
14	6.5	11616.9	6.5	10713.27	6.5	10465.84
15	7	11649.36	7	10769.22	7	10530.69
16	7.5	11673.94	7.5	10809.54	7.5	10590.68
17	8	11691.55	8	10840.91	8	10633.88
18	8.5	11702.74	8.5	10864.37	8.5	10666.94
19	9	11708.58	9	10880.46	9	10690.96
20	9.5	11709.61	9.5	10891.04	9.5	10707.33
21	10	11706.35	10	10897.86	10	10716.37

4. Determine how much the well would produce at 70% water cut with the maximum gas injection rate of 3 mmscf/d.

   Will the gas lift system be able to deliver the target production rate?

Water cut, %	Available gas injection rate, mmscf/d	Liq. Prod. Rate, STB/d
70	3	



Exercise 2 Evaluate ESP performance

An electrical submersible pump (ESP) is a multistage centrifugal pump that is able to provide a significant boost in pressure to produce large volumes of reservoir fluids. Each stage of the ESP consists of an impeller and a diffuser mounted on a shaft. As the shaft rotates, centrifugal forces created by the impellers impart kinetic energy on the fluid, after which the diffusers convert this kinetic energy to pressure, which boosts the fluid as it flows through the pump, to the discharge.

When you browse to select a pump in the ESP Design task, the ESP catalog shows a filtered list of recommended pumps when it launches. The list is ranked based on the efficiency at the design conditions. The pump recommendations take into account the casing ID, equipment clearance, and design production rate at pump intake conditions.

The *intake rate* is the surface rate corrected to the pump intake conditions, which also accounts for the dissolved gas. By default, the only pumps that display in the catalogue, are the ones that fit into the casing with the specified clearance and can handle the design flow rate. To see all the pumps in the catalog, clear the *Show recommended pumps* box.

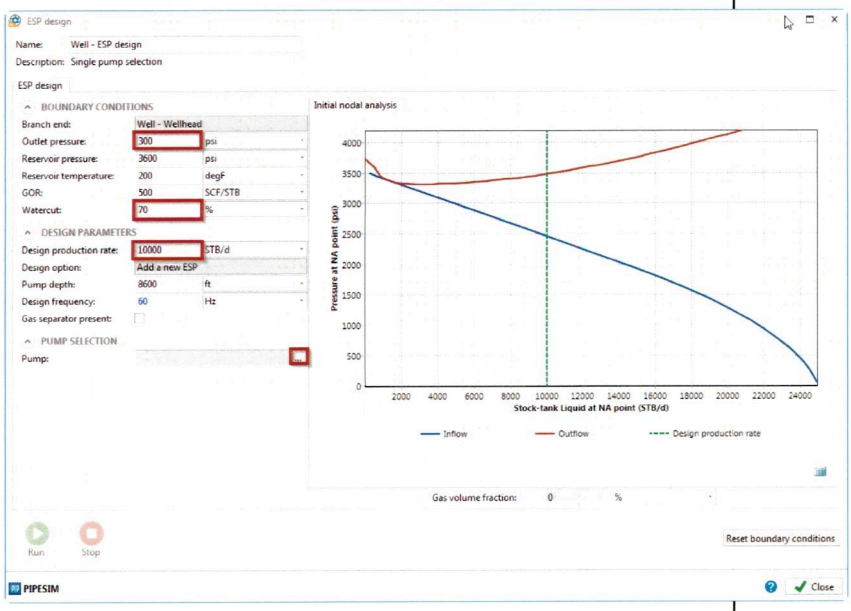
In this exercise, you design an ESP to deliver the design liquid production rate of 10,000 STB/d at 70% water cut. You install this pump in the well and evaluate the well performance over the expected range of water cut conditions.

Continue using the model from the previous exercise.

1. In the **Well editor** window under the **Artificial lift** tab, deactivate the Gas lift injection point, by clearing its *Active* box.
2. Launch the **ESP design** task from the **Home** tab.
3. Enter the outlet pressure of `300` psi.

   The nodal analysis for the current well conditions is automatically run and displayed in the plot on the right.
4. Change the Watercut to `70`%. The nodal analysis plot automatically updates.

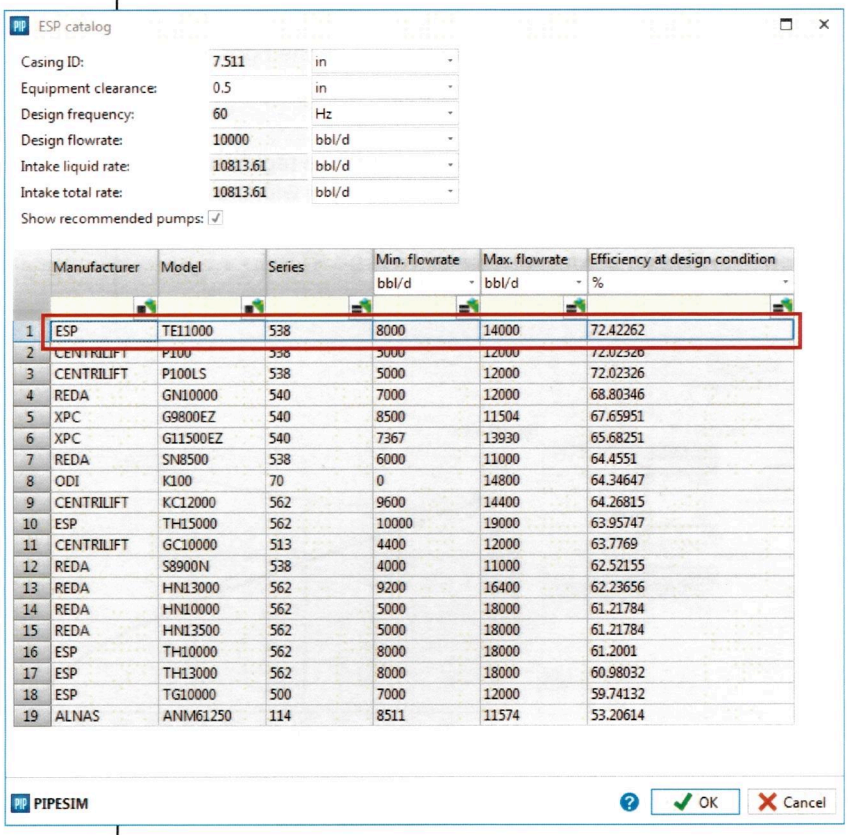
   At this new condition, the well is in the unstable region and is at the point where it soon die. This is consistent with previous exercises.
5. Enter the Design production rate of `10,000` STB/d.
6. In the *Pump Selection* section, browse to select a Pump.



NOTE: The only ESP design option available when there is no installed ESP in the well model, is to **Add a new ESP**. After you install an ESP in the well, you have the option to Replace the ESP at the same depth or Add a new ESP at another depth.

7. The ESP catalog launches. Select the ESP with the highest efficiency, as shown in the figure, and click **OK**.

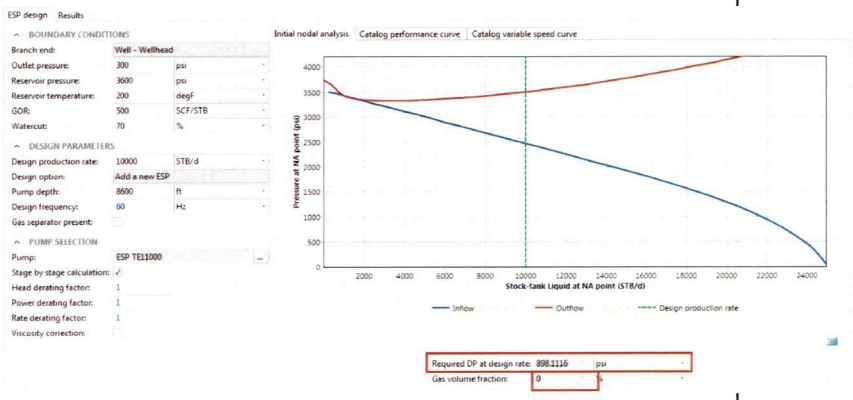
   This pump can handle a range of fluid rates from 8,000 to 14,000 STB/d, which provides some operational flexibility as the well conditions change.



	Manufacturer	Model	Series	Min. flowrate bbl/d	Max. flowrate bbl/d	Efficiency at design condition %
1	ESP	TE11000	538	8000	14000	72.42262
2	CENTRILIFT	P100	538	5000	12000	72.02326
3	CENTRILIFT	P100LS	538	5000	12000	72.02326
4	REDA	GN10000	540	7000	12000	68.80346
5	XPC	G9800EZ	540	8500	11504	67.65951
6	XPC	G11500EZ	540	7367	13930	65.68251
7	REDA	SN8500	538	6000	11000	64.4551
8	ODI	K100	70	0	14800	64.34647
9	CENTRILIFT	KC12000	562	9600	14400	64.26815
10	ESP	TH15000	562	10000	19000	63.95747
11	CENTRILIFT	GC10000	513	4400	12000	63.7769
12	REDA	S8900N	538	4000	11000	62.52155
13	REDA	HN13000	562	9200	16400	62.23656
14	REDA	HN10000	562	5000	18000	61.21784
15	REDA	HN13500	562	5000	18000	61.21784
16	ESP	TH10000	562	8000	18000	61.2001
17	ESP	TH13000	562	8000	18000	60.98032
18	ESP	TG10000	500	7000	12000	59.74132
19	ALNAS	ANM61250	114	8511	11574	53.20614

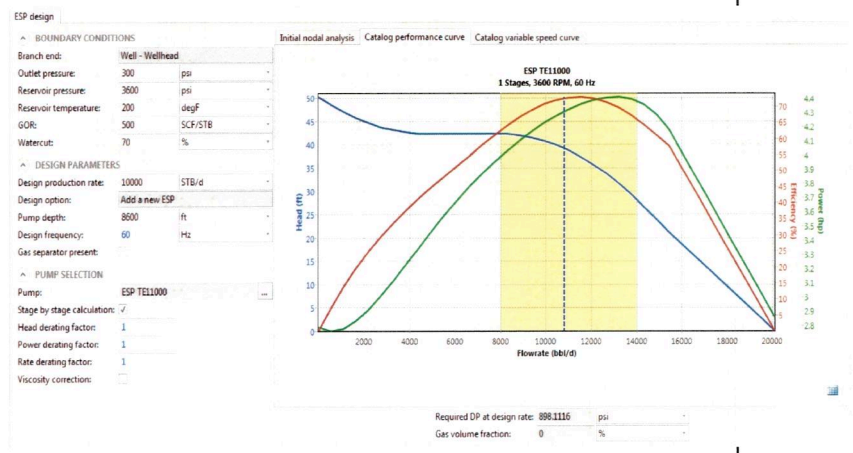
The Required DP at design rate and the Gas volume fraction at the pump intake were calculated. The Required DP at design rate is estimated from the nodal analysis plot as the difference between the outflow curve value and the inflow curve value at the design rate.

For this scenario, the GVF is zero at the pump intake conditions because the fluid is above the bubble point (there is no free gas).



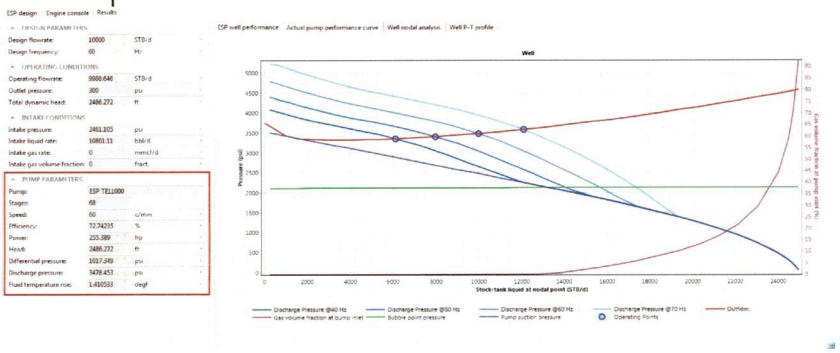
8. Click the Catalog performance curve to see the head, efficiency, and power requirement for one stage of this pump, operating at 60 Hz. All these calculations are based on pump curves provided by the manufacturer, which use water as the reference fluid.

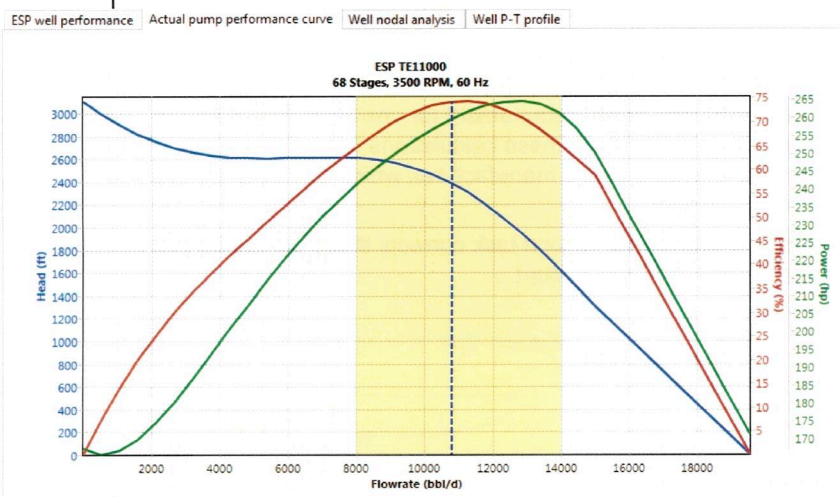
   The **PIPESIM** ESP design task corrects these curves for the actual well fluid being lifted.

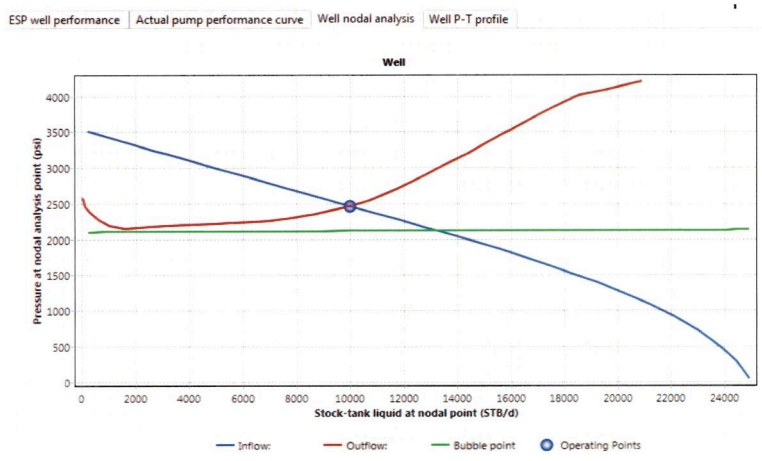


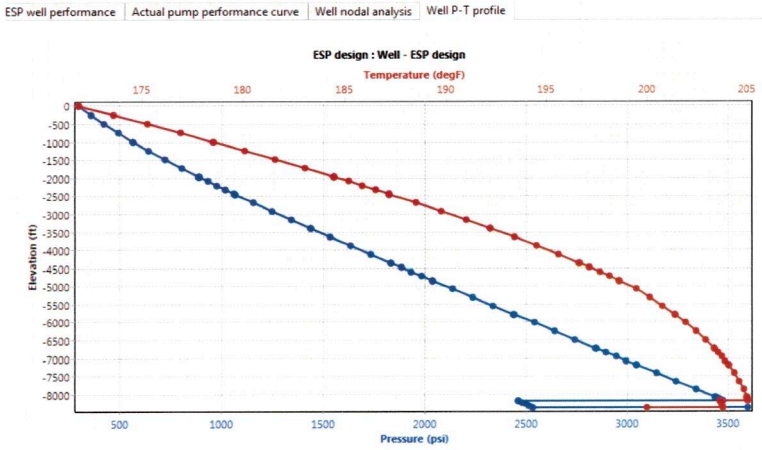
9. Click the Catalog variable speed curve to see the ESP performance at various frequencies.

10. Run the task, review the results on the various tabs, and record your answers in the table.



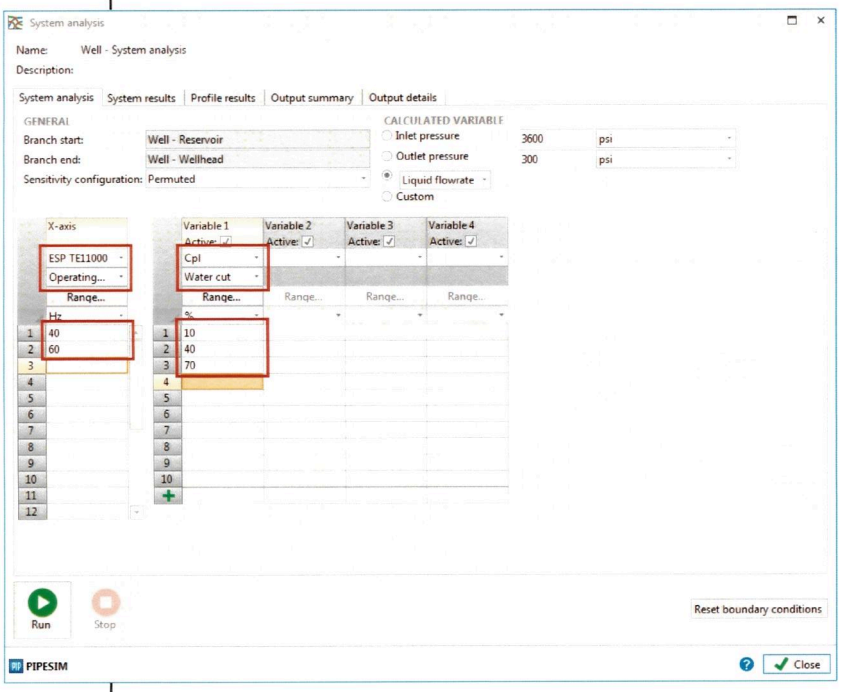


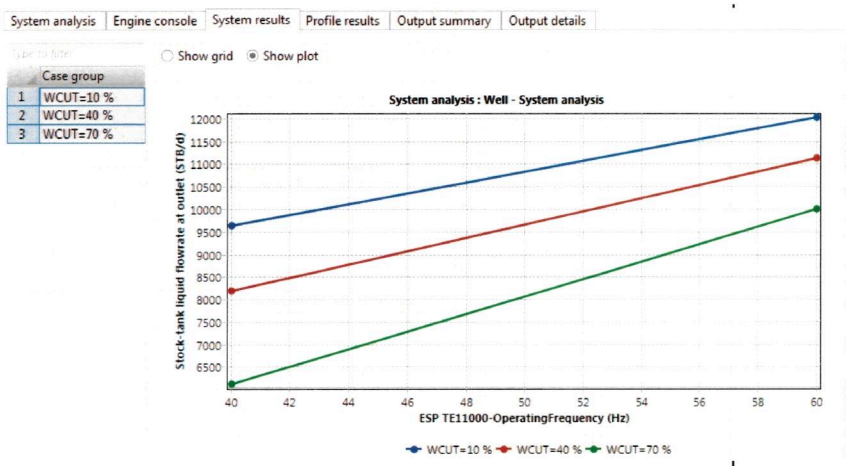




ESP design results (Water cut = 70%)	
ESP Model	TE11000
No. of stages	68
Efficiency, %	72.7
Power, hp	255
Head, ft	2,485
Differential pressure, psi	1,017
Fluid temperature rise, degF	1.4

11. On the **Results** tab, click **Install ESP**. In the **Well editor**, verify that the ESP was installed in the tubing.
12. Exit the **ESP Design** dialog box.
13. Launch the System analysis task and configure it as shown in the figure to evaluate the well performance over various pump frequencies and water cuts.





Based on the design objective and constraint outlined previously, which artificial lift option would you select; Gas Lift or ESP? Why?

Exercise 3 Model multiple completions



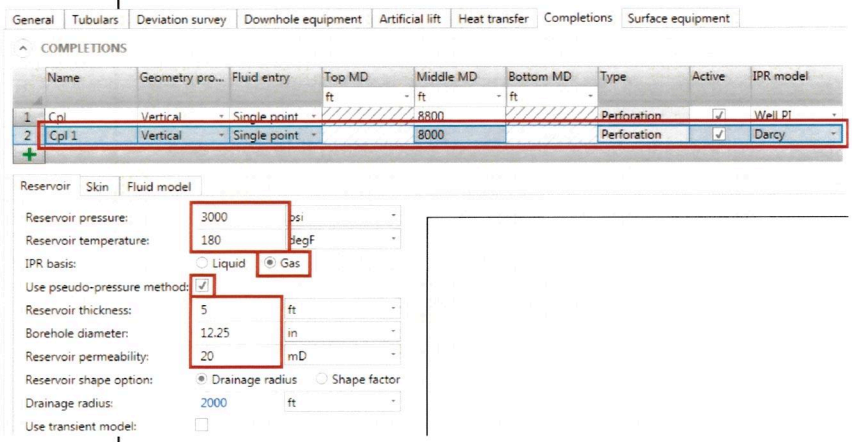
For your sample well, log analysis shows that a shallow gas zone exists uphole from the producing zone. As a future artificial lift option, you investigate the benefits of perforating this zone and self-lifting the well.

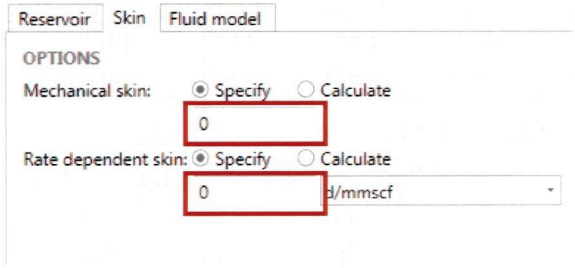
Continue with the model from the previous exercise:

1. Go to the **Artificial lift** tab in the **Well editor** window and delete the Gas lift injection point and ESP.
2. On the **Completions** tab, add a second completion to the well at a depth of `8,000` ft MD.
3. With no test data available, model the reservoir performance of the upper zone using the pseudo-steady state Darcy IPR equation.

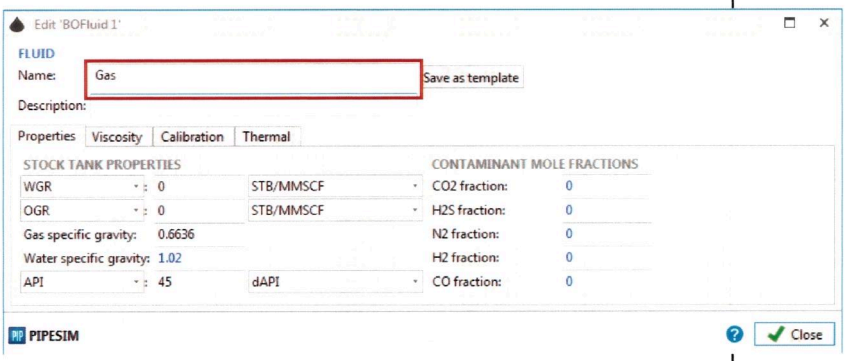
Specify the upper completion using the data in the table. (Use default values for all unspecified parameters.)

Reservoir properties - Upper gas zone	
Property	**Darcy**
Reservoir pressure	3,000 psia
Reservoir temperature	180 degF
Basis of IPR calculation	Gas
Use Pseudo-pressure method	Yes
Thickness	5 feet
Borehole diameter	12.25 inch
Permeability	20 md
Drainage radius	2,000 ft
Mechanical skin	0
Rate Dependent skin	0

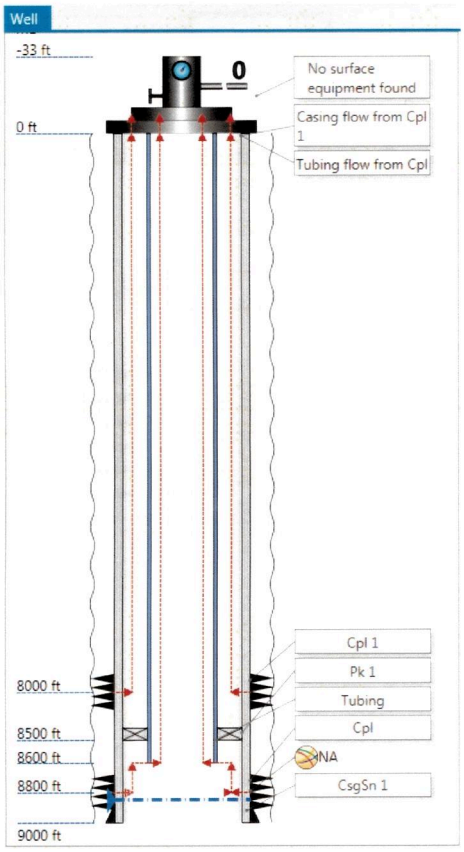




4. Create a new fluid for the completion using the Dry gas template and name it `Gas`.



5. The well schematic diagram in the **Well editor** window updates with changes made to the configuration of the well. The flow path indication lines that were green up to this point have changed to red. This indicates that the well is unsolvable in its current configuration.

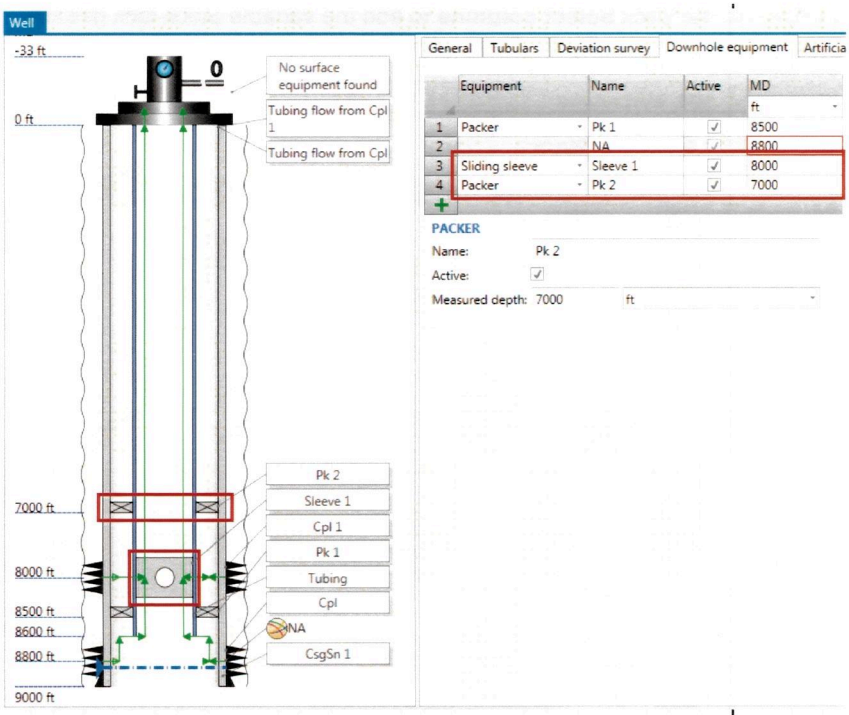


To solve this problem:

a. On the **Downhole equipment** tab, add a Sliding sleeve at 8,000 ft.

b. Select its *Active* box.

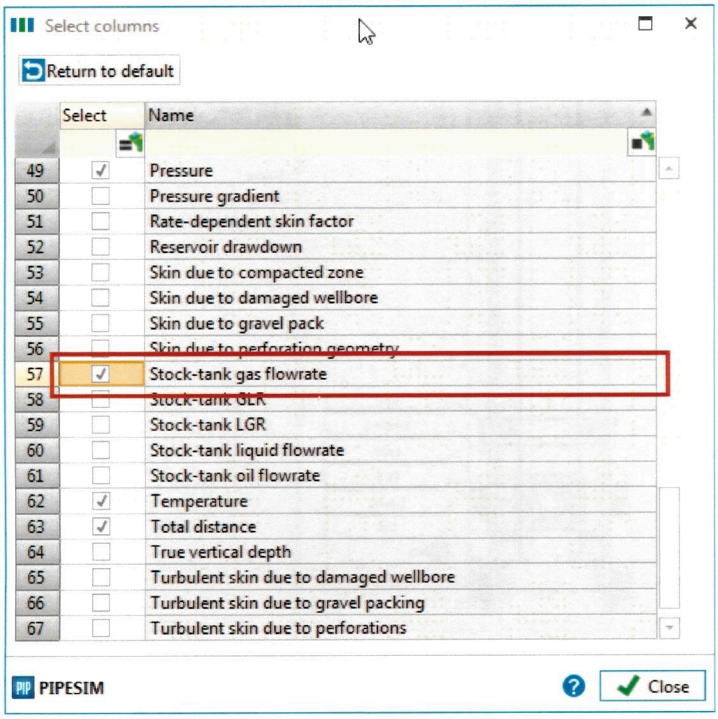
c. Add a second packer at 7,000 ft; to divert the flow of gas from the upper zone into the tubing.



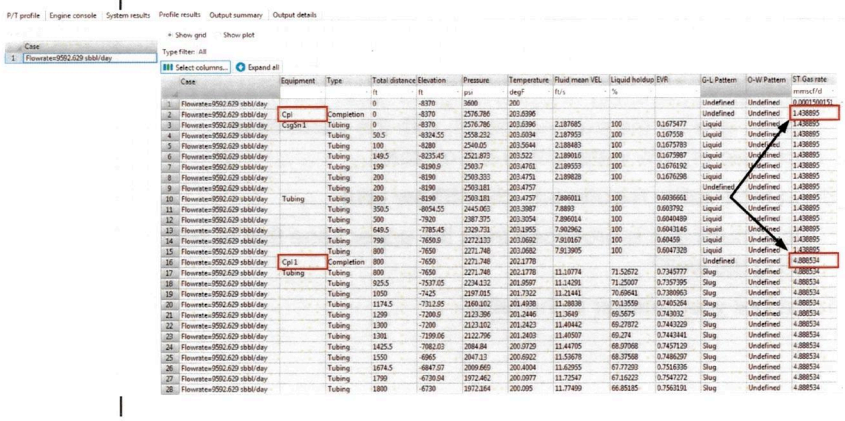
6. Click the **Validations** tab.
7. Currently, there is a validation issue with the depth of the Nodal analysis point because a second completion was added. **PIPESIM** requires that the Nodal analysis point be placed at, or above, the shallowest completion.
8. On the **Downhole equipment** tab, change the depth of the NA (Nodal analysis point) to 8000 ft.
9. To analyze the effect of perforating the upper zone, run a P/T profile task for the existing model, with the worst-case scenario of 70% water cut in the oil zone.
10. On the **Profile results** tab, click **Show grid**.

    How much gas is produced from the upper zone that is self-lifting the well?
11. Record your answer in the table.

12. Click **Select columns** to add the variable Stock-tank gas flow rate to the Profile results grid.



In the figure, the gas contribution from the upper zone is (Cpl1) = 4.888534 - 4.38895 = 3.449 mmscf/d.



Self-lifting results	
Liquid Rate (STB/d)	
Gas Rate (upper zone) (mmscf/d)	

How does the self-lifting compare with the targeted gas lifting modeled in Exercise 1: *Evaluate gas lift performance*?

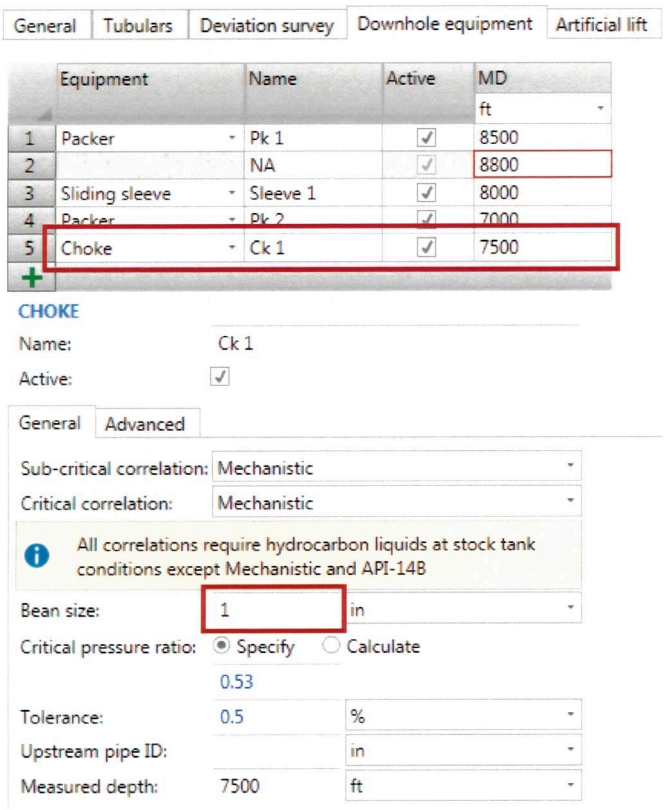
Lesson 7 Downhole chokes

A downhole choke is used to control flow inside a wellbore. All the calculations associated with a downhole choke are similar to that of a surface choke.

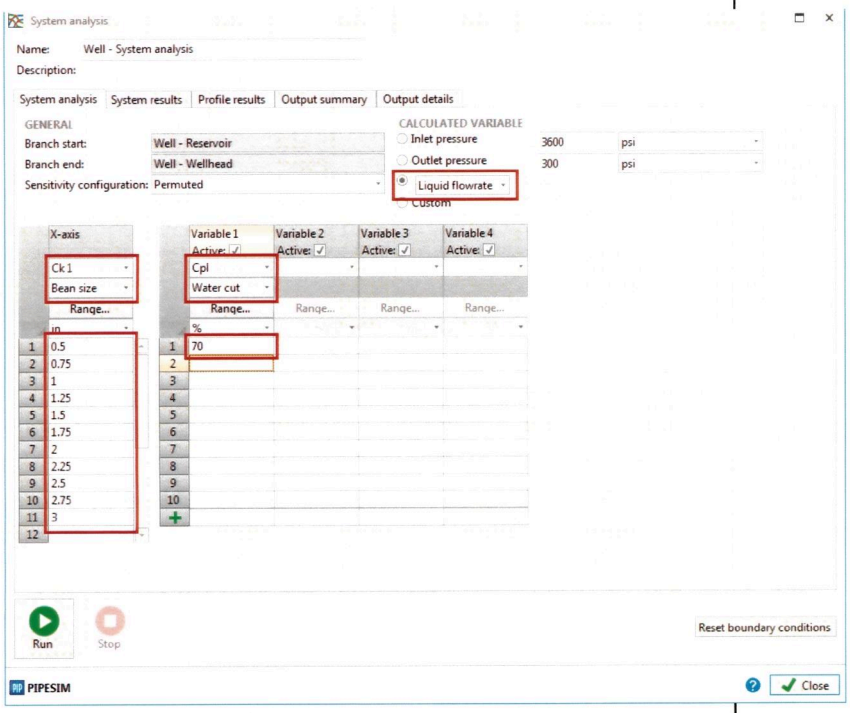
Exercise 1 Model a downhole choke

A downhole choke is used to control the flow of gas from the upper completion in the model from the previous exercise, assuming that there is a surface gas production constraint of 4 mmscf/d per well.

1. In the **Downhole equipment** tab of the **Well editor** window, add a choke to the list and configure it as shown in the figure.

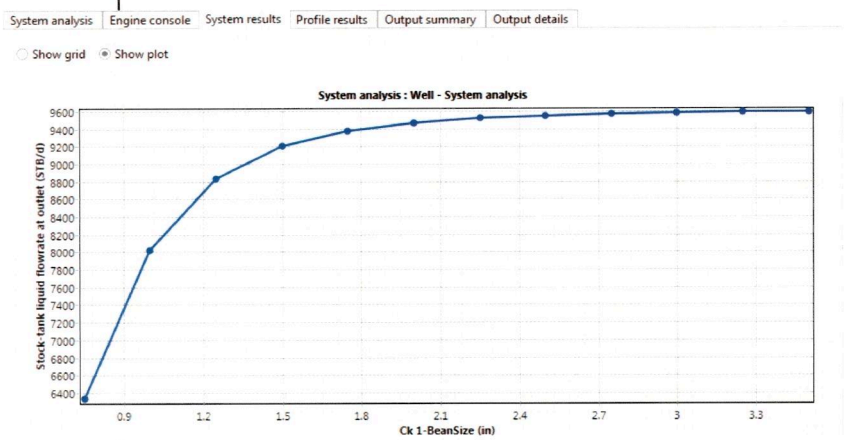


2. Reconfigure the System analysis task to calculate the liquid flow rate for a range of choke bean sizes (orifice sizes) from 0.5 to 3.525 inches, in increments of 0.25 inch and for the worst-case water cut scenario of 70%.



3. Run the task. Double-click the plot, select the Bottom X-axis.
4. Under the *Axis Scales* section, clear the *Automatic min/max* box.

5. Change the axis increment so the plot appears.



6. Determine the required choke bean size to honor the gas constraint of 4 mmscf/d per well and the produced liquid rate at that bean size. Record your results in the table.

   (Double-click the plot, change the Y-axis variable to *Stock-tank gas at outlet*, and reconfigure the axis increment, as needed).

7. The Choke bean size required to constrain the gas rate to 4 mmscf/d (1.05 inches).

Review questions

- What is the purpose of Nodal analysis?
- What is the purpose of performing a fluid calibration with laboratory data?
- What is the purpose of tuning the Productivity Index for the completion with data from a Flowing Gradient Survey?

Summary

In this module, you learned about:

- performing a Nodal analysis
- estimating bottomhole flowing conditions
- calibrating PVT data
- performing IPR matching
- conducting water cut Sensitivity analysis
- evaluating gas lift performance
- designing and installing an ESP
- modeling multiple completions
- installing a downhole choke

NOTES

Module 4 Gas well performance

This module guides you through the workflow of modeling a gas production well as a compositional fluid, rather than as a black oil fluid.

Learning objectives



After completing this module, you will know how to:

- build a simple compositional well model
- calibrate the inflow model
- select optimum tubing size
- evaluate erosion concerns
- model flowline and choke performance
- analyze liquid loading issues

Lesson 1 Compositional fluid modeling



PIPESIM offers full Compositional fluid modeling as an alternative to Black oil modeling. In compositional modeling, you specify individual components (Methane, Ethane, Water, etc.) that comprise the fluid and model the fluid phase behavior using equations of state.

Compositional fluid modeling is generally regarded as more accurate for wet gas, condensate, and volatile oil systems. However, detailed compositional data is less frequently available to a production engineer, which often constrains the engineer to using Black oil fluid models for fluids better suited to compositional modeling.

PIPESIM provides four fluid modes: Black oil, Compositional, PVT file, and MFL file. Refer to the **PIPESIM** Help for details on each of these fluid modes.

In the Compositional fluid mode, **PIPESIM** allows you to choose among three PVT flash packages.

Multiflash	A third-party compositional package (KBC)
Eclipse 300 Flash	A newer interface to the **ECLIPSE** two-phase flash has been introduced, allowing additional Equations of State
GERG 2008 Flash	A two-phase flash using the GERG-2008 Equation of State

It is important to note that the Multiflash package can be accessed in **PIPESIM** in two distinct ways.

Multiflash in the compositional fluid mode (native)

This option uses a subset of the full Multiflash package, specially adapted for the **PIPESIM** interface. This option is enabled when you set the fluid mode to Compositional and choose Multiflash as the PVT package.

The entire fluid definition is done at a global level using the **PIPESIM** interface. The same set of selected models (Equation of State, viscosity, BIP set, etc.) are applied to all individual fluids defined in the workspace. The models available with this option are a subset of the full extent of the models available with the full Multiflash standalone package enabled in **PIPESIM** using MFL file mode. The Gas Well performance exercise (Module 4) use this option, that is Multiflash native.

Multiflash MFL files

This option is enabled when you set the fluid mode to MFL file. The fluid definition is done using MFL files, which are created by launching the Multiflash interface from **PIPESIM**.

This option gives you access to the full extent of the models available in the Multiflash package and is the required option for wax and asphaltene thermodynamics. Refer to the **PIPESIM** Help for more details.

Equations of State

Equations of State (EoS) describe the pressure, volume, and temperature (PVT) behavior of pure components and mixtures. Most thermodynamic and transport properties are calculated from functions derived from an Equation of State.

One of the simplest Equations of State is the ideal gas law, PV = nRT, which is roughly accurate for gases at low pressures and high temperatures.

NOTE: The Black oil model uses this equation along with a compressibility factor (z) to account for non-ideal behavior.

This equation, however, becomes increasingly inaccurate at higher pressures and temperatures and fails to predict condensation from a gas to a liquid. As a result, much more accurate Equations of State have been developed for gases and liquids.

These are the Equations of State available in **PIPESIM 2014.1**. Refer to the **PIPESIM** Help for details.

Multiflash	• 3-parameter Peng-Robinson (1976) • 3-parameter Standard Soave-Redlich-Kwong (1972) • Multi-reference Fluid Corresponding States (CSMA) • Benedict-Webb-Rubin-Starling (BWRS) • Cubic Plus Association (CPA).
ECLIPSE 300 Flash	• 3-parameter Peng-Robinson (1978) • 3-parameter Standard Soave-Redlich-Kwong (1972)
GERG	GERG 2008

Viscosity

Compositional fluid models also use viscosity models based on corresponding state theory. Available viscosity models include:

- Pederson (default)
- Lohrenz-Bray-Clark (LBC)
- Aasberg-Petersen
- PedersenTwu*
- SuperTRAPP*

*available only with Multiflash

The Pedersen model is a predictive corresponding states model, originally developed for oil and gas systems. It is based on accurate correlations for the viscosity and density of the reference substance, which is methane. The model is applicable to both gas and liquid phases.

The SuperTRAPP model is a predictive, extended corresponding states model that uses propane as a reference fluid. It can predict the viscosity of petroleum fluids and well-defined components, over the entire phase range from dilute gas to the dense fluid.

Overall, the SuperTRAPP method is the most versatile method for viscosity predictions and its performance is generally better than the other methods. However, **PIPESIM** uses the Pedersen method as the default, because it also is widely applicable and accurate for oil and gas viscosity predictions.

The choice you make for the Equation of State has a significant impact on the viscosities and other fluid properties predicted by these methods. It is important to research the equations of state before choosing one that is recommended and most accurate for the fluid you are trying to model.

For more information, refer to the **PIPESIM** and Multiflash help for details.

Binary interaction coefficients

Binary interaction coefficients are adjustable factors used to alter the predictions from a model to match experimental data. They are usually generated by fitting experimental vapor-liquid equilibrium (VLE) or liquid-liquid equilibrium (LLE) data to the model.

Binary interaction coefficients apply to pairs of components, although the fitting procedure can be based on both binary and multi-component phase equilibrium information.

PIPESIM has default sets of binary interaction coefficients (Figure 16). They can be overridden, and you can supply your own data (Figure 17).

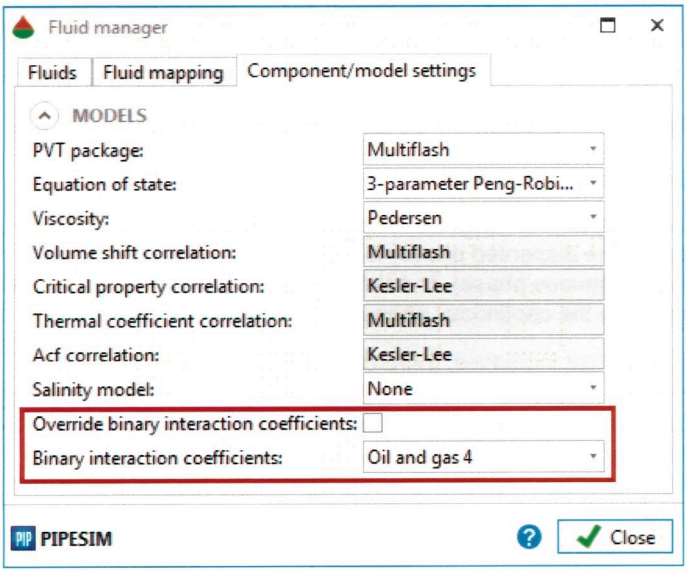
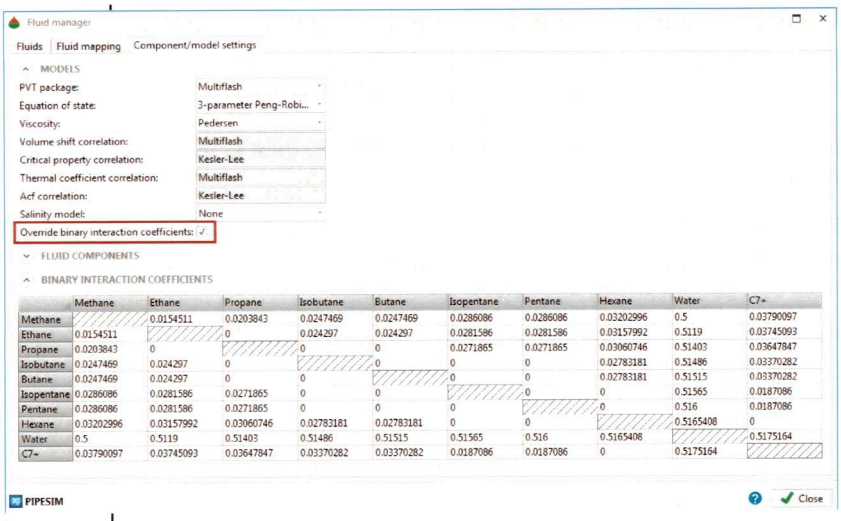


Figure 16 Binary interaction coefficients



	Methane	Ethane	Propane	Isobutane	Butane	Isopentane	Pentane	Hexane	Water	C7+
Methane		0.0154511	0.0203843	0.0247469	0.0247469	0.0286086	0.0286086	0.03202996	0.5	0.03790097
Ethane	0.0154511		0	0.024297	0.024297	0.0281586	0.0281586	0.03157992	0.5119	0.03745093
Propane	0.0203843	0		0	0	0.0271865	0.0271865	0.03060746	0.51403	0.03647847
Isobutane	0.0247469	0.024297	0		0	0	0	0.02783181	0.51486	0.03370282
Butane	0.0247469	0.024297	0	0		0	0	0.02783181	0.51515	0.03370282
Isopentane	0.0286086	0.0281586	0.0271865	0	0		0	0	0.51565	0.0187086
Pentane	0.0286086	0.0281586	0.0271865	0	0	0		0	0.516	0.0187086
Hexane	0.03202996	0.03157992	0.03060746	0.02783181	0.02783181	0	0		0.5165408	0
Water	0.5	0.5119	0.51403	0.51486	0.51515	0.51565	0.516	0.5165408		0.5175164
C7+	0.03790097	0.03745093	0.03647847	0.03370282	0.03370282	0.0187086	0.0187086	0	0.5175164	

Figure 17 Override binary interaction coefficients

Emulsion viscosities

An *emulsion* is a mixture of two immiscible liquid phases. One phase (the dispersed phase) is carried as droplets in the other (the continuous phase). In oil/water systems at low water cuts, oil is usually the continuous phase.

As water cut increases, there comes a point at which *phase inversion* occurs and water becomes the continuous phase. This is the *critical water cut of phase inversion*, also known as the *cutoff*, which typically occurs between 55% and 70% water cut. The viscosity of the mixture is usually highest at, and just below, the cutoff.

Emulsion viscosities can be many times higher than the viscosity of either phase alone. A number of methods for predicting emulsion viscosity are available for Compositional fluids in **PIPESIM** on the **Viscosity** tab (Figure 18).

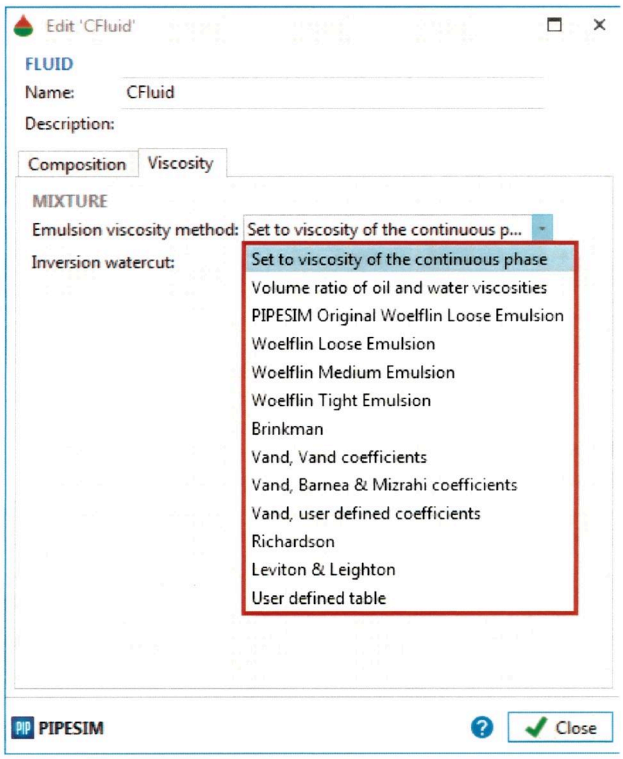
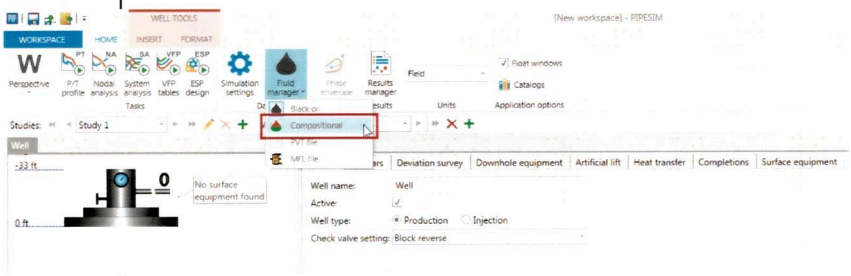


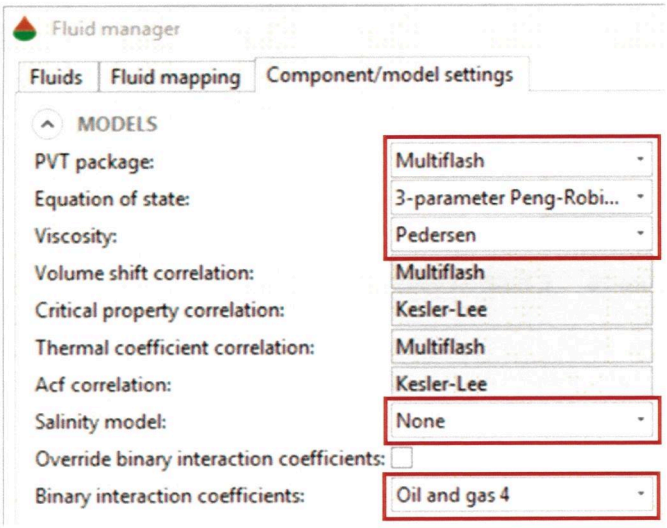
Figure 18 Emulsion viscosity options

Exercise 1 Create a compositional fluid model

1. launch **PIPESIM** and c-Launch **PIPESIM** and create a new well-centric workspace.
2. On the **Home** tab, select **Compositional** from the *Fluid manager* drop-down list. This launches the **Compositional fluid editor**.



3. In the **Component/model settings** tab of the **Fluid Manager**, select the following PVT package and models.



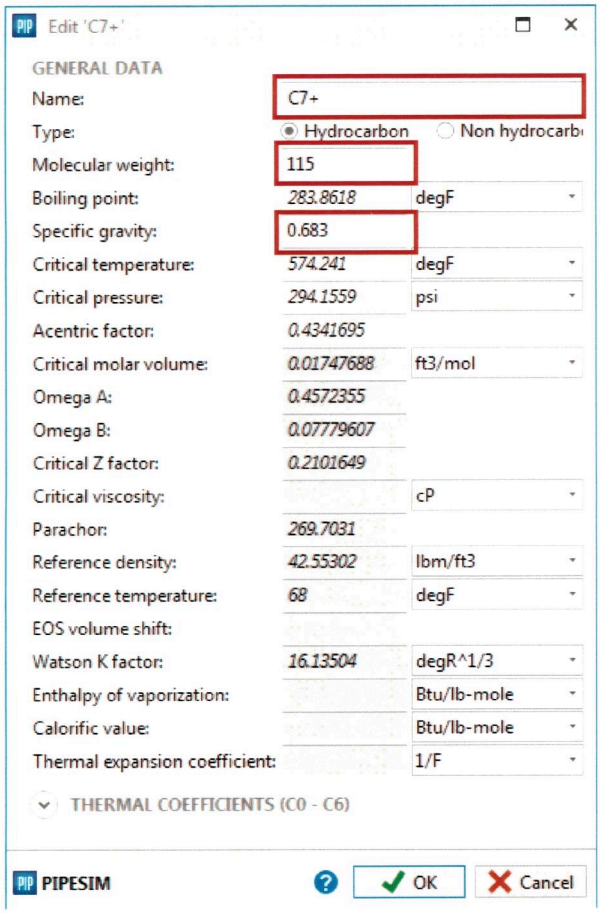
NOTE: If you do not have the license features required to use Multiflash, select the **E300** PVT package.

4. Add the following components to the fluid template by selecting the boxes next to each of them in the *Fluid Components* list. There should be 9 components in total.

Composition	
Methane	Isopentane
Ethane	Pentane
Propane	Hexane
Isobutane	Water
Butane	

5. Create a new C7+ pseudo-component.
   a. Click **New** at the top of the *Fluid Components* section.
   b. Enter only the Name, Molecular weight, and Specific gravity for the C7+ component, as shown in the figure.
   c. All other properties are automatically calculated based on the properties you specified.
   d. Click **OK** when complete. (Refer to the figure that follows.)

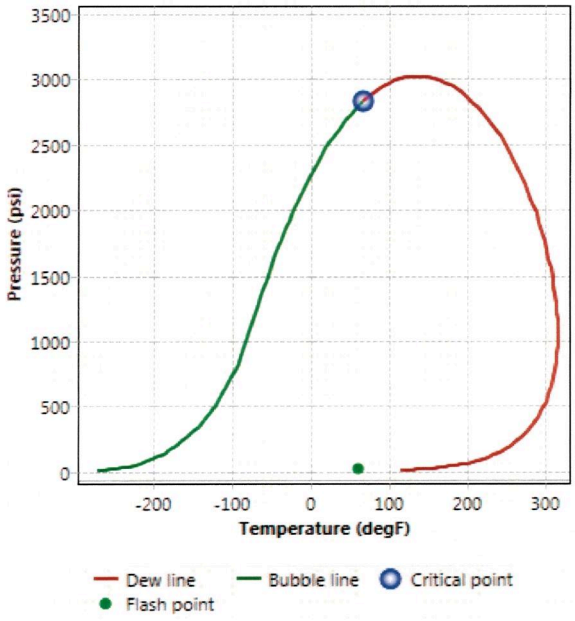
NOTE: In the interface, the *italic font* indicates calculated fields (such as Critical pressure and Acentric factor). Normal (non-italicized) font indicates fields for user input.



6. Go to the **Fluids** tab and click  to create a new fluid from the components you just added to the fluid template.
7. Double-click the row of the newly-created fluid to open the **Fluid** editor.
8. Enter the moles for each component as shown in the table. The phase diagram automatically updates as you enter the moles for each component.

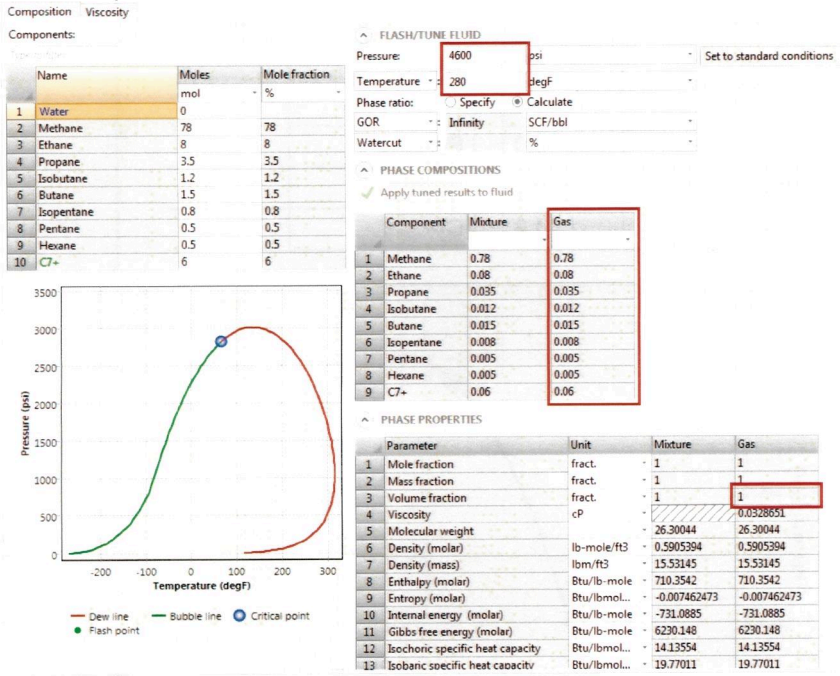
Component	Moles	Component	Moles
Water	0.0	Butane	1.5
Methane	78.0	Isopentane	0.8
Ethane	8.0	Pentane	0.5
Propane	3.5	Hexane	0.5
Isobutane	1.2	C7+	6.0

9.



In the *Flash/Tune fluid* section of the same window, you can enter any pressure and temperature and the fluid is flashed at those conditions. The phase properties and compositions resulting from the flash appears. Flash the fluid at the reservoir conditions: `4600` psi and `280` degF.

- What phases are present?
- What percentage of each phase is present?



Composition Viscosity

Components:

	Name	Moles	Mole fraction
		mol	%
1	Water	0	
2	Methane	78	78
3	Ethane	8	8
4	Propane	3.5	3.5
5	Isobutane	1.2	1.2
6	Butane	1.5	1.5
7	Isopentane	0.8	0.8
8	Pentane	0.5	0.5
9	Hexane	0.5	0.5
10	C7+	6	6

FLASH/TUNE FLUID

Pressure: 4600 psi — Set to standard conditions

Temperature: 280 degF

Phase ratio: Specify / Calculate

GOR: Infinity SCF/bbl

Watercut: %

PHASE COMPOSITIONS

Apply tuned results to fluid

	Component	Mixture	Gas
1	Methane	0.78	0.78
2	Ethane	0.08	0.08
3	Propane	0.035	0.035
4	Isobutane	0.012	0.012
5	Butane	0.015	0.015
6	Isopentane	0.008	0.008
7	Pentane	0.005	0.005
8	Hexane	0.005	0.005
9	C7+	0.06	0.06

PHASE PROPERTIES

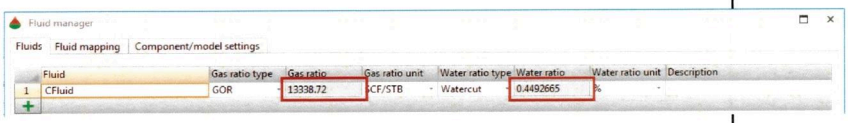
	Parameter	Unit	Mixture	Gas
1	Mole fraction	fract.	1	1
2	Mass fraction	fract.	1	1
3	Volume fraction	fract.	1	1
4	Viscosity	cP		0.0328651
5	Molecular weight		26.30044	26.30044
6	Density (molar)	lb-mole/ft3	0.5905394	0.5905394
7	Density (mass)	lbm/ft3	15.53145	15.53145
8	Enthalpy (molar)	Btu/lb-mole	710.3542	710.3542
9	Entropy (molar)	Btu/lbmol...	-0.007462473	-0.007462473
10	Internal energy (molar)	Btu/lb-mole	-731.0885	-731.0885
11	Gibbs free energy (molar)	Btu/lb-mole	6230.148	6230.148
12	Isochoric specific heat capacity	Btu/lbmol...	14.13554	14.13554
13	Isobaric specific heat capacity	Btu/lbmol...	19.77011	19.77011

10. Change the number of moles of water in the fluid to `1.89`. You observe that a very small amount of a liquid water phase appears. Change the number of moles to `1.88`, and notice that the fluid reverts to a single gas phase.

    The fluid is at the water dew point. It is virtually saturated with water and cannot hold any more water in the vapor phase.

NOTE: Water can be carried along with the gas in the vapor phase or entrained in the gas in droplet form. There is a maximum amount of water vapor that a gas is able to hold at any given temperature and pressure. A gas is completely saturated when it contains the maximum amount of water vapor for the given pressure and temperature conditions.

11. Change the number of moles of water to `1.88`.
12. Close the fluid editor and return to the **Fluids** tab of the **Fluid manager** to see the GOR and Water cut values for the fluid. They should match the values in the figure.



Lesson 2 Gas well deliverability

Based on the analysis for flow data obtained from a large number of gas wells, Rawlins and Schellhardt (1936) presented a relationship between the gas flow rate and pressure drawdown that can be expressed as:

$$Q_{sc} = C(P_R^2 - P_{WF}^2)^n$$

where:

Q_{sc} = Gas rate (mmscf/d)

P_R = Average static reservoir pressure (psia)

P_{WF} = Flowing bottomhole pressure (psia)

C = Flow coefficient (mmscf/d/psi^2)

n = Non-Darcy exponent

The exponent n is intended to account for the additional pressure drop caused by high-velocity gas flow, such as flow caused by turbulence. Depending on the flowing conditions, the exponent n can vary from 1.0 for completely laminar flow to 0.5 for fully turbulent flow.

The performance coefficient C in the equation is included to account for the following parameters:

- Reservoir rock properties
- Fluid properties
- Reservoir flow geometry

This equation is commonly referred to as the *Deliverability* or *Back-pressure* equation. The coefficients of the equation (n and C) are determined from well deliverability tests. When they are determined, the gas flow rate Q_{sc} can be calculated at any bottomhole flowing pressure P_{WF} and IPR curve can be constructed.

Deliverability testing has been used for more than sixty years by the petroleum industry to characterize and determine the flow potential of gas wells.

There are essentially three types of deliverability tests:

- Conventional deliverability (back-pressure)
- Isochronal
- Modified isochronal

Essentially, these tests consist of flowing wells at multiple rates sequentially and measuring the bottomhole flowing pressures as a function of time. The stabilized flow rates and bottomhole pressures are then plotted on a log-log plot and fit with a straight line. The exponent n is calculated as the slope of the line.

The flow coefficient C is calculated from the equation itself using the calculated n value, as well as the rate and flowing bottomhole pressure from one stabilized well test during the flow period.

Exercise 1 Calculate gas well deliverability



In this exercise, you construct a simple gas well model and perform a simulation to calculate the well deliverability.

1. Construct a simple well model with these listed parameters. Any information not explicitly provided should be left to its default value in **PIPESIM**.

Casing data	
Casing ID	8.681 inches
Casing wall thickness	0.472 inches
Casing bottom MD	11,200 ft
Casing roughness	0.001 inches
Tubing data	
Tubing ID	3.476 inches
Tubing wall thickness	0.262 inches
Tubing bottom MD	10,950 ft
Tubing roughness	0.001 inches
Downhole equipment	
Packer depth	10,000 ft
Heat transfer data	
Heat transfer coefficient	2 Btu/(h.degF.ft^2)
Soil temperature at wellhead	30 degF
Completion data	
Completion depth	11,000 feet
IPR model	Well PI
Reservoir pressure	4,600 psia
Reservoir temperature	280 degF
IPR basis	Gas
Gas PI	1E-06 mmscf/d/psi^2

2. On the **Completions** tab, click the **Fluid model** tab.

Exercise 1 Calculate gas well deliverability



In this exercise, you construct a simple gas well model and perform a simulation to calculate the well deliverability.

1. Construct a simple well model with these listed parameters. Any information not explicitly provided should be left to its default value in **PIPESIM**.

Casing data	
Casing ID	8.681 inches
Casing wall thickness	0.472 inches
Casing bottom MD	11,200 ft
Casing roughness	0.001 inches
Tubing data	
Tubing ID	3.476 inches
Tubing wall thickness	0.262 inches
Tubing bottom MD	10,950 ft
Tubing roughness	0.001 inches
Downhole equipment	
Packer depth	10,000 ft
Heat transfer data	
Heat transfer coefficient	2 Btu/(h.degF.ft^2)
Soil temperature at wellhead	30 degF
Completion data	
Completion depth	11,000 feet
IPR model	Well PI
Reservoir pressure	4,600 psia
Reservoir temperature	280 degF
IPR basis	Gas
Gas PI	1E-06 mmscf/d/psi^2

2. On the **Completions** tab, click the **Fluid model** tab.

3. Map the compositional fluid you created in the previous task, to the completion by selecting it from the drop-down list.
4. Save your workspace.
5. Launch the **P/T profile** task from the **Home** tab.
6. Select Gas flow rate as the Calculated Variable.
7. Enter an Outlet Pressure of 800 psi.
8. Leave the Default profile plot set to Elevation vs. Pressure, then click **Run**.
9. Review the Profile results (grid and plot) and record your answers in the table.

Results	
Gas Rate (mmscf/d)	
Flowing bottomhole pressure (psi)	
Bottomhole temperature (degF)	
Wellhead temperature (degF)	



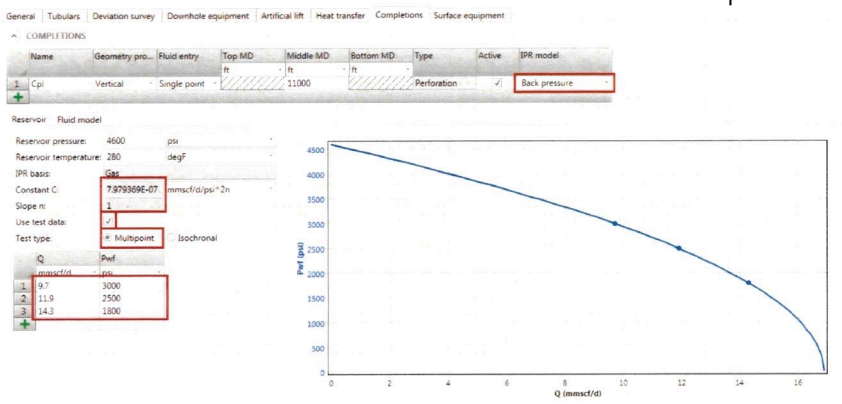
Exercise 2 Calibrate the inflow model using multipoint test data

In this exercise, you use a different IPR model; the Back pressure equation. You calibrate it with multipoint well test data. The C and n parameters are tuned to match the well test data. Continue using the same model.

1. Go to the **Completions** tab of the Well editor.
2. Change the IPR model to Back pressure.
3. On the **Reservoir** tab, select *Use test data*.
4. Set the Test Type to Multipoint.
5. Enter the test data in the following table.

Multipoint test data	
Q_G (mmscf/d)	**Pwf (psi)**
9.7	3,000
11.9	2,500
14.3	1,800

The IPR plot auto-generates, and values of C and n are calculated.



6. Re-run the **P/T profile** task using the same boundary conditions from the previous exercise. Record your answers in the table.

   How do the results from both exercises compare?

Results	Well PI	Calibrated back pressure
Gas Rate (mmscf/d)		
Flowing bottomhole pressure (psi)		
Bottomhole temperature (degF)		
Wellhead temperature (degF)		

Back pressure equation	
Parameter C (mmscf/d/psi^2n)	
Parameter n	



Lesson 3 Erosion prediction

Erosion has been long recognized as a potential source of problems in oil and gas production systems. Erosion can occur in solids-free fluids, but usually, it is caused by entrained solids (typically sand).

Currently, the API 14 E method is the only one supported in **PIPESIM** 2014 for erosion velocity calculations. However, **PIPESIM** Classic versions (2012 and older) support both the API 14 E model and the Salama model.

The API 14 E model comes from the American Petroleum Institute, Recommended Practice, number 14 E. This is a solids-free model that calculates only an erosion velocity (no erosion rate).

The erosion velocity V_e is calculated with the formula:

$$Ve = \frac{C}{\sqrt{\rho_m}}$$

Which you can evaluate with the actual flowing velocity to provide the erosional velocity ratio:

$$EVR = \frac{V_{actual}}{V_e}$$

where

V_{actual} = actual velocity of fluid

V_e = API 14E Erosional velocity limit

r_m = mixture density of fluid (lbm/ft3)

C = empirical constant representing pipe material

EVR = erosional velocity ratio

The following values of C in oilfield units are suggested in literature:

- C = 100 for continuous, non-corrosive, solids-free service
- C = 125 for intermittent, non-corrosive, solids-free service
- C = 150-200 for continuous, corrosive, solids-free service
- C = 250 for intermittent, corrosive, solids-free service

NOTE: Continuous and intermittent corrosive fluids are treated with a corrosion inhibitor or corrosion-resistant material.

The recommended value of C in engineering units, which also is the **PIPESIM** default, is 100. This default value corresponds to 122 in SI units. It has been noted that this is a conservative value.

The current practice for eliminating erosional problems in piping systems is to limit the flow velocity to the flow calculated by this API equation.

Exercise 1 Select a tubing size

In this exercise, you perform a Nodal analysis to select an optimum tubing size. The available tubing sizes have IDs of 2.441 inches, 2.992 inches, 3.476 inches, and 3.958 inches.

Your final decision is based on these criteria:

- Flow rate (The higher the better, until the erosional velocity is reached. This is because more liquid droplets are carried at higher velocities, which increases the erosion risk).
- Erosional velocity ratio (<1)
- Tubing cost (Generally increases with size).

Continue using the model from the previous exercise:

1. Go to **Home > Simulation settings > Erosion/Corrosion**.
2. Confirm that the Erosion velocity constant (C value) of 100 is entered for the API 14e erosion model.
3. Launch the **Nodal analysis** task from the **Home** tab.
4. Select Bottomhole as the nodal point, when prompted.

5. Enter 800 psia as the Outlet Pressure.
6. On the **Sensitivities** tab, enter the available tubing IDs under the Outflow Sensitivity:
   - 2.441 inches
   - 2.992 inches
   - 3.476 inches
   - 3.958 inches
7. Run the model and view the *Profile* results (not the *System* results).
8. Double-click the plot and change the X-axis variable to Erosional Velocity Ratio.

   Which tubing size best satisfies the decision criteria? Choose the smallest size that is free from erosion issues, unless the increase in gas rate is significant (> 5%).

   Selected Tubing Size _____ inch
9. Record the results in the table for the selected tubing size and specify this tubing size in subsequent exercises and procedures.

Results	
Selected Tubing ID (inches)	
Gas rate (mmscf/d)	
Flowing bottomhole pressure (psi)	
Flowing bottomhole temperature (degF)	
Flowing wellhead temperature, degF	
Max Erosional velocity ratio	

Lesson 4 Choke modeling



Wellhead chokes are used to limit production rates to meet surface constraints, protect surface equipment from slugging, avoid sand problems because of high drawdown, and control flow rate to avoid water or gas coning. Placing a choke at the wellhead increases the wellhead pressure and, thus, the flowing bottomhole pressure, which reduces the production rate.

Pressure drop across wellhead chokes is usually significant. Various choke flow models are available for critical (sonic) and sub-critical flow.

Sound waves and pressure waves are both mechanical waves. When the fluid flow velocity in a choke reaches the traveling velocity of sound in the fluid for the in situ condition, the flow is known as *sonic flow*.

Under sonic flow conditions, the pressure wave downstream of the choke cannot go upstream through the choke because the medium (fluid) is traveling in the opposite direction at the same velocity. As a result, a pressure discontinuity exists at the choke, which means that the downstream pressure does not affect the upstream pressure.

Because of the pressure discontinuity at the choke, any change in the downstream pressure cannot be detected from the upstream pressure gauge; likewise, any change in the upstream pressure cannot be detected from the downstream pressure gauge.

Sonic flow is a unique choke feature that is desirable because it stabilizes the well production rate and separator operating conditions.

Sonic flow at a choke depends on the downstream-to-upstream pressure ratio (Figure 19). If this pressure ratio is less than a critical pressure ratio, sonic (critical) flow exists. If this pressure ratio is greater than, or equal to, the critical pressure ratio, sub-sonic (sub-critical) flow exists.

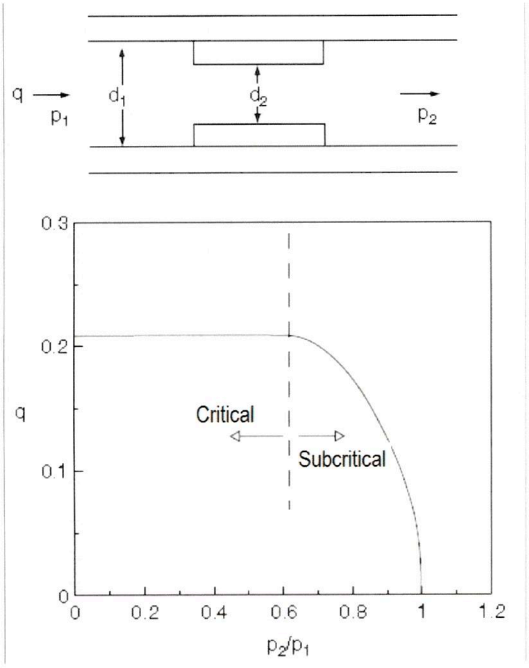


Figure 19 Effect on flow rate of pressure decrease across a choke

The critical pressure ratio is approximately 0.55 for natural gas, and a similar constant is used for oil flow.

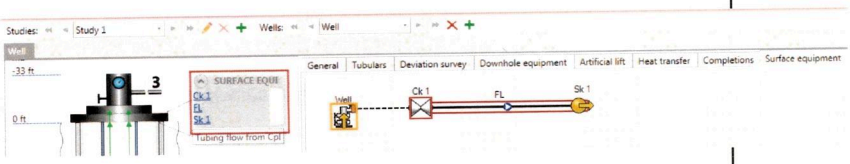
In some wells, chokes are installed in the lower section of tubing strings. This choke arrangement reduces wellhead pressure and enhances oil production rate as a result of gas expansion in the tubing string. For gas wells, a downhole choke can reduce the risk of gas hydrates. A major disadvantage of using a downhole choke is that replacing it is costly.

Exercise 1 Model a flowline and choke

In this exercise, you modify the well from the previous task to include a flowline and choke. You then use the gas rate calculated in the previous exercise to determine the choke bean size that results in a manifold (end of flowline) pressure of 710 psi.

Continue using the model from the previous exercise.

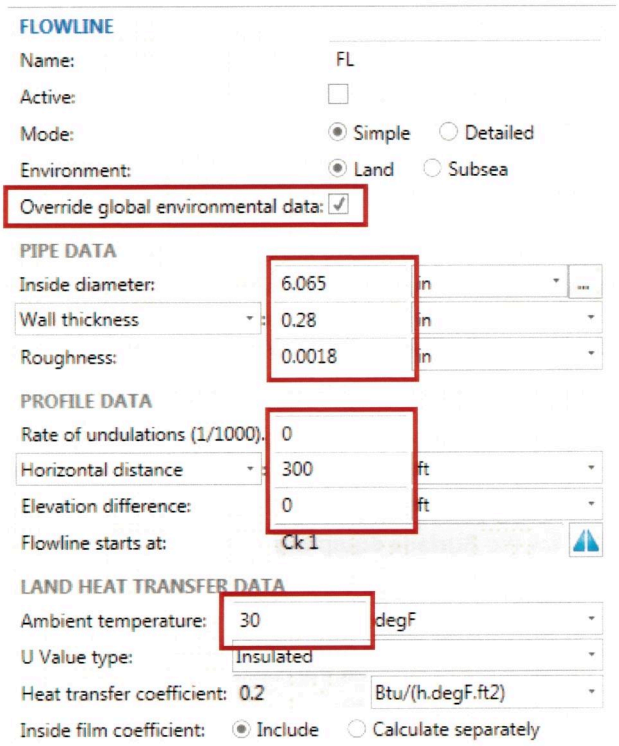
1. On the **Tubulars** tab of the well editor, make sure that the tubing ID is set to the optimum tubing size determined from the previous exercise (3.476 inches).
2. Click the **Surface equipment** tab of the well editor. Then click the main **Insert** tab to expose the equipment that can be added.
3. Insert a choke and a sink. Connect the wellhead to the choke using a connector and the choke to the sink using a flowline, as shown in the figure.



4. Click the choke and enter a Bean size of `1 in`.

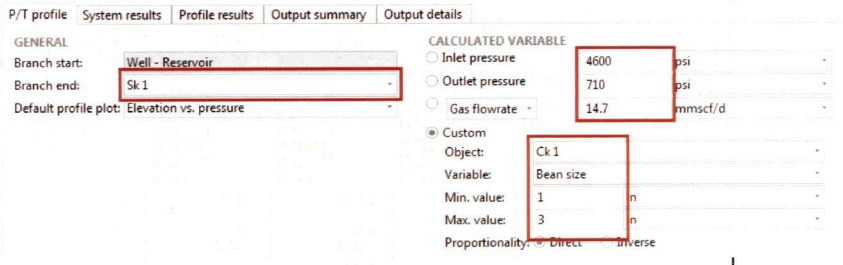
NOTE: You can enter any Bean size. The sensitivity that runs varies the bean size to determine the correct size for the desired outlet pressure.

5. Click the flowline and configure it as shown in the figure.



6. Verify that the *Override global environmental data* box is selected.
7. Enter an Ambient temperature of `30` degF.
8. Launch the P/T Profile task from the **Home** tab. Change the branch end to the Sink to ensure that the flowline and choke are included in the simulated profiles.
9. Make these selections from the drop-down lists:
   - Calculated Variable: Custom
   - Object: Ck1 (Choke)
   - Variable: Bean size
10. Enter a bean size range of `1` to `3` inches and leave the Proportionality set to *Direct*.

11. Change the Outlet pressure (at the Sink) to `710` psi and enter the Gas flow rate obtained from the previous exercise (14.7 mmscf/d).
12. The P/T profile dialog box should look like the figure.

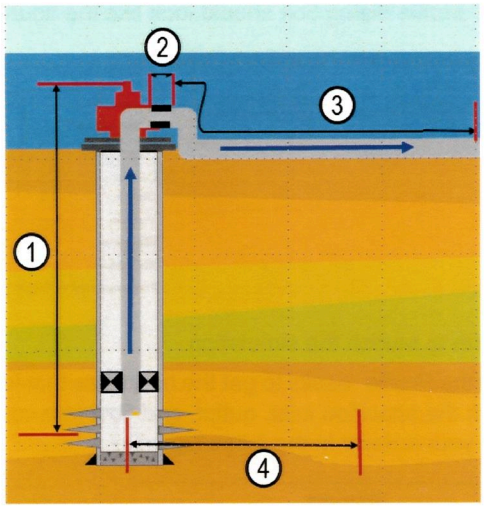


13. Click **Run** to launch the simulation.
14. Review the Profile results to get the bean size that is required to match the specified inlet, outlet, and flow rate conditions. Record your answer in the following table.

Results	
Po = 710 psia	
Choke size (inches)	

15. Click the choke in the **Surface equipment** tab of the well editor and enter the calculated choke bean size (from the previous step).
16. Run the P/T profile task with outlet pressure as the Calculated Variable. Review the Profile results and verify that the calculated sink pressure is 710 psi.

17. Review the System and Profile results (plot and grid) and determine the individual pressure drops for the reservoir, tubing, choke, and flowline and record them in the table.



1 ΔP Tubing

2 ΔP Wellhead choke

3 ΔP Flowline

4 ΔP Reservoir

Results	
Updated choke size = 1.50 inches	
Static reservoir pressure (psi)	
Flowing bottomhole pressure (psi)	
Flowing wellhead pressure (psi)	
Flowing pressure immediately downstream of choke (psi)	
Outlet pressure (psi)	
Pressure losses across system	
Δ Reservoir (psi)	
Δ Tubing (psi)	
Δ Choke (psi)	
Δ Flowline (psi)	

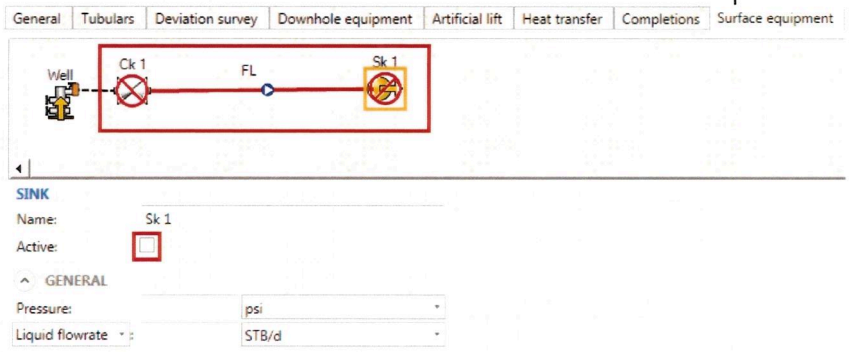
Exercise 2 Predict future production rates

In this exercise, you estimate future gas production rates based on the expected reservoir pressure decline with time. You do this with the System analysis task.

Continue using the model from the previous exercise.

1. You must deactivate the equipment downstream of the wellhead because you run this simulation up to the wellhead.
2. On the **Surface equipment** tab of the well editor, deactivate the Choke, Flowline, and Sink by clicking each of them, one at a time, and clear their *Active* boxes.

NOTE: Right-click each component and select **Deactivate**. These objects are highlighted in red to indicate they are inactive, as shown in the figure.



3. Launch the System Analysis task.
4. Select Gas flow rate as the Calculated variable.
5. Set the Outlet pressure (wellhead) to 800 psi.
6. In the *X-axis* column, select *System Data* as the sensitivity object and *Inlet pressure* as the sensitivity variable.
7. Enter the following Inlet (reservoir) pressures:
   - 4,600 psi
   - 4,200 psi
   - 3,800 psi
   - 3,400 psi

8. Run the model and record your results in the table.

Results	
Reservoir pressure (psi)	**Gas rate (mmscf/d)**
4600	
4200	
3800	
3400	



Lesson 5 Liquid loading

Gas wells usually produce with liquid water and/or condensate in the form of mist droplets or a film along the pipe walls. As the gas flow velocity in the well drops because of reservoir depletion, the carrying capacity of the gas decreases. When the gas velocity drops below a critical level, the gas is unable to lift the liquids and they begin to accumulate in the wellbore. This accumulation is known as *liquid loading*.

Liquid loading increases the flowing bottomhole pressure, which reduces the gas production rate. A lower gas production rate implies a lower gas velocity, which ultimately, causes the well to stop producing or die.

Turner droplet model

In gas wells operating in the annular-mist flow regime, liquids flow as individual particles (droplets) in the gas core and as a liquid film along the tubing wall.

By analyzing a large database of producing gas wells, Turner found that a force balance performed on a droplet could predict if the liquids would flow upwards (drag forces) or downwards (gravitational forces).

If the gas velocity is above a critical velocity, the drag force lifts the droplet; otherwise, the droplet falls and liquid loading occurs (Figure 20).

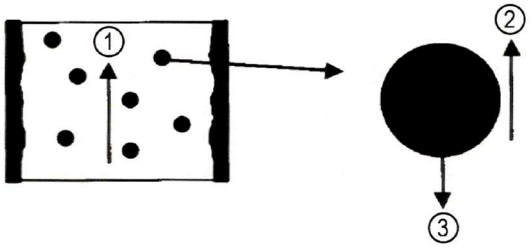


Figure 20 Turner Droplet model

1 Gas

2 Drag

3 Gravity

When the drag on a droplet is equal to its weight, the gas velocity is at the critical velocity. Theoretically, at the critical velocity, the droplet would be suspended in the gas stream, moving n upward nor downward. Below the critical velocity, the droplet falls and liquids accumulate in the wellbore.

In practice, the critical gas velocity is generally defined as the minimum gas velocity in the tubing string required to move droplets upward.

The general form of Turner's equation is given by:

$$V_t = \frac{1.593\sigma^{\frac{1}{4}}(\rho_l - \rho_g)^{\frac{1}{4}}}{\rho_g{}^{\frac{1}{2}}}$$

where:

V_t = Terminal velocity of liquid droplet (ft/s)

σ = Interfacial tension (dynes/cm)

ρ_l = Liquid phase density (lbm/ft^3)

ρ_g = Gas phase density (lbm/ft^3)



NOTE: The Turner equation applies to vertical or near vertical uphill flow and assumes a continuous gas phase with small dispersed liquid droplets entrained in it. **PIPESIM** does not calculate the liquid loading in pipe sections where these conditions are not met.

Liquid loading calculations are performed in every operation and are available for review in plots and reports. Output files are generated upon request in **PIPESIM 2014** by selecting the *Show engine output files* option under **Workspace > Options > Advanced > Engine options**.

PIPESIM calculates a Liquid Loading Velocity Ratio (LLVR) that), which is the minimum lift velocity (terminal/critical velocity) calculated by the Turner equation, divided by the actual mixture velocity. A LLVR >1 indicates a liquid loading problem because the fluid is flowing at a velocity lower than the minimum velocity required to lift the liquids and prevent loading.

The Liquid Loading Gas Rate line can be displayed on the Nodal analysis system plot when the X-axis is configured to display gas rate. For every point on the outflow curve, the value of the Liquid Loading Velocity Ratio is calculated and the liquid loading gas rate line is plotted at the specific rate where the liquid loading velocity ratio is equal to 1.



NOTE: The reported value for the Liquid Loading Gas Rate line comes from interpolation of the outflow curve between two points, one with a velocity ratio below 1 and another with a velocity ratio above 1. As a result, the accuracy of the results depends on the number of points on the outflow curve.

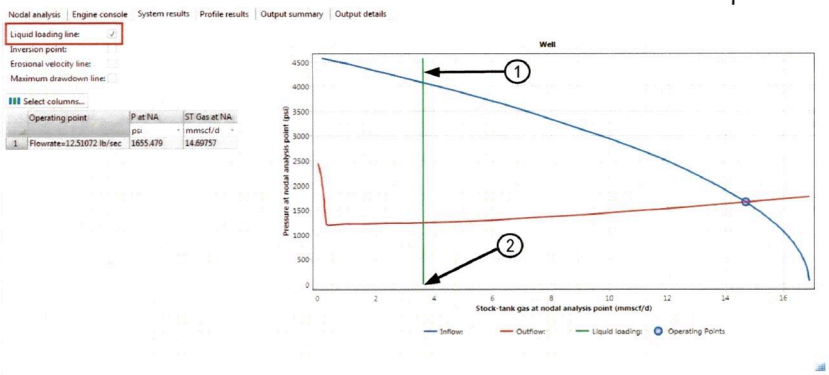


Exercise 1 Determine a critical gas rate to prevent well loading

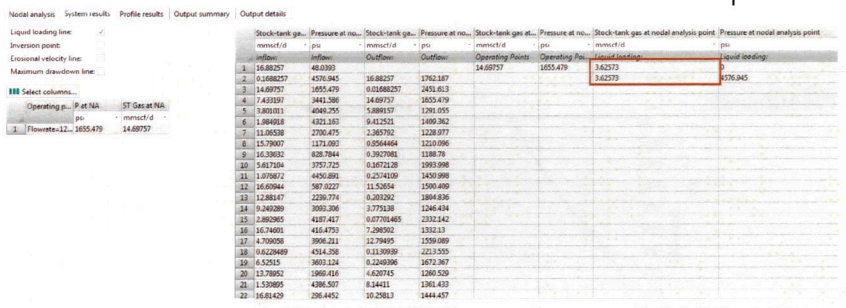
In this exercise, you perform a Nodal Analysis to determine the critical gas rate that prevents liquid loading, and you validate the result by reviewing the liquid loading velocity ratio along the profile.

Continue using the model from the previous exercise.

1. Launch the Nodal Analysis task and delete any Sensitivities that are present.
2. Set the Outlet Pressure to 800 psia and run the model.
3. On the **System results** tab, select the box for the *Liquid loading line*. The plot appears with the Liquid loading line (1) and the Critical gas rate (2).



4. Click the **View data in a table** icon at the bottom right of the plot and extract the critical gas rate.

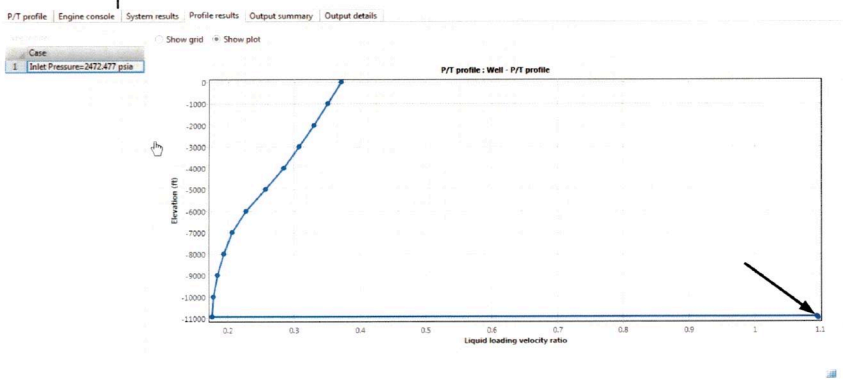


Results	
Critical gas rate (Nodal analysis plot) (mmscf/d)	3.63

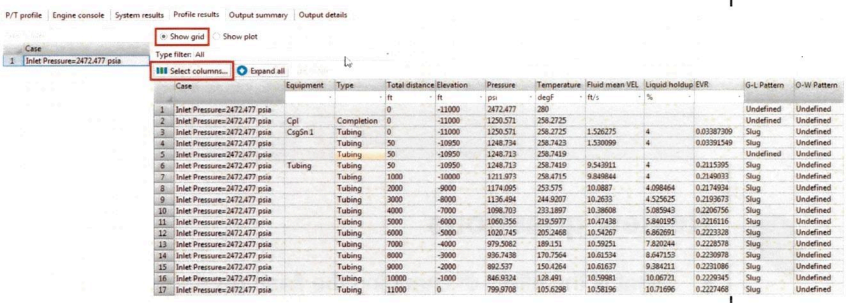
5. For this well scenario, the operating flow rate (14.7 mmscf/d) is far above the critical gas rate to avoid liquid loading (3.63 mmscf/d), so the well does not have liquid loading issues at these conditions.

6. Validate the critical gas rate from the nodal analysis plot by running a P/T profile task at the same conditions (flow rate and outlet pressure).
   a. Launch the P/T profile task.
   b. Select Inlet pressure as the Calculated variable.
   c. Enter a value of `800` psia as the Outlet pressure.
   d. Enter the critical gas flow rate value (`3.63` mmscf/d), from the previous step, as the *Gas flow rate*.
   e. Run the task.
   f. Double-click the Profile results plot and change the X-axis variable to display the *Liquid loading velocity ratio*.
   g. Verify that n the outflow section, which is where the liquid loading is calculated. (This section consists of the tubing and a short section of the casing up to the mid-perforation).
   h. Verify that the maximum liquid loading velocity ratio (LLVR) at the critical gas flow rate of 3.63 mmscf/d, is approximately equal to 1. This means the P/T profile results are consistent with the Nodal analysis results. (Refer to the figure.)

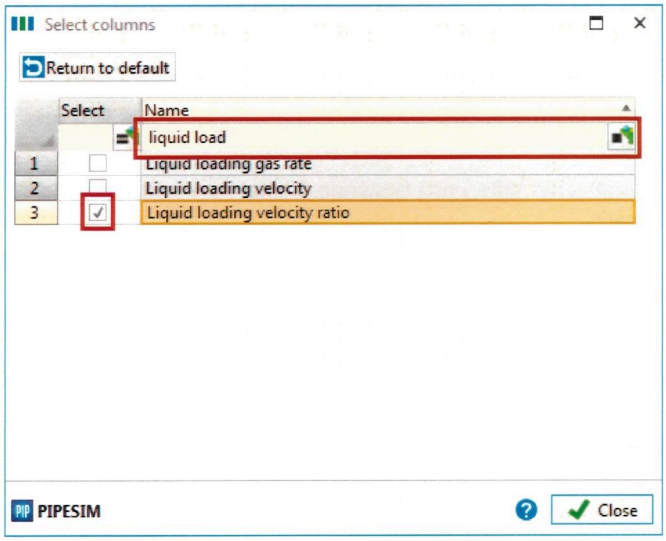
      In the plot, where is the LLVR highest and why?



7. To see the LLVR in the grid results, on the **Profile** results tab:
   a. Select **Show grid**, then click **Select columns**.
   b. Filter to the variable Liquid loading velocity ratio by entering its first few letters, as shown in the figure.
   c. Select the box beside *Liquid loading velocity ratio* and click **Close**.

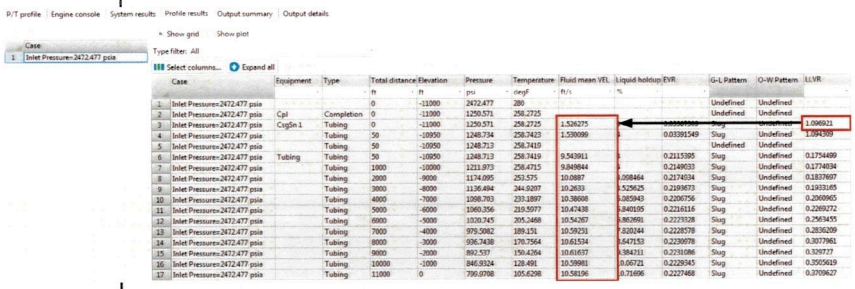


	Case	Equipment	Type	Total distance	Elevation	Pressure	Temperature	Fluid mean VEL	Liquid holdup	EVR	G-L Pattern	O-W Pattern
				ft	ft	psi	degF	ft/s	%			
1	Inlet Pressure=2472.477 psia			0	-11000	2472.477	280				Undefined	Undefined
2	Inlet Pressure=2472.477 psia	Cpl	Completion	0	-11000	1250.571	258.2725				Undefined	Undefined
3	Inlet Pressure=2472.477 psia	CsgSn 1	Tubing	0	-11000	1250.571	258.2725	1.526275	4	0.03387309	Slug	Undefined
4	Inlet Pressure=2472.477 psia		Tubing	50	-10950	1248.734	258.7423	1.530099	4	0.03391549	Slug	Undefined
5	Inlet Pressure=2472.477 psia		Tubing	50	-10950	1248.713	258.7419				Undefined	Undefined
6	Inlet Pressure=2472.477 psia	Tubing	Tubing	50	-10950	1248.713	258.7419	9.543911	4	0.2115395	Slug	Undefined
7	Inlet Pressure=2472.477 psia		Tubing	1000	-10000	1211.973	258.4715	9.849844	4	0.2149033	Slug	Undefined
8	Inlet Pressure=2472.477 psia		Tubing	2000	-9000	1174.095	253.575	10.0887	4.098464	0.2174934	Slug	Undefined
9	Inlet Pressure=2472.477 psia		Tubing	3000	-8000	1136.494	244.9207	10.2633	4.525625	0.2193673	Slug	Undefined
10	Inlet Pressure=2472.477 psia		Tubing	4000	-7000	1098.703	233.1897	10.38608	5.085943	0.2206756	Slug	Undefined
11	Inlet Pressure=2472.477 psia		Tubing	5000	-6000	1060.256	219.5977	10.47438	5.840195	0.2216116	Slug	Undefined
12	Inlet Pressure=2472.477 psia		Tubing	6000	-5000	1020.745	205.2468	10.54267	6.862691	0.2223328	Slug	Undefined
13	Inlet Pressure=2472.477 psia		Tubing	7000	-4000	979.5082	189.151	10.59251	7.820244	0.2228578	Slug	Undefined
14	Inlet Pressure=2472.477 psia		Tubing	8000	-3000	936.7438	170.7564	10.61534	8.647153	0.2230978	Slug	Undefined
15	Inlet Pressure=2472.477 psia		Tubing	9000	-2000	892.537	150.4264	10.61637	9.384211	0.2231086	Slug	Undefined
16	Inlet Pressure=2472.477 psia		Tubing	10000	-1000	846.9324	128.491	10.59981	10.06721	0.2229345	Slug	Undefined
17	Inlet Pressure=2472.477 psia		Tubing	11000	0	799.9708	105.6298	10.58196	10.71696	0.2227468	Slug	Undefined



8. The grid should appear, updated with LLVR.

9. The Liquid loading velocity (the last column on the right) is highest at the bottomhole where the pressure is highest and the fluid velocity is lowest.



	Case	Equipment	Type	Total distance	Elevation	Pressure	Temperature	Fluid mean VEL	Liquid holdup	EVR	G-L Pattern	O-W Pattern	LLVR
				ft	ft	psi	degF	ft/s	%				
1	Inlet Pressure=2472.477 psia			0	-11000	2472.477	280				Undefined	Undefined	
2	Inlet Pressure=2472.477 psia	Cpl	Completion	0	-11000	1250.571	258.2725				Undefined	Undefined	
3	Inlet Pressure=2472.477 psia	CogSn 1	Tubing	0	-11000	1250.571	258.2725	1.526275			Slug	Undefined	1.096921
4	Inlet Pressure=2472.477 psia		Tubing	50	-10950	1248.734	258.7423	1.530086	1	0.03391549	Slug	Undefined	1.094389
5	Inlet Pressure=2472.477 psia		Tubing	50	-10950	1248.713	258.7419				Undefined	Undefined	
6	Inlet Pressure=2472.477 psia	Tubing	Tubing	50	-10950	1248.713	258.7419	9.543911	1	0.2115395	Slug	Undefined	0.1754499
7	Inlet Pressure=2472.477 psia		Tubing	1000	-10000	1211.973	258.4715	9.849844	1	0.2149033	Slug	Undefined	0.1774834
8	Inlet Pressure=2472.477 psia		Tubing	2000	-9000	1174.095	253.575	10.0887	1.008464	0.2174034	Slug	Undefined	0.1837697
9	Inlet Pressure=2472.477 psia		Tubing	3000	-8000	1136.494	244.9207	10.2633	1.025625	0.2193673	Slug	Undefined	0.1933165
10	Inlet Pressure=2472.477 psia		Tubing	4000	-7000	1098.703	233.1897	10.38608	1.085943	0.2206756	Slug	Undefined	0.2060965
11	Inlet Pressure=2472.477 psia		Tubing	5000	-6000	1060.356	219.5977	10.47438	1.840195	0.2216116	Slug	Undefined	0.2269272
12	Inlet Pressure=2472.477 psia		Tubing	6000	-5000	1020.745	205.2466	10.54267	1.865691	0.2223328	Slug	Undefined	0.2583455
13	Inlet Pressure=2472.477 psia		Tubing	7000	-4000	979.5082	189.151	10.59251	7.820244	0.2228578	Slug	Undefined	0.2836209
14	Inlet Pressure=2472.477 psia		Tubing	8000	-3000	936.7438	170.7564	10.61534	1.647153	0.2230978	Slug	Undefined	0.3077961
15	Inlet Pressure=2472.477 psia		Tubing	9000	-2000	892.537	150.4264	10.61637	1.384211	0.2231086	Slug	Undefined	0.326727
16	Inlet Pressure=2472.477 psia		Tubing	10000	-1000	846.9324	128.491	10.59881	1.60721	0.2228945	Slug	Undefined	0.3505619
17	Inlet Pressure=2472.477 psia		Tubing	11000	0	799.9708	105.6298	10.58196	1.71696	0.2227468	Slug	Undefined	0.3709627

Review questions

- What is the effect of changing the flash package used in **PIPESIM**?
- How does the viscosity of an emulsion compare to the viscosity of the two phases from which it forms?
- What are the three types of gas well deliverability tests?
- What are the criteria for optimum tubing selection in a gas well?
- What is the difference between critical flow and sub-critical flow?
- What effect does changing the downstream pressure have if the choke is in critical flow?
- What are the advantages and disadvantages of using downhole chokes instead of wellhead chokes?

Summary

In this module, you learned about:

- building a simple well model
- calibrating the inflow model
- performing a nodal analysis at bottomhole
- performing a system analysis
- selecting optimum tubing size

- erosion prediction
- modeling flowline and choke performance
- liquid loading in gas wells

NOTES

Module 5 Horizontal well design

This module shows you how to use **PIPESIM** to design a horizontal well and evaluate horizontal well performance.

Learning objectives



After completing this module, you will know how to:

- determine the production from a horizontal well
- optimize horizontal well length

Lesson 1 Inflow performance for horizontal completions



The main advantage of a horizontal well, as compared to a vertical well, is its enhanced reservoir contact area and subsequent enhanced well productivity.

There are four scenarios in which a horizontal well can be a superior alternative to a vertical well (Cooper, 1988):

Thin reservoirs	The increased area of contact of the horizontal well with the reservoir is reflected by the Productivity Index (PI). Typically, the PI for a horizontal well can be increased by a factor of 4, compared to a vertical well penetrating the same reservoir.
Heterogeneous reservoirs	When heterogeneous reservoirs exist, the horizontal well can effectively intersect isolated productive zones that you can miss. A horizontal well also can intersect natural vertical fractures in a reservoir.

Reduce water/ gas coning	A horizontal well allows you to produce the same rate at a lower pressure drawdown than a vertical well. This helps delay the onset of water or gas breakthrough. Even though the production per unit well length can be small, the long well length provides high production rates.
Vertical permeability	If the ratio of vertical permeability to horizontal permeability is high, a horizontal well can produce more economically than a vertical well.

These IPR methods are available in **PIPESIM** for designing horizontal wells:

Steady state production	The simplest forms of horizontal well productivity equations are the steady-state analytical solutions, which assume that the pressure at any point in the reservoir does not change with time. According to Joshi (1991), even though few reservoirs operate under steady-state conditions, steady-state solutions are widely used for these reasons: • Analytical derivation is easy. • The concepts of expanding drainage boundary over time, effective wellbore radius, and shape factors allow for a straightforward conversion to transient or pseudo-steady state results. • Steady-state mathematical results can be verified experimentally. The steady-state distributive productivity index equation is based upon Joshi's *Review of Horizontal and Drainhole Technology*, SPE 16868. The equation is based on the assumption that the horizontal well drains an ellipsoidal volume around the wellbore of length, L.

Pseudo-Steady State Production

Often, it is desirable to calculate productivity from a reservoir with unique boundary conditions, such as a gas cap or bottom water drive, finite drainage area, or well location. In these instances, pseudo-steady state equations are employed.

Pseudo-steady state conditions begin when the pressure disturbance created by the well is felt at the boundary of the well drainage area.

The Pseudo-steady state productivity index is based on Babu and Odeh's SPE paper 18298. It is recommended that you read this reference before applying the equation. The equation is based upon the Pseudo-steady state IPR well model applied to a rectangular drainage area.

Distributed Productivity Index Method

This option uses a straight line PI value for liquid or gas. The distributed productivity index relationship is

$Q = J(P_{ws} - P_{wf})L$ for liquid reservoirs

OR

$Q = J(P_{ws}^2 - P_{wf}^2)L$ for gas reservoirs,

where:

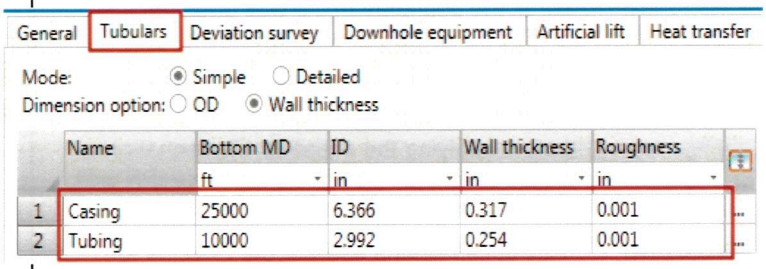
J = distributed productivity index.

PIPESIM uses a technique in which the horizontal completion is subdivided into vertical cross sections and flow is treated independently from other cross sections. This multiple source concept leads to a pressure gradient from the blind-end (toe) to the producing-end (heel) that, if neglected, results in over-predicting deliverability.

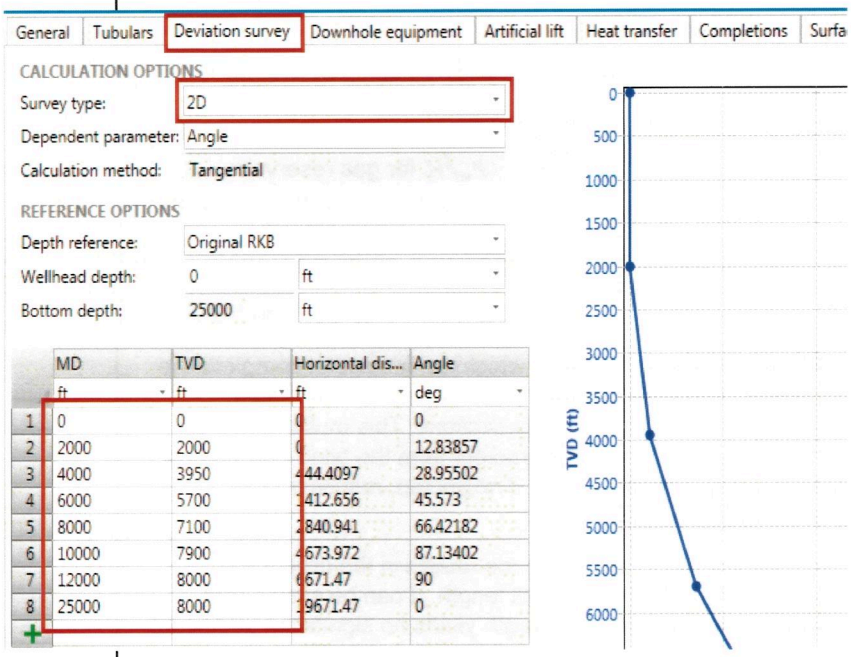
The reduced drawdown at the toe results in the production leveling off as a function of well length. It can be shown that drilling beyond an optimum length yields no significant additional production.

Exercise 1 Horizontal well performance

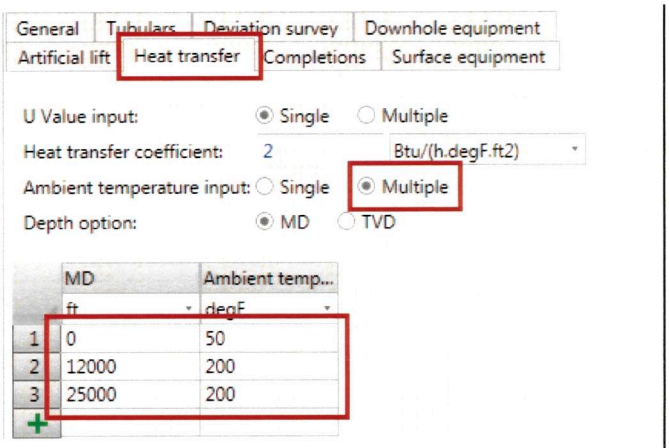
1. Create a new, well-centric workspace.
2. Construct a horizontal well model with the physical parameters shown in the figures.
3. Enter the parameters on four tabs in the well editor: **Tubulars**, **Deviation survey**, **Heat transfer**, and **Completions,** as shown.

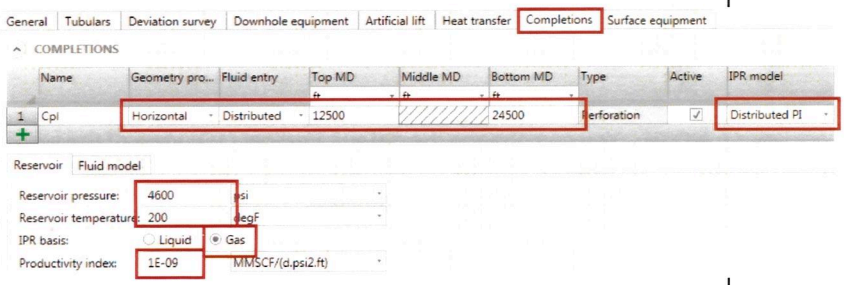


	Name	Bottom MD	ID	Wall thickness	Roughness
		ft	in	in	in
1	Casing	25000	6.366	0.317	0.001
2	Tubing	10000	2.992	0.254	0.001

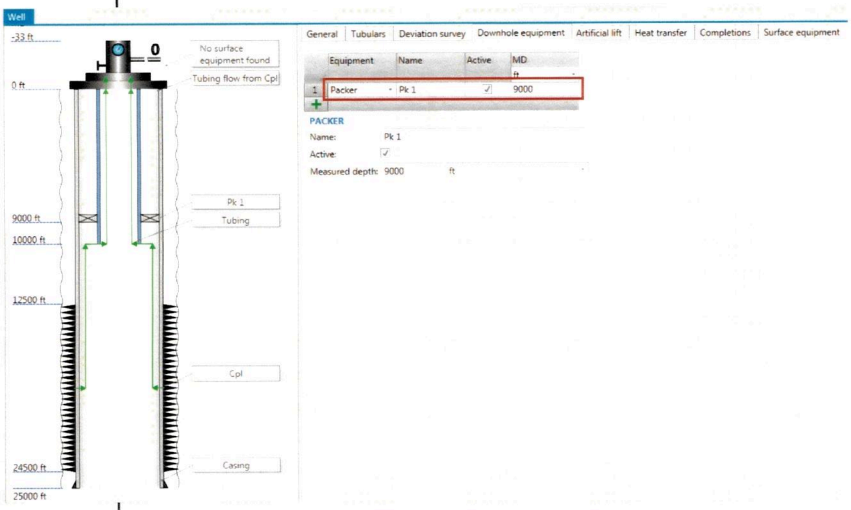


	MD	TVD	Horizontal dis...	Angle
	ft	ft	ft	deg
1	0	0	0	0
2	2000	2000	0	12.83857
3	4000	3950	444.4097	28.95502
4	6000	5700	1412.656	45.573
5	8000	7100	2840.941	66.42182
6	10000	7900	4673.972	87.13402
7	12000	8000	6671.47	90
8	25000	8000	19671.47	0


General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Completions
Surface equipment
U Value input:
Single
Multiple
Heat transfer coefficient:
2
Btu/(h.degF.ft2)
Ambient temperature input:
Single
Multiple
Depth option:
MD
TVD
MD
Ambient temp...
ft
degF
1
0
50
2
12000
200
3
25000
200

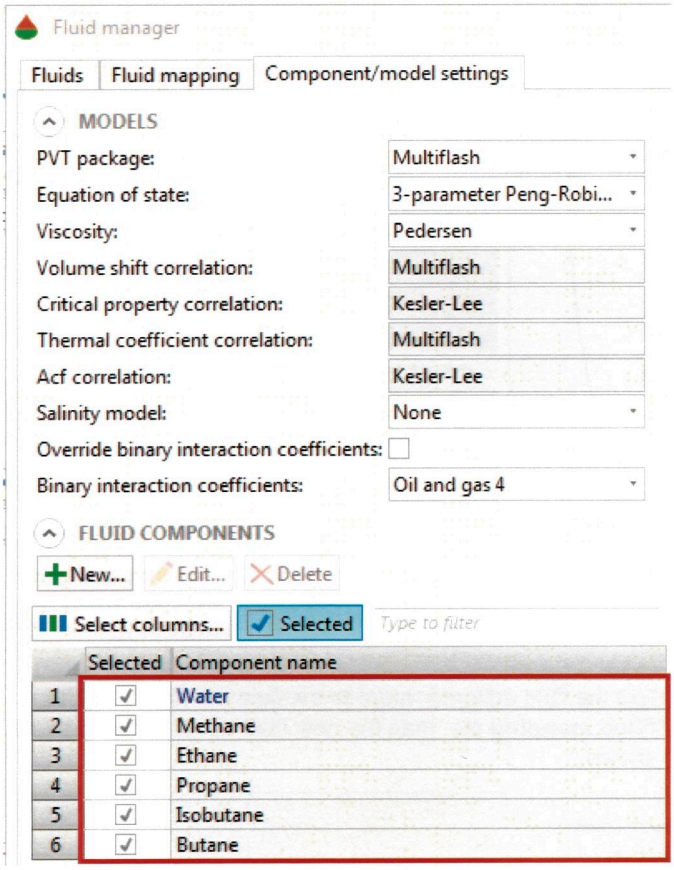

General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Completions
Surface equipment
COMPLETIONS
Name
Geometry pro...
Fluid entry
Top MD
Middle MD
Bottom MD
Type
Active
IPR model
1
Cpl
Horizontal
Distributed
12500
24500
Perforation
Distributed PI
Reservoir
Fluid model
Reservoir pressure:
4600
psi
Reservoir temperature:
200
degF
IPR basis:
Liquid
Gas
Productivity index:
1E-09
MMSCF/(d.psi2.ft)

4. On the **Downhole equipment** tab, add a packer at 9,000 ft to restrict flow to the tubing. The well schematic appears.

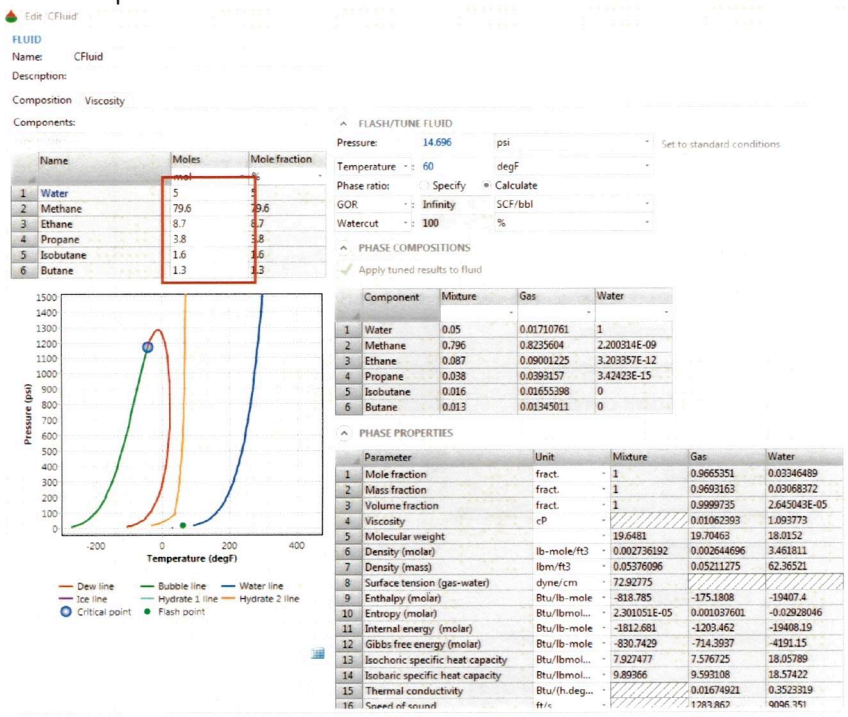


NOTE: You can change the default Vertical (1D) view of the well schematic to two-dimensional (2D) on the **Format** tab, in the Style group of features.

5. Create a compositional fluid using the Multiflash PVT package. Leave all the default model selections and add the components shown in the figure, on the **Component/model settings** tab of the **Compositional fluid manager**.



6. On the **Fluids** tab, add a new row and double-click it to launch the fluid editor. Enter the composition shown in the figure.

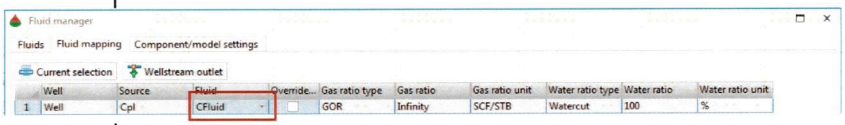


	Name	Moles	Mole fraction
		mol	%
1	Water	5	5
2	Methane	79.6	79.6
3	Ethane	8.7	8.7
4	Propane	3.8	3.8
5	Isobutane	1.6	1.6
6	Butane	1.3	1.3

	Component	Mixture	Gas	Water
1	Water	0.05	0.01710761	1
2	Methane	0.796	0.8235604	2.200314E-09
3	Ethane	0.087	0.09001225	3.203357E-12
4	Propane	0.038	0.0393157	3.42423E-15
5	Isobutane	0.016	0.01655398	0
6	Butane	0.013	0.01345011	0

	Parameter	Unit	Mixture	Gas	Water
1	Mole fraction	fract.	1	0.9665351	0.03346489
2	Mass fraction	fract.	1	0.9693163	0.03068372
3	Volume fraction	fract.	1	0.9999735	2.645043E-05
4	Viscosity	cP		0.01062393	1.093773
5	Molecular weight		19.6481	19.70463	18.0152
6	Density (molar)	lb-mole/ft3	0.002736192	0.002644696	3.461811
7	Density (mass)	lbm/ft3	0.05376096	0.05211275	62.36521
8	Surface tension (gas-water)	dyne/cm	72.92775		
9	Enthalpy (molar)	Btu/lb-mole	-818.785	-175.1808	-19407.4
10	Entropy (molar)	Btu/lbmol...	2.301051E-05	0.001037601	-0.02928046
11	Internal energy (molar)	Btu/lb-mole	-1812.681	-1203.462	-19408.19
12	Gibbs free energy (molar)	Btu/lb-mole	-830.7429	-714.3937	-4191.15
13	Isochoric specific heat capacity	Btu/lbmol...	7.927477	7.576725	18.05789
14	Isobaric specific heat capacity	Btu/lbmol...	9.89366	9.593108	18.57422
15	Thermal conductivity	Btu/(h.deg...		0.01674921	0.3523319
16	Speed of sound	ft/s		1283.862	9096.351

7. Exit the fluid editor to return to the fluid manager. On the **Fluid mapping** tab, map the new fluid to the completion, as shown.



	Well	Source	Fluid	Override...	Gas ratio type	Gas ratio	Gas ratio unit	Water ratio type	Water ratio	Water ratio unit
1	Well	Cpl	CFluid		GOR	Infinity	SCF/STB	Watercut	100	%

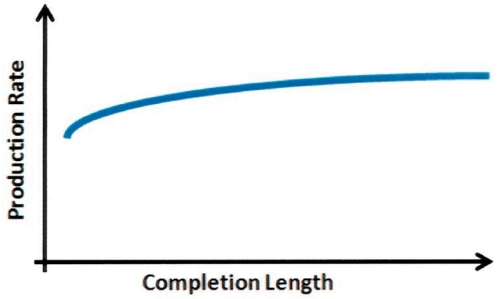
8. Run a **P/T profile** task with an outlet pressure of 300 psi to determine the production rate from the well.

   Production Rate (Gas) = _____ mmscf/d

9. Earlier in the lesson, it was mentioned that drilling beyond an optimum length would yield no significant additional production from a horizontal well.

   Verify this statement by sensitizing the length of the completion and re-running the P/T profile task.

10. Plot the data in Excel to see the validity of the statement. You should see a result similar to the plot shown in the figure.



Review questions



- What are the advantages of a horizontal well over a vertical well?
- What are the basic completion models in **PIPESIM** for horizontal wells?
- Explain the shape of a horizontal well length versus a production rate curve.

Summary

In this module, you learned about:

- constructing a horizontal well
- the effect on production of horizontal well length

NOTES

Module 6 Subsea tieback design

The offshore frontier poses some of the greatest technical challenges facing the oil and gas industry, particularly as we venture into ever deeper waters and more remote locations. Development costs can be substantial, and to be economically viable, many new production systems must be designed to accommodate subsea multiphase flow across long distances.

Managing costs over extended distances introduces many complex risks, and reliability becomes a key concern because of the potential for downtime and high intervention costs. Characterizing and managing these risks requires detailed multidisciplinary engineering analysis and has led to the emergence of a new field known as Flow Assurance.

The design of subsea tiebacks requires multiphase flow simulation to assure that fluids are safely and economically transported from the bottom of a well all the way to the downstream processing facility.

Four flow assurance issues are discussed in this module: hydrates, heat loss, erosion, and liquid slugging.

Learning objectives



After completing this module, you will know how to:

- size a subsea tieback line and riser
- determine the pipeline insulation requirements
- determine methanol injection to inhibit hydrate formation
- screen the results for severe slugging at the riser base
- size a slug catcher

Lesson 1 Flow assurance considerations

This case study involves a client with an existing operation in the North Sea.

Scenario: The client has made a new gas condensate discovery and plans to develop it using four wells producing into a subsea manifold, flowing through a subsea tieback, and up a riser to an existing platform. The oil and gas are separated on the platform, with the oil pumped to shore and the gas compressed to shore.

NOTE: A *tieback* is the pipeline on the ocean floor that connects the production from the new wells to the existing facility.

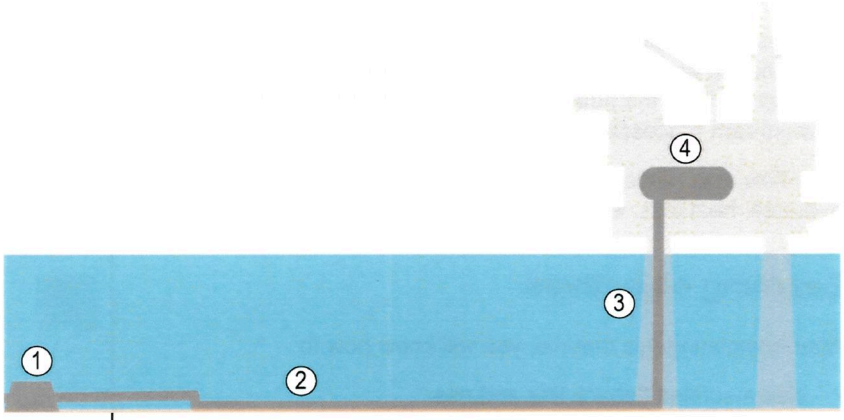


Figure 21 Case study in the North Sea

1. Subsea manifold
2. Subsea tieback
3. Riser
4. Host

Exercise 1 Size the subsea tieback and riser

In this exercise, you create a compositional fluid, build a network model, and determine the optimum diameters for the subsea tieback and the riser. Use the **PIPESIM** defaults if no specification is provided.

1. Launch **PIPESIM** and create a new, network-centric workspace. When you create a network-centric workspace, it launches in the **Network** perspective. You can easily switch from network to well perspective, and vice versa, by selecting the perspective from the drop-down list.



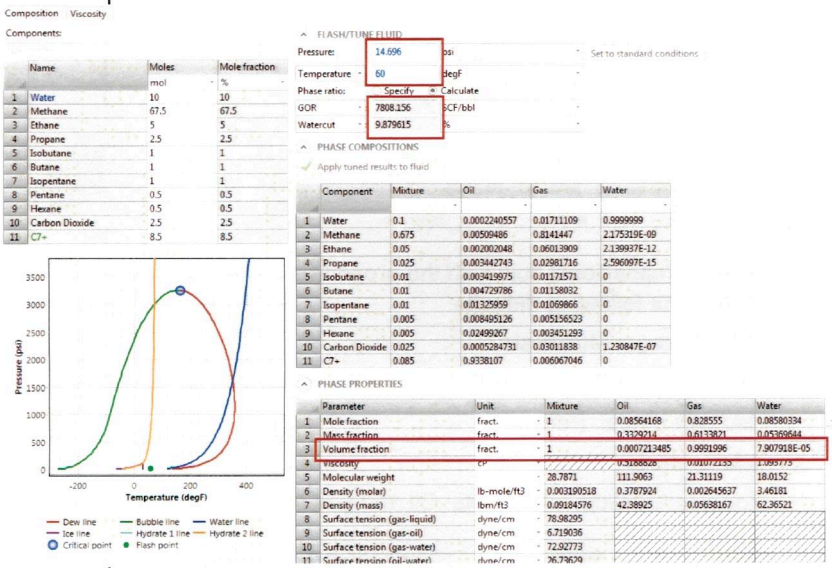
2. Create a compositional fluid with the properties listed in the table. Use the following defaults:
   - PVT package: Multiflash
   - Equation of State: 3-parameter Peng-Robinson
   - Viscosity model: Pedersen
   - Salinity model: None
   - Binary interaction coefficients: Oil and gas 4

Component	Moles		Component	Moles
Water	10.0		Isopentane	1.0
Methane	67.5		Pentane	0.5
Ethane	5.0		Hexane	0.5
Propane	2.5		Carbon dioxide	2.5
Isobutane	1.0		C7+	8.5
Butane	1.0			

Pseudocomponent	Molecular weight	Specific gravity
C7+	115	0.683

NOTE: If you do not have the license features required to use Multiflash, select the **ECLIPSE 300** PVT package. However, you cannot perform some of the steps in this tutorial (for example hydrates precipitation) without the **Multiflash** PVT package.

The fluid properties at standard conditions are shown in the figure.

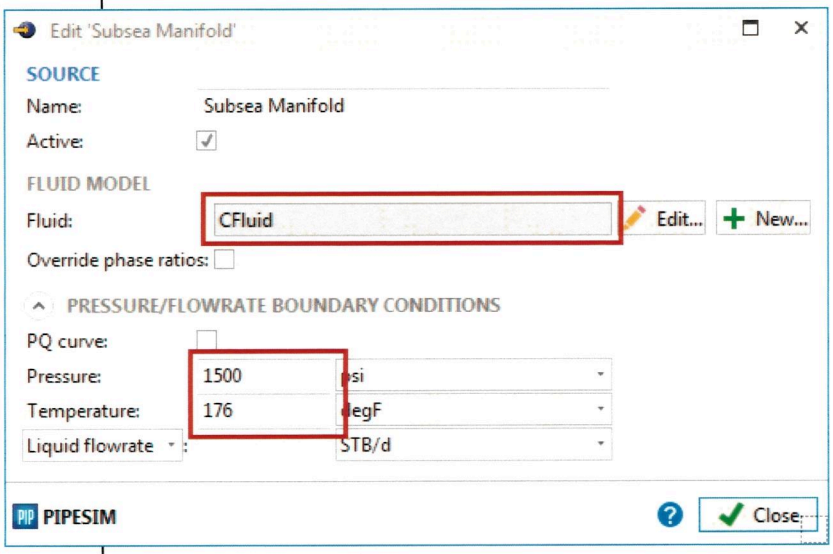


In the subsequent steps, you construct a **PIPESIM** model to replicate the network in the figure.



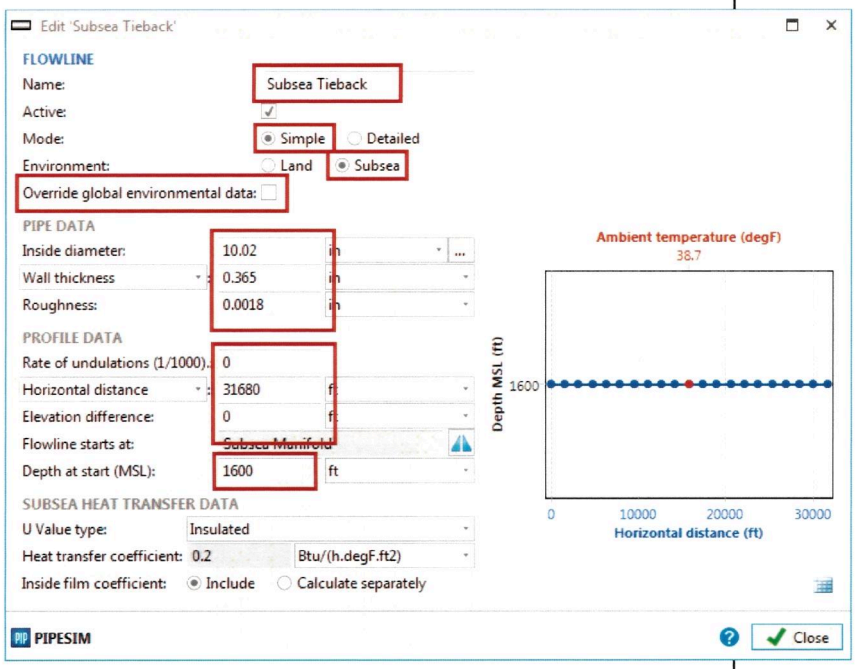
3. From the **Insert** tab, place a Source, Junction, and Sink on the Network schematic, as shown in the preceding figure.
4. Connect the source to the junction using a flowline from the **Insert** tab.
5. Connect the junction to the sink using a riser, also from the **Insert** tab.
6. Double-click the Source and rename it `Subsea Manifold`.
7. Map the compositional fluid you created, to the Subsea Manifold (Source), by selecting it from the *Fluid* drop-down menu.

8. Enter the data for the Subsea Manifold.



9. Without closing the dialog box for the Subsea Manifold, click the Flowline to switch the dialog box to the **Flowline editor**.

10. Rename the flowline `Subsea Tieback` and enter the flowline data.

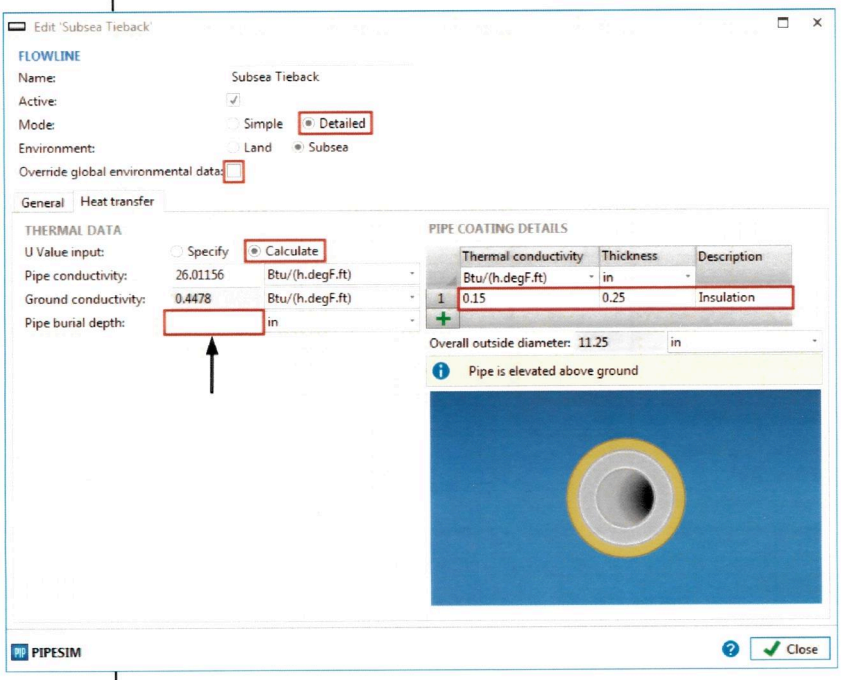


NOTE: Selecting the box *Override global environmental data* implies that you want to ignore the global environmental settings under **Home > Simulation settings > Environmental** and specify your own. To specify seawater temperature gradient and fluid velocity appropriate for the location, select the Detailed flowline mode and the *Calculate U Value input* option on the **Heat transfer** tab in the flowline editor.

11. Switch the Flowline Mode to **Detailed** so you enter more detailed heat transfer information.
12. Click the **Heat transfer** tab.
13. Set the *U Value input* option to Calculate.

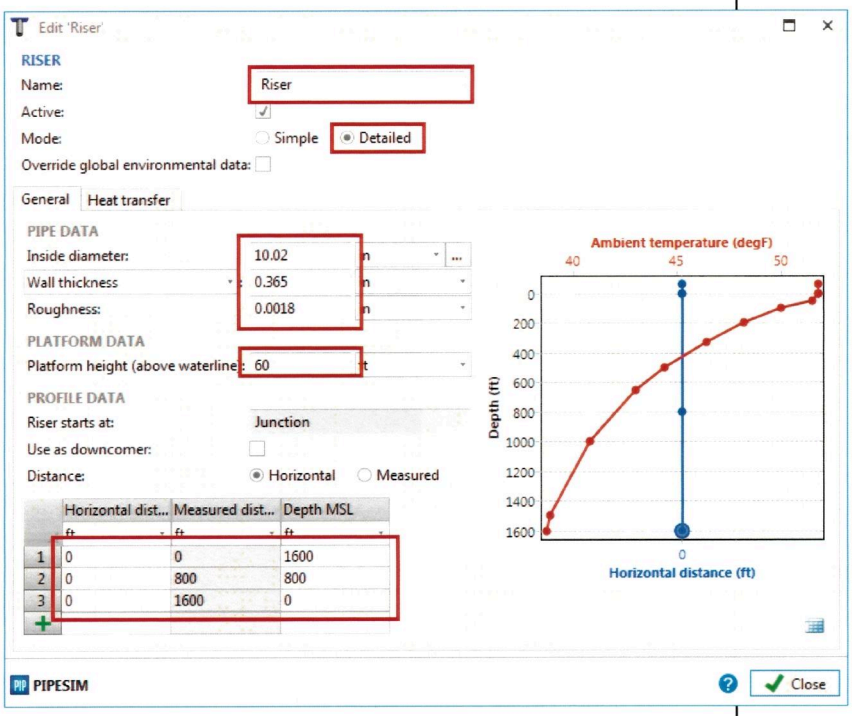
14. Enter 1 layer of insulation as in the figure.

    A blank value in the Pipe burial depth field indicates that the flowline is lying on the sea floor (not buried).



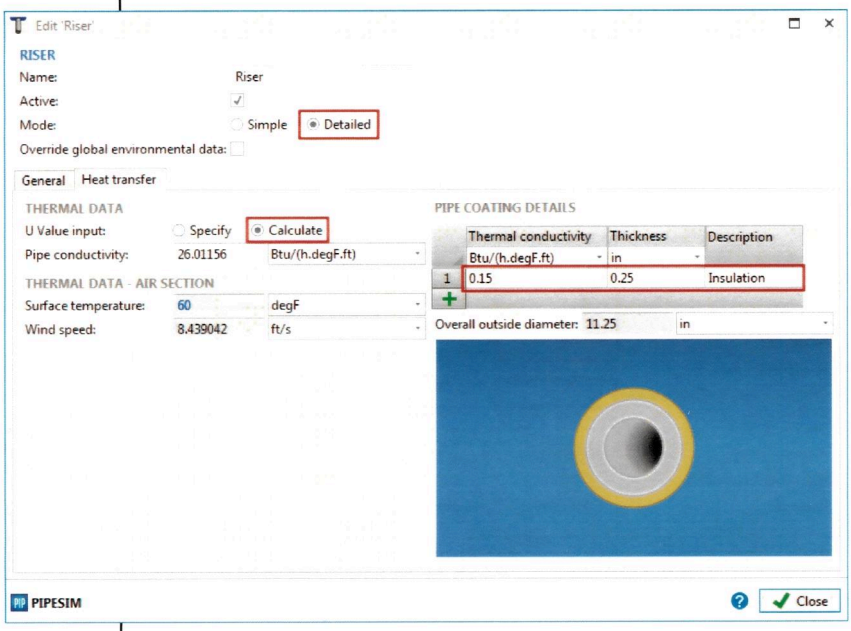
15. Without closing the **Flowline editor** dialog box for the Subsea Tieback, click the Riser to switch to the dialog box for editing it.
16. Rename the riser `Riser`.
17. Change the Mode to Detailed.

18. Enter the information on the **General** tab. The riser is vertical and produces up to a platform that is 60 ft above sea level.



19. Click the **Heat transfer** tab.
20. Set the *U Value input* option to Calculate.

21. Enter 1 layer of insulation.

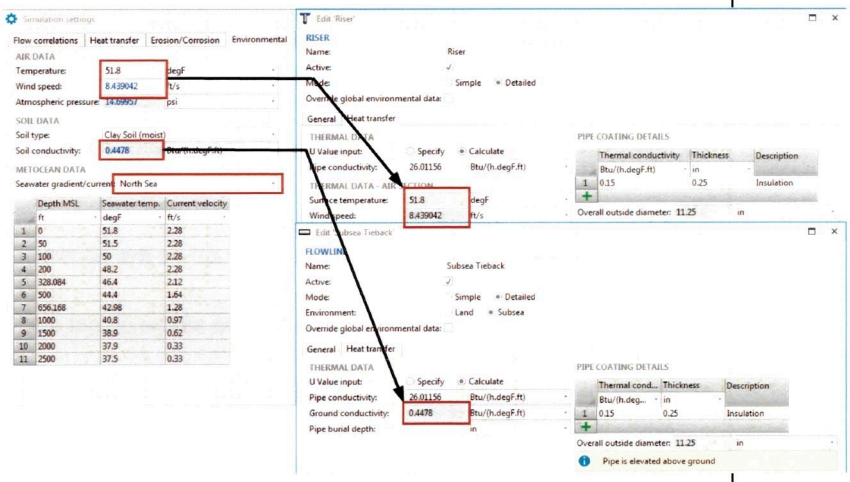


The ambient temperature and wind speed are read-only because you have chosen to use the global environment data under **Home > Simulation settings > Environmental** by clearing the *Override global environment data* box.

In a subsequent step, you review the default global environment settings and change them.

22. Exit the dialog box and save your workspace.

23. Modify the global environmental settings.
    a. Go to **Home > Simulation settings > Environmental.**
    b. Change the ambient air temperature and sea water gradient, as shown in the figure on the left. These global values are used for the heat transfer calculations for the flowline and riser because you cleared the *Override global environmental data* box in both the flowline and riser editors.



24. While still in the **Simulation settings** tab, click the **Flow correlations** tab and select the following flow correlations:
    - Vertical flow correlation = Hagedorn & Brown (Duns & Ros map)
    - Horizontal flow correlation = Beggs & Brill Revised
25. Design the system based on the following criteria and constraints:
    - Design production rate = 14,000 STB/d (normal scenario). The system achieves a maximum rate of 16,000 STB/d (if the wells produce more than expected). It also can handle a turn-down scenario when the production is expected to drop to 8,000 STB/d.
    - For all production rates, the arrival pressure at the Sink must not drop below 400 psia.

- Available flowline and riser sizes:
  - 7.981 inches (wall thickness = 0.322 inches)
  - 10.02 inches (wall thickness = 0.365 inches)
  - 12 inches (wall thickness = 0.375 inches).

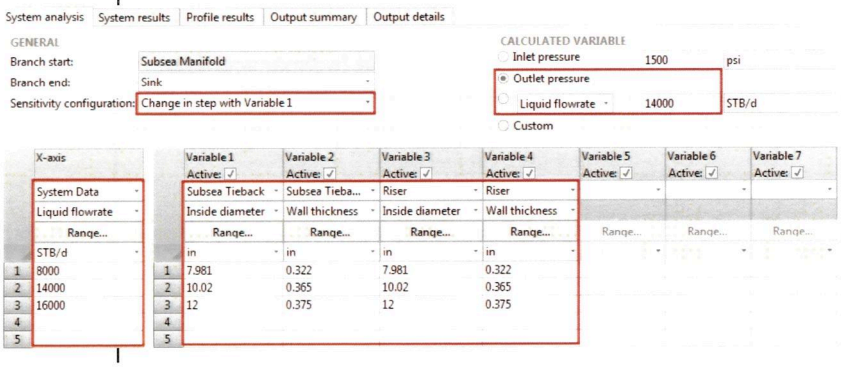
  Flowline and riser sizes must be the same and for all scenarios and the erosional velocity limit must not be exceeded.
- Cost: The bigger the flowline and riser, the higher the cost. The objective is to select the minimum diameter sizes that satisfy the target rate and constraints previously specified.

26. Select the Subsea Manifold and launch the System analysis task.
27. Select Outlet pressure as the Calculated Variable.
28. Enter any value for the Liquid flow rate, such as the normal rate of 14,000 STB/d.
29. For the X-axis variable, select System Data as the object and Liquid flow rate as the variable.
30. Enter the range of flow rates for the design: `8000`, `14000`, and `16000` STB/d.
31. Change the Sensitivity configuration to Change in step with Variable 1 by selecting it from the drop-down menu.



TIP: Review the topic, *System Analysis Properties*, to understand the differences between the sensitivity configuration options.

32. Configure the additional sensitivity variables for the Subsea Tieback and Riser, as in the figure.



33. Click **Run**.
34. Review the System results plot.

    Why are there no results for some cases?

35. Determine the minimum diameter of the tieback and riser that satisfies the arrival pressure requirement (>400 psia) for all flow rates. Record your results.

Results	
Minimum Tieback and Riser diameters that satisfy the minimum arrival pressure constraint of 400 psi for all flow rates (inches)	

36. Double-click the Systems plot and change the Y-axis variable to display *Erosional velocity ratio maximum*.
37. Verify that the selected flowline ID does not exceed the erosional velocity ratio limit of 1.0 for the expected flow rates.
38. Record your answers in the following table.

Results	
Property	**Value**
Selected Tieback and Riser ID based on 400 psi arrival pressure constraint (inches)	
Max. erosional velocity ratio for selected ID	
Minimum arrival pressure for selected ID (psi)	
Maximum arrival pressure for selected ID (psi)	

Lesson 2 Hydrates

Gas hydrates are crystalline compounds with a snow-like consistency that occur when small gas molecules come into contact with water at or below a specified temperature. The hydrate formation temperature increases with increasing pressure, which means that the hydrate risk is greatest at higher pressures and lower temperatures.

When hydrates form inside pipelines, they can form plugs that obstruct flow. In even worse scenarios, where the presence of hydrate plugs have gone undetected, pipeline depressurization has resulted in the plugs being dislodged unexpectedly, resulting in serious injury and even fatalities.

Hydrate forming molecules most commonly include methane, ethane, propane, carbon dioxide, and hydrogen sulfide.

Three hydrate crystal structures have been identified: Structures I, II, and H. The properties of Structures I and II hydrates are well-defined. Research into the structure of H hydrates is relatively new, and their properties are less well defined.

Hydrates can easily form downstream of a choke where fluid temperature can drop into the hydrate formation region because of Joule-Thompson cooling effects.

Figure 22 shows a typical gas hydrate curve that is useful for subsea pipeline design and operations. Above the curve is the hydrate formation region; a fluid containing water and gas in this pressure and temperature region forms hydrates.

Many factors can impact the hydrate curve, including fluid composition, water salinity, and the presence of hydrate inhibitors.

NOTE: Generating Hydrate curves requires the **PIPESIM Multiflash Hydrate** package and cannot be done with the **E300** or **GERG** flash packages.

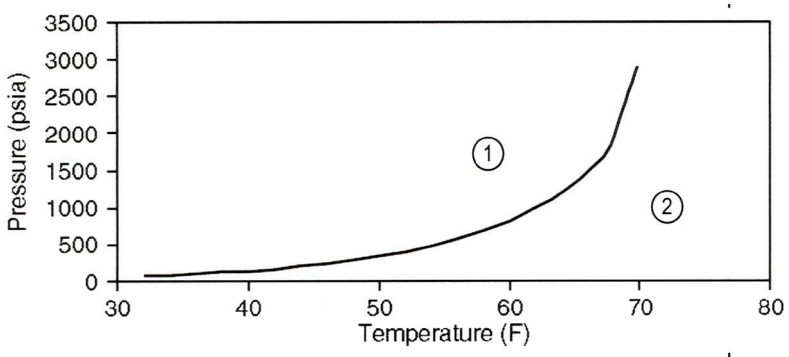


Figure 22 Hydrate curve

1 Hydrate region

2 Non-hydrate region

Hydrate mitigation strategies in PIPESIM

There are two common strategies available to mitigate hydrates formation: thermal insulation and chemical inhibitors. Both strategies can be simulated in **PIPESIM**. Thermal insulation carries a higher up-front capital cost, whereas chemical inhibition carries a higher operational cost.

Thermal insulation

The heat transfer between fluid in the pipeline and the environment surrounding the pipeline depends on the temperature gradient and the thermal conductivity of the material between the two. There are two options for modeling the heat transfer in **PIPESIM**: *Input U value* and *Calculate U value*.

Input U value allows you to define an overall heat transfer coefficient (U value). The heat transfer rate per unit area is calculated based on the pipe outside diameter.

Calculate U value computes the overall heat transfer coefficient based on these parameters:

- Pipe coatings
  - Thickness of the pipe coating
  - K (thermal conductivity) of the material

- Pipe material conductivity
- Ambient fluid (air or water)
- Ambient fluid velocity (The faster the fluid flows over the pipe, the greater the heat loss.)
- Pipe burial depth
- Ground conductivity (for flow lines only).

Chemical inhibitors

Thermodynamic inhibitors can be used to shift the hydrate line (to the left in the curve in Figure 22), which lowers the hydrate formation temperature and increases the hydrate-free operating envelope. Examples of inhibitors include methanol and ethylene glycol. The effects of this type of inhibitor can be modeled with **PIPESIM**.

Kinetic and anti-agglomerate inhibitors comprise a category known as Low Dosage Hydrate Inhibitors (LDHIs). These inhibitors do not lower the hydrate formation temperature; instead, they help prevent the nucleation and agglomeration of hydrates to avoid blockage formation. The effects of these types of inhibitors cannot be modeled with **PIPESIM**.



Exercise 1 Select tieback insulation thickness

In this exercise, you update the model with the tieback and riser ID you selected in the previous exercise, and determine the insulation thickness required to maintain the system outside of the hydrate formation envelope.

The worst case scenario in terms of hydrate risk is when the flow rate through the system is the lowest. In this case, the heat loss is the greatest. This scenario is the turndown case of 8,000 STB/d.

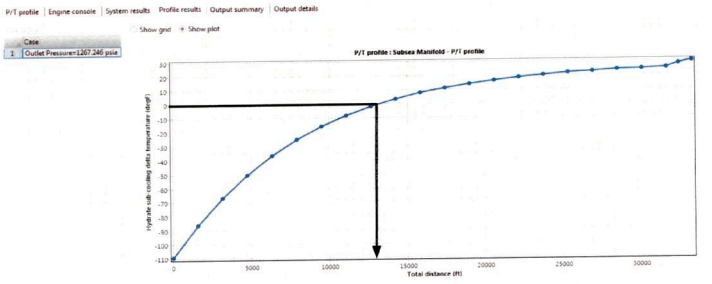
1. Double-click the Subsea tieback and enter the inside diameter and wall thickness determined from the previous sizing task.
2. Repeat Step 1 for the Riser (ID and Wall thickness for Tieback and Riser = 10.02 in and 0.365 in, respectively).
3. Navigate to **Home > Simulation settings > Heat transfer**.
4. Check the *Hydrate subcooling* box. This option enables the calculation of the hydrate sub-cooling temperature difference,

which is the difference between the hydrate formation temperature and the flowing fluid temperature ($T_{hyd} - T_f$).

If this difference is positive, the fluid is in the hydrate formation region at that location in the system.

5. Still under *Simulation settings*, click the **Output variables** tab.
6. Change the Report template to Flow Assurance. This template includes the important flow assurance variables; specifically the following hydrate variables of interest.
   - Hydrate formation temperature (profile variable)
   - Hydrate sub-cooling delta temperature (profile variable)
   - Maximum hydrate sub-cooling temperature differential (system variable)
7. Save your workspace.
8. Click the Subsea Manifold.
9. Set Outlet pressure as the Calculated variable and set the Liquid flow rate set to the turndown rate of `8,000` STB/d.
10. Run the P/T profile task.
11. Double-click the Profile results plot and change the Y-axis variable to *Hydrate sub-cooling delta temperature*.
    - With the current insulation thickness of 0.25 in, is there a hydrate risk?
    - At what point in the system does the fluid temperature drop below the hydrate formation temperature?

NOTE: The arrow points to where the system drops into the hydrate formation region at a depth of 12,627 feet.

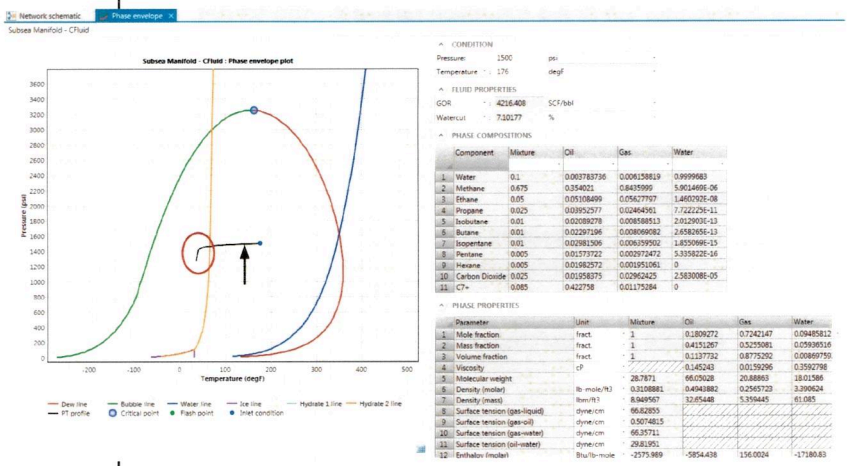


12. (Optional) Evaluate the hydrate risk using from the phase envelope viewer.
    a. Click the Subsea manifold.
    b. On the **Home** tab, click Phase envelope. The plot appears.
    c. Observe how the calculated flowing P/T profile line intersects with the hydrate formation line.

    As determined in the previous step, the system drops into the hydrate formation region (red circle in the figure) at approximately 12,672 ft, which is somewhere in the long subsea tieback; the tieback has a total length of 31,680 ft.

    Clearly, the current insulation thickness of 0.25 inch is insufficient to mitigate against hydrate formation. (Refer to the figure that follows.)

NOTE: The arrow points to the flowing P-T profile that is simulated by **PIPESIM**.



13. Determine the appropriate insulation thickness.
    a. Increase the thickness in 0.25 in increments.
    b. Run the P/T profile task until the entire system is hydrate-free.

NOTE: You must honor the constraint that the same insulation thickness be used on both the tieback and the riser.

c. Record your results in the table.

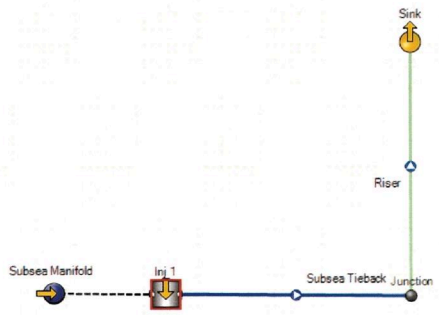
Results	
Property	**Value**
Req. Insulation thickness (inches)	

NOTE: Currently, there is no option to sensitize on coating thickness in any of the tasks (in this case, insulation thickness). You must manually change the insulation thickness in the tieback and riser objects and run the P/T profile task repeatedly until the objective is met.

Exercise 2 Determine the methanol requirement

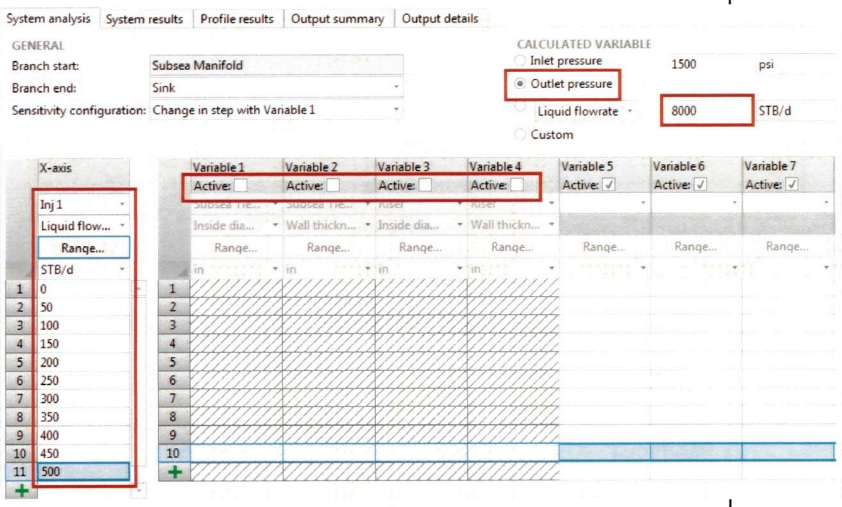
Assume the flowline and riser have been insulated, but they are under-insulated with only 0.50 inches of insulation. In this exercise, you determine the required methanol injection rate to ensure that hydrates do not form in the system.

1. Double-click the Subsea tieback and enter an insulation thickness of 0.5 in to model the scenario where it is under-insulated.
2. Click the Riser and repeat the previous step.
3. On the **Insert** tab, select an Injection Point and place it between the Subsea Manifold and the Subsea Tieback.
4. Connect the Subsea Manifold to the Injector with a connector.
5. Connect the left end of the Subsea Tieback to the right end of the Injector.



6. Launch the **Fluid manager** from the **Home** tab.
7. Click the **Component/model settings** tab.
8. Add Methanol to the *Fluid Components* list by selecting the box beside it.
9. Click the **Fluids** tab.
10. To create a new fluid, click +.
11. Double-click the row for the new fluid and rename it `Methanol`.
12. Enter `100` moles for Methanol in the Components grid and click **Close**.
13. Still on the **Fluid manager**, click the **Fluid mapping** tab.
14. Map the Methanol fluid to the Injector (Inj1) by selecting it from the drop-down list.
15. Exit the **Fluid manager**.
16. Double-click the Injector and specify a fluid injector temperature of `68` degF and any liquid flow rate. (Later, you sensitize on this variable.)
17. Click the Subsea manifold and launch the System Analysis task.
18. Set Outlet Pressure as the Calculated Variable and specify a Liquid flow rate of 8,000 STB/d.
19. For the X-axis variable, change the sensitivity object to the Injector (Inj1) and the sensitivity variable to Liquid flow rate by selecting them from the drop-down lists.

20. Enter a methanol injection rate range from 0 to 500 STB/d in increments of 50 STB/d.
21. Remove all other sensitivity variables by deleting their values or clearing the *Active* boxes for them.

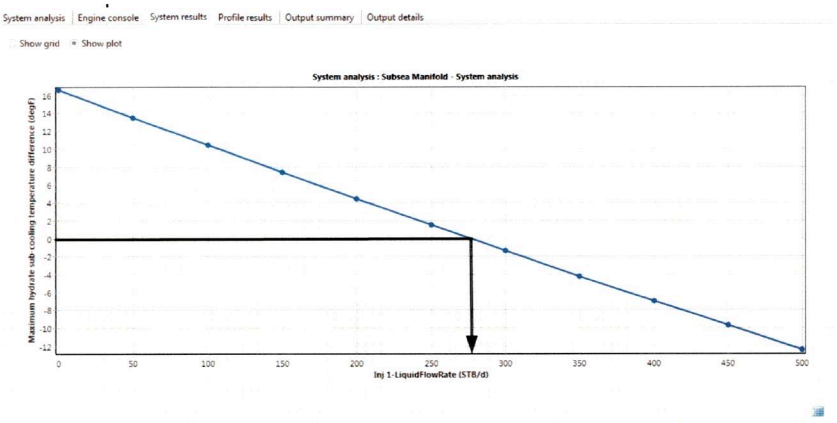


22. Click **Run**.
23. Double-click the System results plot and change the Y-axis variable to Maximum Hydrate sub-cooling temperature difference.

    This value is the maximum value of the Hydrate sub-cooling temperature difference throughout the system for each of the cases run.

24. From the plot, determine the required Methanol injection rate to maintain the flowing fluid temperature above the hydrate formation temperature, at every point in the system (such as Maximum Hydrate sub-cooling temperature difference < 0).

TIP: Refine the range of the methanol injection rate to more accurately determine the minimum amount required. (as shown in the figure that follows).



System analysis | Engine console | System results | Profile results | Output summary

Show grid ◉ Show plot

	Inj 1-LiquidFlowRate	Maximum hydrate sub-cooling temperature difference
	STB/d	degF
1	250	1.456166
2	255	1.166818
3	260	0.878606
4	265	0.5915146
5	270	0.3055663
6	275	0.02076162
7	280	-0.2779596
8	285	-0.5655088
9	290	-0.8263805
10	295	-1.117773
11	300	-1.409137

NOTE: A positive Maximum hydrate sub-cooling temperature difference indicates that the fluid temperature is below the hydrate stability temperature and the system precipitates hydrates at that pressure and temperature condition.

Results	
Property	**Value**
Req. Methanol Injection Volume (STB/d)	

Lesson 3 Severe riser slugging

Severe slugging in risers can occur in a multiphase transport system consisting of a long flowline followed by a riser. Severe slugging is a transient phenomenon that can be split into four steps (Figure 23).

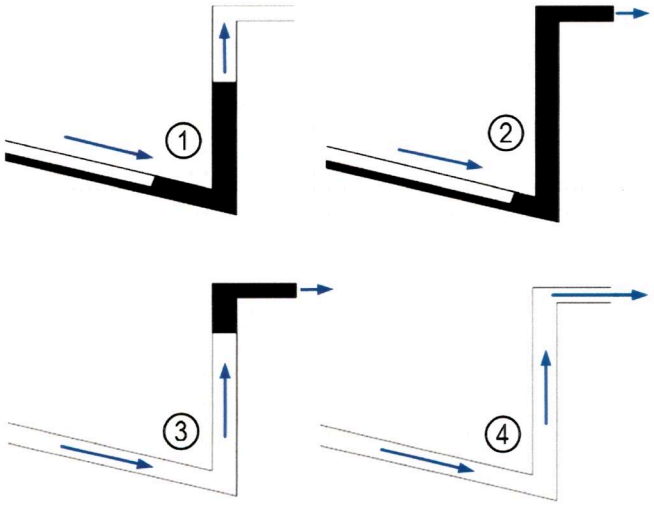


Figure 23 Four slugging steps

1. In this step, gas velocity is insufficient to carry liquid droplets up the riser. Droplets start to accumulate at the base of the riser, then block the riser and cause increased back-pressure on the pipeline.
2. In slug production, the liquid level reaches the riser outlet, and the liquid slug begins to be produced until gas reaches the riser base.
3. In bubble penetration, gas is again supplied to the riser, so the hydrostatic pressure decreases. As a result, the gas flow rate increases.
4. This step corresponds to gas blowdown. When the gas produced at the riser bottom reaches the top, the pressure is minimal, and the liquid no longer is gas-lifted. The liquid level falls, and a new cycle begins.

PIPESIM does not rigorously model severe slugging associated with risers as this is a transient phenomenon, but it does report a dimensionless indicator of the likelihood of slugging occurring (the Severe Slugging Indicator - Pots).

Severe slugging is most prevalent in cases in which a long flowline precedes a riser, especially for cases in which the flowline inclination angle is negative going into the riser (as in Figure 23).

In cases of severe slugging, a slug catcher must be sized to be able to receive a volume of liquid at least equal to the volume of the riser. However, severe slugging can be mitigated by measures such as topsides choking or riser base gas lift.

The *Severe Slugging Indicator* is the ratio between the pressure build-up rates of the gas phase and that of the liquid phase in a flowline followed by a vertical riser:

$$PI\text{-}SS = \frac{ZRT/_{MW_G}}{gL_F\overline{\alpha}_{GF}W_L}$$

where:

Z = Gas compressibility factor

R = Gas universal constant

T = Temperature (K)

M = Molecular weight of gas

W_G = Gas mass flow rate (kg/s)

W_L = Liquid mass flow rate (kg/s)

g = Acceleration because of gravity (m/s^2)

L_F = Flowline length (m)

$\overline{\alpha}_{GF}$ = Average flowline gas holdup

Severe slugging is expected when the Severe Slugging Indicator number is equal to, or less than, 1. This model can be used to determine the onset of severe slugging, but it cannot predict the length of severe slugs, where they are located, or how fast they will are produced into the separator. Refer to the **PIPESIM** Help topic *Liquid by Sphere* for details).

Exercise 1 Screen for severe riser slugging



Continue using the model from the previous exercise.

1. To deactivate the methanol injector, right-click it and click **Deactivate** on the menu.
2. Reset the tieback/riser insulation thickness to the value obtained earlier that prevents hydrate formation in the system (1").
3. Click the Subsea Manifold.
4. Launch the **System analysis** task from the **Home** tab.
5. Set Outlet Pressure as the **Calculated Variable** and leave the default Inlet pressure and Liquid flow rate values at their current settings.
6. For the X-axis, select System data as the sensitivity object and Liquid flow rate as the sensitivity variable.
7. Enter the following flow rates: 8000, 14000, and 16000 STB/d.
8. Click **Run**.
9. Double-click the System results plot and change the Y-axis variable to display the variable *Severe Slugging Indicator - Pots*. This represents the minimum value of the Severe Slugging Indicator number along the subsea tieback at the base of the riser.
10. Record your answers in the table.

Results			
Severe slugging	**8,000 STB/d**	**14,000 STB/d**	**16,000 STB/d**
Severe slugging indicator			



Lesson 4 Slug catcher sizing

PIPESIM is frequently used to estimate the capacity requirements for slug catchers. A *slug catcher* is a pressure vessel with sufficient volume to buffer the downstream process system from slugs of liquid coming from the upstream system.

For offshore platforms, the designer must balance the high cost of adding a larger vessel to the platform against the potential of a large slug overwhelming the liquids handling capacity and shutting down the entire system.

There are three typical scenarios to consider in the sizing of slug catchers for this type of system:

- Hydrodynamic slugging
- Pigging
- Ramp-up

Hydrodynamic slugging

Hydrodynamic slugging is caused by the instability of waves on the gas-liquid interface in stratified flow under certain flowing conditions. When these slugs are numerous, the pipeline is said to be operating in the slug flow regime. Most multiphase production systems experience hydrodynamic slugging.

Usually, it is impractical to design a pipeline system that completely avoids slugging problems. Also, because hydrodynamic slugs grow as they progress along the pipe, long pipelines can produce large hydrodynamic slugs. Severe riser slugging, which was modeled in the previous exercise, is a special case of hydrodynamic slugging involving a riser.

PIPESIM calculates the mean slug length as a function of distance traveled by using the SSB or Norris Correlations. A continuous intermittent flow regime in the pipeline is required for slugs to form. A probabilistic model (based on experimental data from Prudhoe Bay field data in Alaska) is applied to calculate the largest slug out of 10, 100, and 1,000 occurrences. The 1/1000 (one in one thousand) slug length is often considered as one of the criteria to determine slug catcher volume requirement.

The slug prediction output from **PIPESIM** yields the length and frequency for the selected slug size correlation:

- Mean slug length (distribution is assumed skewed log normal)
- 1 in 1,000 slug length and frequency
- 1 in 100 slug length and frequency
- 1 in 10 slug length and frequency

The preceding probabilities represent various levels of confidence regarding the maximum slug size. For example, a 1 in 1,000 slug length of 50 meters indicates there is a 0.1% probability of the maximum slug length exceeding 50 meters.

Pigging

In multiphase flow in horizontal and upwardly inclined pipe, the gas usually travels faster than the liquid because of lower density and lower viscosity. This is referred to as *slippage*. Multiphase flow correlations predict the slip-ratio, which depends on many factors, such as fluid properties, pipe diameter, and flow regime. The definition of *liquid holdup* was discussed earlier in the training.

In steady-state flow, the gas usually travels faster than the liquid when the flow direction is upwards, so it slips past the liquid. For the volume flow rate to remain constant, the area of the pipe occupied by the gas must shrink. This gives rise to a higher liquid volume fraction than if the gas traveled at the same velocity, resulting in *liquid holdup*, as illustrated in Figure 24.

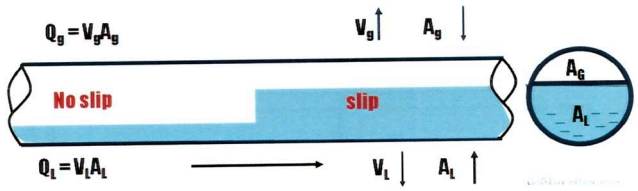


Figure 24 Liquid holdup

During a pigging operation, a solid object with an outside diameter slightly less than the inside diameter of the pipeline is sent through the line to push out liquids and debris. As a pipeline is pigged

(Figure 25), a volume of liquid builds up ahead of the pig and is expelled into the slug catcher as the pig approaches the exit.

PIPESIM considers that the pig travels at the mean *fluid* velocity. It uses the liquid holdup in the pipeline to calculate the volume of liquid that is swept along in front of the pig as it moves. This calculation is reported as the sphere generated liquid volume (SGLV).

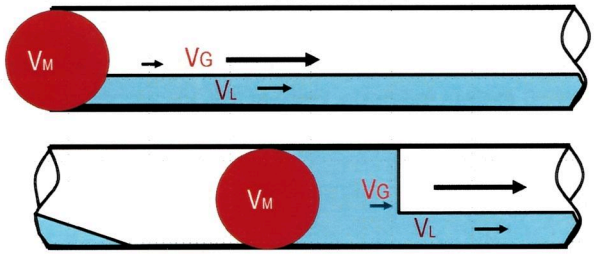


Figure 25 Pigging operation

Ramp-up

When the flow rate into a pipeline increases, the overall liquid holdup typically decreases because the gas can more efficiently sweep out the liquid phase. When a rate increase (ramp-up) occurs, the liquid volume in the pipeline is accelerated resulting in a surge.

A ramp-up operation is illustrated in Figure 26. **PIPESIM** predicts the liquid surge rate using Cunliffe's Method. For more details, see the Cunliffe's method reference in the **PIPESIM** help system.

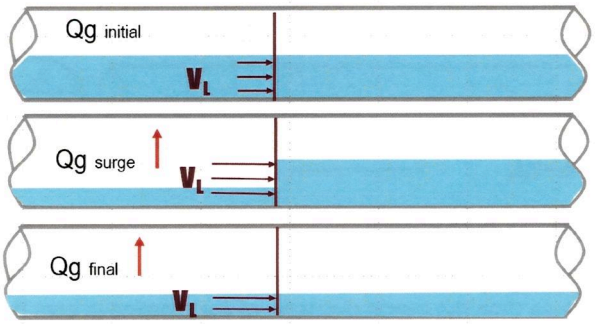


Figure 26 Ramp-up operation

Evaluation of each scenario

For a more detailed analysis of slug catcher sizing, you also should consider the drainage rates of the primary separator and slug catcher. Hydrodynamic slugs and pig-generated slugs typically occur over a short duration (minutes), while the surge created by a ramp-up operation can occur over a long duration (hours/days). Detailed slugging calculations are better performed in a transient multiphase simulator, such as **OLGA**.

Exercise 1 Size a slug catcher



In this exercise, you screen for severe slugging and determine the required size of the slug catcher based on the largest volume of the following criteria, multiplied by a safety factor of 1.2.

Consider these criteria:

- Hydrodynamic slugging, which typically generates the largest slugs of all the scenarios. The design is based on the statistical 1/1000 population slug size, as determined by using the SSB or Norris Correlations.
- The requirement to handle liquid swept in front of a pig.
- Transient effects, such as the requirement to handle the liquid slug generated when the production flow is ramped up from 8,000 to 16,000 STB/d, such as ramp-up surge.



NOTE: The slug-catcher is sized to address only the scenarios described here, and it does not address severe riser slugging.It is assumed that severe riser slugging can be mitigated with topsides choking or riser-based gas lift.

Continue using the model from the previous exercise.

1. From the **Home** tab, go to **Simulation settings > Output variables**. The **Flow Assurance** report template is selected.
2. To make a copy so you can customize it, click the **Clone** button.
3. Enter a name for the cloned report.
4. To see all the selected profile variables., click the **Profile** tab. To see the complete list of profile variables, click the **Selected** tab to deactivate it.

5. Add the following profile variables of interest that are not included in the Flow Assurance template, by selecting the box next to each of them.



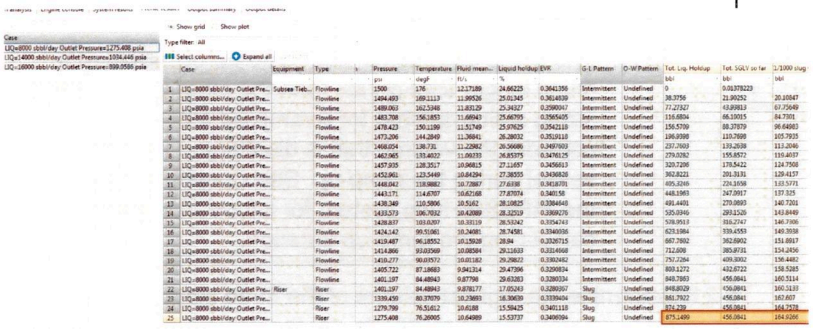
TIP: To filter the list, enter the first few letters of each variable in the *Type to filter* box.

   - Sphere-generated liquid volume from section (profile variable)
   - Total sphere-generated liquid volume so far (profile variable)
   - Cumulative liquid holdup (profile variable)

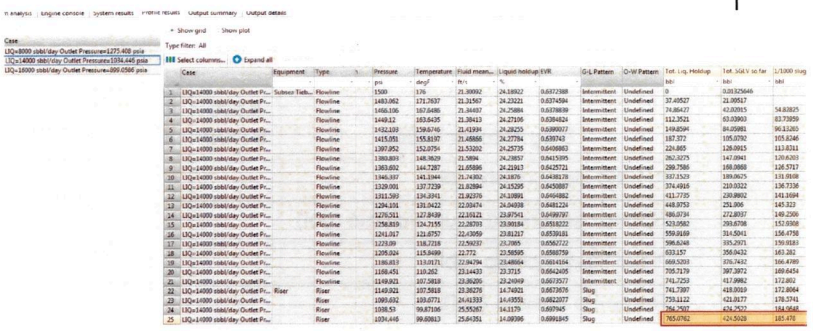
6. Click the **System** tab and add the following system variables, if missing.
   - Sphere generated liquid volume (system variable)
   - 1 in 1000 slug volume (profile and system variables)
7. Exit the dialog box and save your workspace.
8. Launch the P/T profile task and set it up with Outlet pressure as the Calculated variable.
9. Leave the default values for the boundary conditions.
10. Leave the default values for the boundary conditions.
11. Select System Data as the sensitivity object and Liquid flow rate as the sensitivity variable and enter the three liquid flow rates that have been used in the previous exercises (`8000`, `14000`, and `16000` STB/d).
12. Click **Run**.
13. On the **Profile results** tab, select *Show grid*, then click **Select Columns** and add the following three variables to the grid.
    - Total/Cumulative liquid-holdup
    - 1 in 1000 slug volume
    - Total SGLV so far
14. Click **Close** to exit the **Select columns** dialog box.
15. Scroll to the end of the Profile results grid to see these variables.
16. Change the units for all three variables to bbl.

17. For each sensitivity rate, extract the maximum 1/1000 Slug Volume and the Total Sphere Generated Liquid Volume (SGLV) at the system outlet and record them.
18. Compare the Slug Volume and SGLV with the answers shown in the figures.

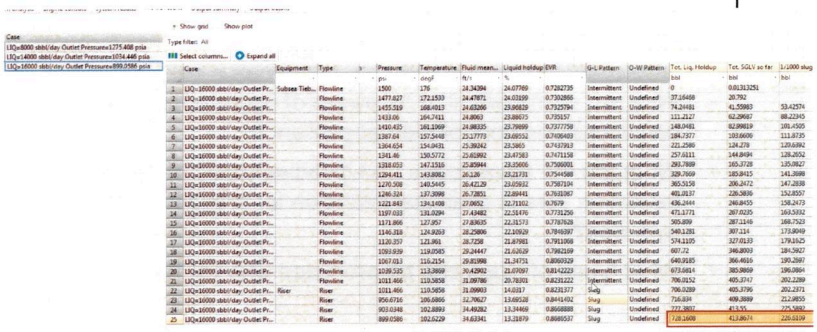
8,000 STB/d



14,000 STB/d



16,000 STB/d



19. Assuming the design is based on a ramp-up volume from 8,000 to 16,000 STB/d, the ramp-up volume is the difference in total liquid holdup in the system between the 8,000 STB/d and 16,000 STB/d cases.

    Extract the total liquid holdup at 16,000 STB/d and at 8,000 STB/d, and subtract the two values.

20. Compare your answer with the table that follows.



NOTE: The surge associated with ramp-up occurs over a much longer time period than the other cases. When sizing a slug catcher for a ramp-up scenario, consider the volume that is discharged from the vessel during the ramp-up. See Cunliffe's Method in the **PIPESIM** help system for information about calculating the ramp-up duration.

21. Determine the highest volume of the three scenarios and apply a safety factor of 20% to get the design volume for the slug catcher.
22. Compare it with the answers in the table.

Results			
Slug catcher sizing	**8,000 STB/d**	**14,000 STB/d**	**16,000 STB/d**
1/1000 slug volume (bbl)			
Sphere generated liquid volume (bbl)			
Total liquid holdup (bbl)			
Ramp-up volume (bbl)			
Design volume for slug catcher (bbl)			



Review questions

- Why is it so important to be aware of hydrate formation conditions and control their appearance?
- What are the advantages and disadvantages of thermal insulation versus chemical inhibition for prevention of hydrates?
- What types of slugs can be predicted with **PIPESIM**?

- Why should the SGLV not be greater than the cumulative liquid holdup?
- Can **PIPESIM** be used for transient analysis?

Summary

In this module, you learned about:

- sizing the subsea tieback line and riser
- determining the pipeline insulation requirements
- determining methanol injection to inhibit hydrate formation
- screening the results for severe slugging at the riser base
- sizing a slug catcher

NOTES

Module 7 Looped gas gathering network

It is important to be able to model networks as complete systems taking into account the interaction of the different components (such as compressors, separators, and wells) producing into common gathering systems. The wellhead pressure and, by extension, the deliverability of any particular well is influenced by the backpressure imposed by the production system.

Modeling the network as a whole allows the engineer to determine the effects of changes such as adding new wells, adding compression, looping flow lines, and changing the separator pressure.

In this module, you learn how to build a gathering network and perform a network simulation to evaluate the deliverability of the complete production system.

Learning objectives



After completing this module, you will know how to:

- build a network model
- specify the network boundary conditions
- solve the network and establish the deliverability

Lesson 1 Model a gathering network



Network models are constructed using the same procedures learned through the earlier lessons.

Boundary conditions

To solve a network model, enter the correct number of boundary conditions. Boundary nodes have only one connecting branch, such as a production well, injection well, source, or sink.

The number of boundary conditions required for a model is determined by the model's Degrees of Freedom:

> Degrees of Freedom = number of wells (production and injection) + number of sources + number of sinks

For example, a 3-well production system producing fluid to a single delivery point has 4 degrees of freedom (3+1), regardless of the network configuration between the well and the sink.

Each boundary can be specified in terms of:

- Pressure
- Flow rate
- Pressure/Flow rate (PQ) curve.

Additionally, these conditions must be satisfied:

- The number of pressure, flow rate, or PQ specifications must equal the Degrees of Freedom of the model.
- At least one pressure must be specified.
- At each source (production well and source) the fluid temperature must be set.

PIPESIM validates that the correct number and types of boundary conditions are set before the simulation run can be initiated.

Solution criteria

A network has converged when the pressure balance and mass balance at each node are in the specified tolerance. The calculated pressure at each branch entering and leaving a node is averaged, and the tolerance of each pressure is calculated from this equation:

If all $Ptol$ values are in the specified network tolerance, that node has passed the pressure convergence test. This process is repeated for each node. The total mass flow rate into and out of a node are averaged. The tolerance is calculated from this equation:

If the *Mtol* value is in the specified network tolerance, that node has passed the mass convergence test. This calculation is repeated for each node.

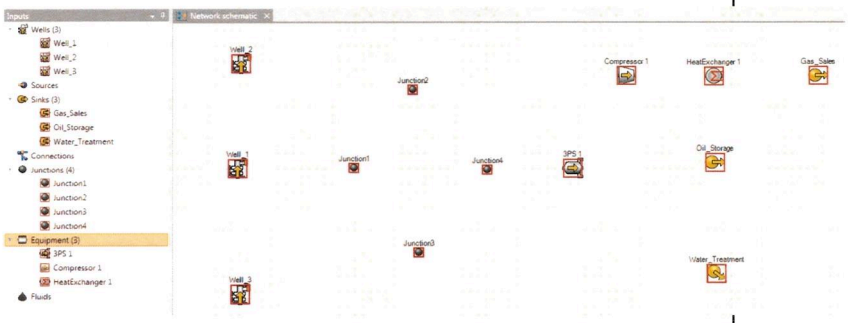
The network has converged when all of the foregoing conditions are satisfied.

Exercise 1 Model a pipeline network



In this case study, your goal is to establish the deliverability of a production network. The network consists of three producing gas wells in a looped gathering system that deliver the commingled stream to a single delivery point.

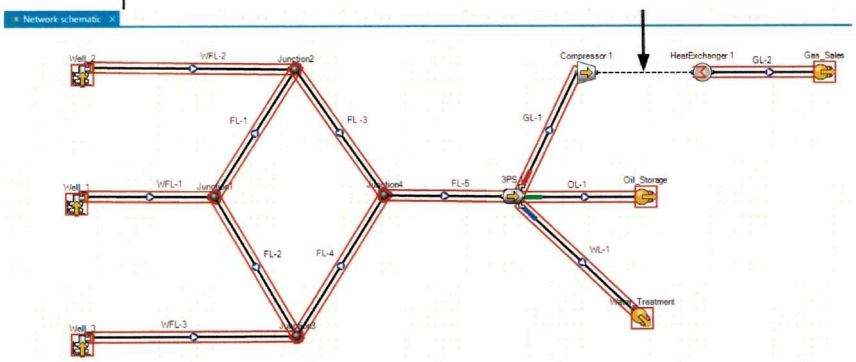
1. Launch **PIPESIM** and create a new, network-centric workspace.
2. From the **Insert** tab, add the following equipment:
   - Wells 3 (Use the *Simple Vertical* template for all wells.)
   - Junctions 4
   - 3-phase separator 1
   - Compressor 1
   - Heat exchanger 1
   - Sinks 3
3. Rename and arrange them exactly as shown in the figure.



4. Connect the inserted objects using flow lines and connectors so that the network diagram exactly matches the figure.

5. Rename all the flow lines exactly as indicated in the figure.

NOTE: A connector is shown between Compressor 1 and HeatExchanger 1.



NOTE: A *connector* is a line used to represent two separate objects that are connected, but they have zero length between them. There is no pressure drop between objects linked with a connector. For example, a connector can be used to connect a wellhead and a choke.

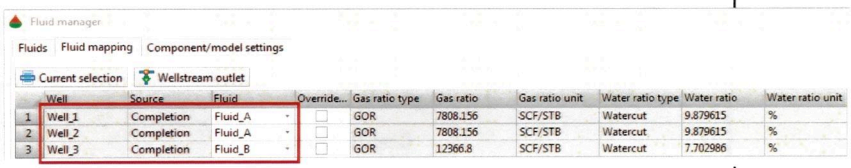
6. Create two new compositional fluids for the three wells based on the compositions listed in the table.
   a. Set the PVT package to Multiflash.
   b. Select the 3-parameter Peng-Robinson option as the Equation of State.
   c. Leave all other model settings to their default values, but name the fluids as listed.

NOTE: There are 11 components.

Component	Moles	
	Fluid_A (Well_1 & Well_2)	Fluid_B (Well_3)
Water	10.0	6.0
Methane	67.5	71.0
Ethane	5.0	6.0
Propane	2.5	3.0

Component	Moles	
	Fluid_A (Well_1 & Well_2)	Fluid_B (Well_3)
Isobutane	1.0	1.0
Butane	1.0	1.0
Isopentane	1.0	1.0
Pentane	0.5	0.5
Hexane	0.5	0.5
Carbon Dioxide	2.5	4.0
C7+	8.5	6.0
Name	**Molecular weight**	**Specific gravity**
C7+	115	0.683

7. After you create the fluids, map Well_1 and Well_2 to Fluid_A and Well_3 to Fluid_B in the **Fluid manager**.



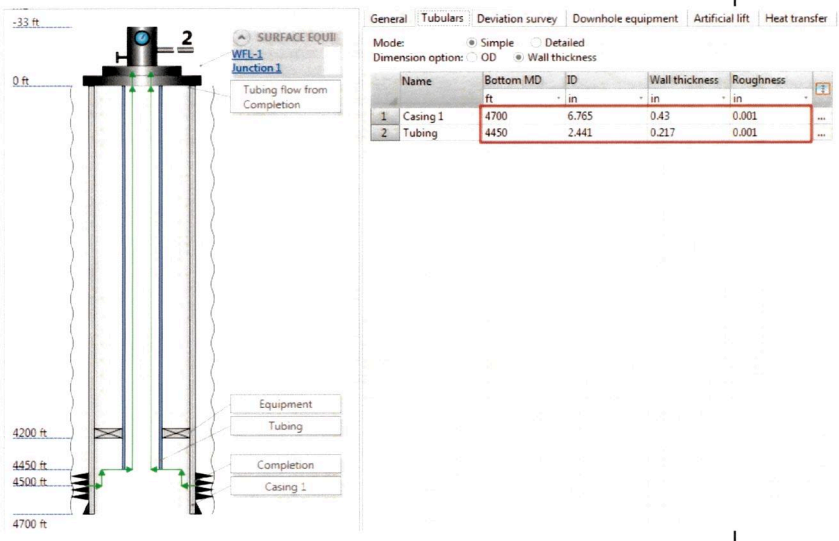
8. Save your workspace.
9. In a previous step, you created all three wells using the Simple Vertical template. Now, you edit the default template values for each well and change them to the values listed in the table.

Properties	Unit	Well_1	Well_2	Well_3
Well type	-	Vertical	Vertical	Vertical
Casing data				
Casing (Bottom MD)	ft	4700	4600	4800
Casing ID	inch	6.765	6.765	6.765
Casing wall thickness	inch	0.43	0.43	0.43

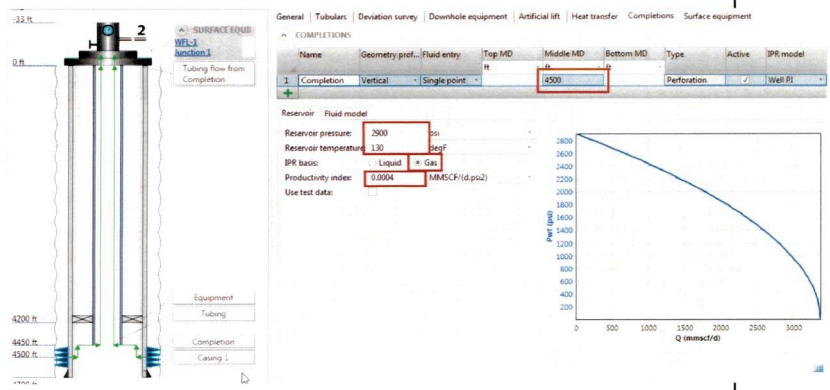
Properties	Unit	Well_1	Well_2	Well_3
Tubing data				
Tubing (Bottom MD)	ft	4,450	4,350	4,550
Tubing ID	inch	2.441	2.441	2.441
Tubing wall thickness	inch	0.217	0.217	0.217
Properties	**Unit**	**Well_1**	**Well_2**	**Well_3**
Roughness (casing & tubing)	inch	0.001	0.001	0.001
Packer depth (MD)	ft	4,200	4,100	4,300
Heat transfer data				
Wellhead ambient temperature	degF	60	60	60
Heat transfer coefficient	Btu/ (h.degF.ft^2)	2	2	2
Completion data				
Fluid entry	-	Single point	Single point	Single point
Completion depth (MD)	ft	4,500	4,400	4,600
IPR model		Well PI	Well PI	Well PI
Reservoir pressure	psia	2,900	2,900	3,100
Reservoir temperature	degF	130	130	140
Gas PI	mmscf/d/ psi^2	0.0004	0.0004	0.0005

The three wells should look like the figures that follow.

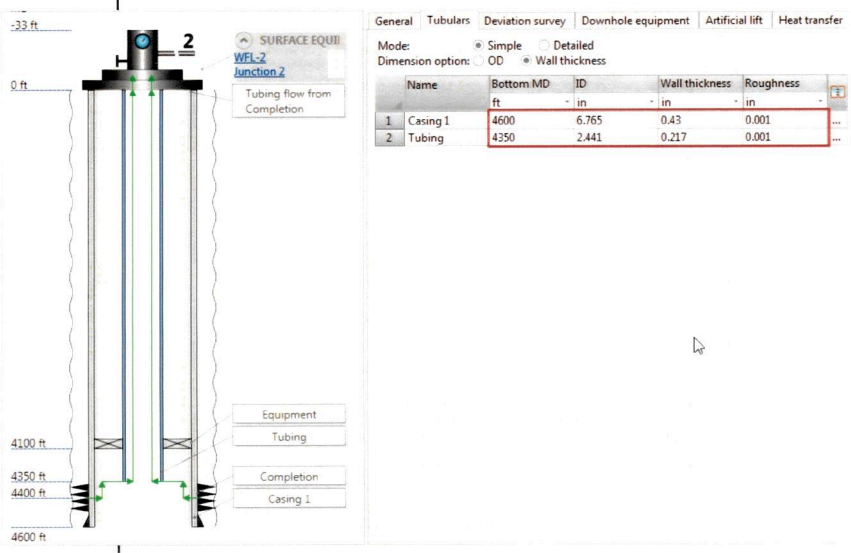
Well_1


-33 ft
0 ft
4200 ft
4450 ft
4500 ft
4700 ft
SURFACE EQUIP
WFL-1
Junction 1
Tubing flow from Completion
Equipment
Tubing
Completion
Casing 1
General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Mode: Simple Detailed
Dimension option: OD Wall thickness
Name | Bottom MD | ID | Wall thickness | Roughness
ft | in | in | in
1 Casing 1 | 4700 | 6.765 | 0.43 | 0.001
2 Tubing | 4450 | 2.441 | 0.217 | 0.001

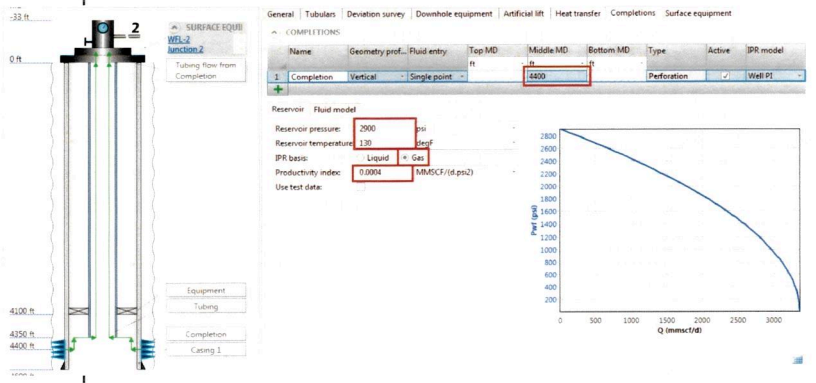
Well_1


General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Completions
Surface equipment
COMPLETIONS
Name | Geometry prof... | Fluid entry | Top MD | Middle MD | Bottom MD | Type | Active | IPR model
1 Completion | Vertical | Single point | 4500 | Perforation | Well PI
Reservoir
Fluid model
Reservoir pressure: 2900 psi
Reservoir temperature: 130 degF
IPR basis: Liquid Gas
Productivity index: 0.0004 MMSCF/(d.psi2)
Use test data:
Pwf (psi)
Q (mmscf/d)

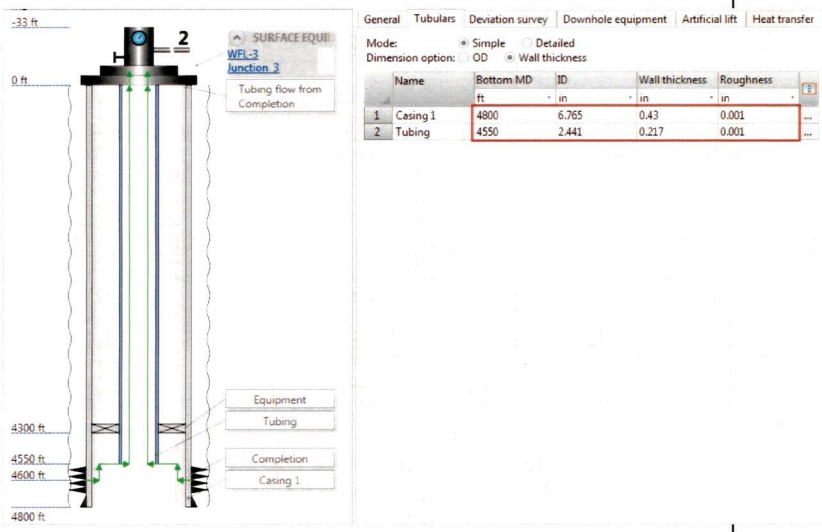
Well_2


-33 ft
0 ft
4100 ft
4350 ft
4400 ft
4600 ft
2
SURFACE EQUI
WFL-2
Junction 2
Tubing flow from Completion
Equipment
Tubing
Completion
Casing 1
General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Mode: Simple Detailed
Dimension option: OD Wall thickness
Name
Bottom MD
ID
Wall thickness
Roughness
ft
in
in
in
1 Casing 1 4600 6.765 0.43 0.001
2 Tubing 4350 2.441 0.217 0.001

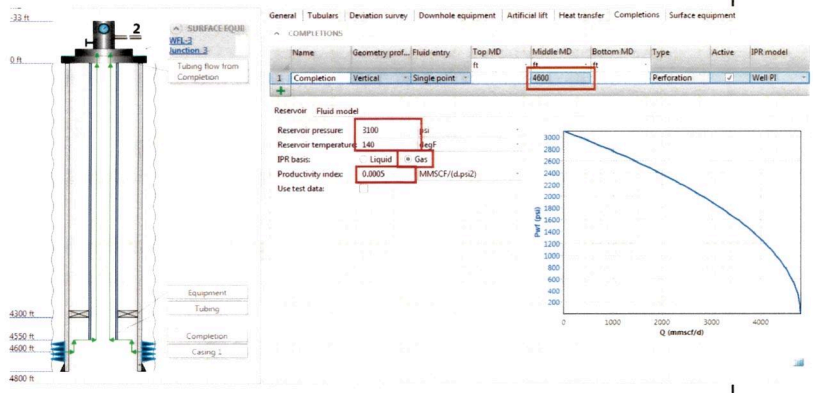
Well_2


-33 ft
0 ft
4100 ft
4350 ft
4400 ft
2
SURFACE EQUI
WFL-2
Junction 2
Tubing flow from Completion
Equipment
Tubing
Completion
Casing 1
General
Tubulars
Deviation survey
Downhole equipment
Artificial lift
Heat transfer
Completions
Surface equipment
COMPLETIONS
Name
Geometry prof...
Fluid entry
Top MD
Middle MD
Bottom MD
Type
Active
IPR model
1 Completion Vertical Single point 4400 Perforation Well PI
Reservoir
Fluid model
Reservoir pressure: 2900 psi
Reservoir temperature: 130 degF
IPR basis: Liquid Gas
Productivity index: 0.0004 MMSCF/(d.psi2)
Use test data:
Pwf (psi)
Q (mmscf/d)

Well_3



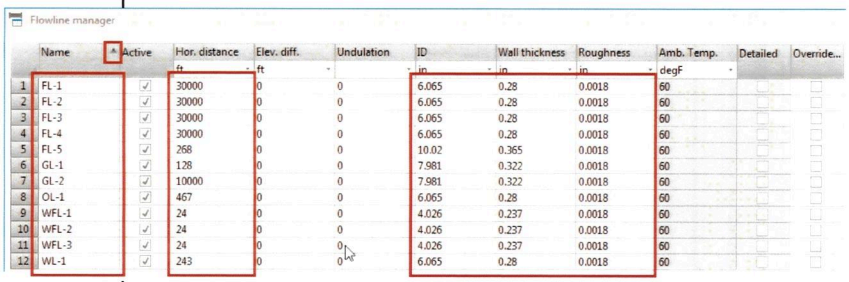
Well_3



10. Click the **Flowline manager** on the **Home** tab.

11. Specify the flowline details, as shown in the diagram.

TIP: Double-click the *Name* column to sort in alphabetical order to make it easier to enter the data).

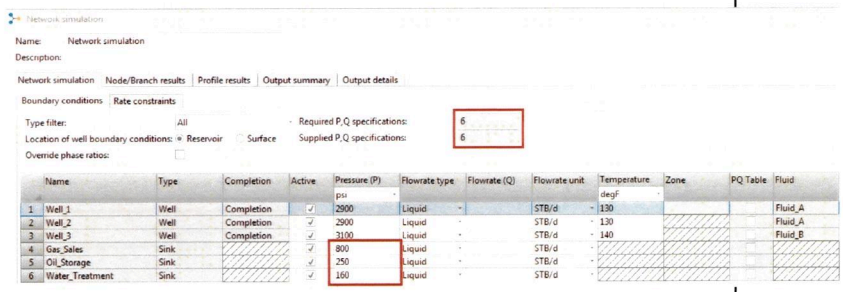

Flowline manager

	Name	Active	Hor. distance	Elev. diff.	Undulation	ID	Wall thickness	Roughness	Amb. Temp.	Detailed	Override...
			ft	ft		in	in	in	degF		
1	FL-1	✓	30000	0	0	6.065	0.28	0.0018	60		
2	FL-2	✓	30000	0	0	6.065	0.28	0.0018	60		
3	FL-3	✓	30000	0	0	6.065	0.28	0.0018	60		
4	FL-4	✓	30000	0	0	6.065	0.28	0.0018	60		
5	FL-5	✓	268	0	0	10.02	0.365	0.0018	60		
6	GL-1	✓	128	0	0	7.981	0.322	0.0018	60		
7	GL-2	✓	10000	0	0	7.981	0.322	0.0018	60		
8	OL-1	✓	467	0	0	6.065	0.28	0.0018	60		
9	WFL-1	✓	24	0	0	4.026	0.237	0.0018	60		
10	WFL-2	✓	24	0	0	4.026	0.237	0.0018	60		
11	WFL-3	✓	24	0	0	4.026	0.237	0.0018	60		
12	WL-1	✓	243	0	0	6.065	0.28	0.0018	60		

TIP: To populate/fill the same value down a column, such as the constant roughness value of 0.0018 inches shown in the figure, enter the value into the top row you want to fill down from, and press F3.

12. Exit the **Flowline manager**. All the flow lines are now valid (they are no longer red.)
13. Double-click the Separator (3PS) and select Gas as the Production stream.
14. Leave the default value of 100% for both the Gas/Oil and Water/Oil efficiencies.
15. Without exiting the **Separator** dialog box, click the Compressor and enter a Pressure differential of `400` psi and an Efficiency of `70`%.
16. Without exiting the **Compressor** dialog box, select the Heat Exchanger and enter a Pressure differential of `15` psi and Discharge temperature of `120` degF.
17. Exit the dialog box and save the workspace.
18. Navigate to *Home* > *Simulation settings* > *Flow correlations.*
19. Select Beggs & Brill Revised as the global vertical and horizontal multiphase flow correlations.
20. From the **Home** tab, launch the Network simulation task.

21. Configure the boundary conditions for the simulation task.

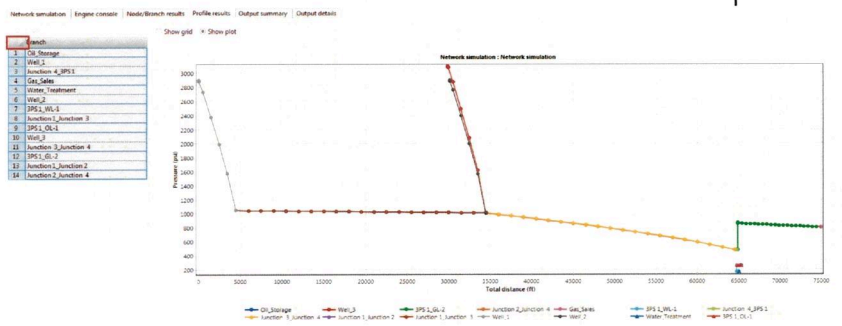


NOTE: The **Run** button is active only when the required number of P,Q specifications have been supplied and there are no validation issues.

22. Click **Run** to launch the simulation.

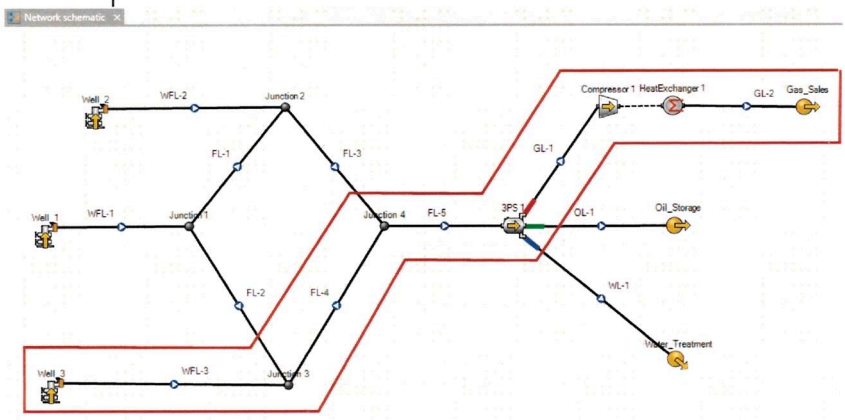
23. View the Profile results.

- To select all the branches and display them in a plot, click the box.
- To display only one branch or specific set of branches, select the individual row or select multiple rows.



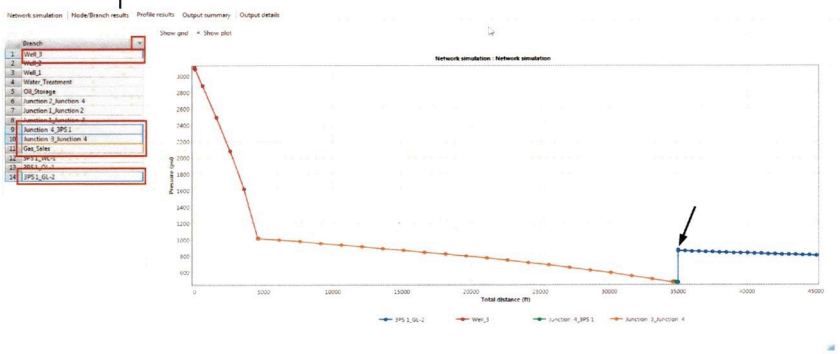
24. To plot the profile results for the flow path from Well_3 to the Gas_Sales sink, as annotated in the figure, select the highlighted branches.

    Observe the 400 psi pressure boost provided by the Compressor.



Double-click the arrow in the *Branch* column to sort in decreasing alphabetical order.

NOTE: The arrow in the figure points to the 400 psi pressure differential from the Compressor.



25. Review the Node/Branch results and determine the network deliverability.

26. Record your results in the following table.

Results	
Gas flow rate to Gas Sales (mmscf/d)	
Oil flow rate going to Oil Storage (STB/d)	
Water flow rate to treatment (STB/d)	

Network simulation | Node/Branch results | Profile results | Output summary | Output details

Display mode: Node Branch

Type filter: All

Select columns... Expand all

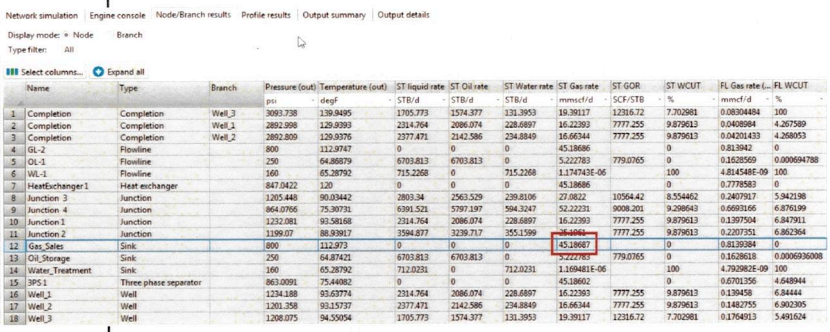
	Name	Type	Branch	Pressure (out)	Temperature (out)	ST liquid rate	ST Oil rate	ST Water rate	ST Gas rate	ST GOR	ST WCUT	FL Gas rate (...	FL WCUT
				psi	degF	STB/d	STB/d	STB/d	mmscf/d	SCF/STB	%	mmcf/d	%
1	Completion	Completion	Well_1	2892.616	129.936	2441.181	2200.001	241.1792	17.10997	7777.255	9.879613	0.0431485	4.268525
2	Completion	Completion	Well_3	3093.472	139.9474	1778.293	1641.312	136.9816	20.21559	12316.72	7.702981	0.0865786	100
3	Completion	Completion	Well_2	2892.422	129.9343	2504.962	2257.481	247.4805	17.55701	7777.255	9.879613	0.0442845	4.268998
4	Compressor 1	Compressor		870.2161	150.0063	0	0	0	49.55957		0	0.8911371	0
5	GL-2	Flowline		800	112.9419	0	0	0	49.55957		0	0.8919303	0
6	OL-1	Flowline		250	58.31163	7281.533	7281.533	0	3.149002	432.4641	0	0.0551954	0.0002705463
7	WL-1	Flowline		160	58.67066	749.3323	0	749.3323	6.21659E-07		100	4.332506E-09	100
8	HeatExchanger 1	Heat exchanger		855.2161	120	0	0	0	49.55957		0	0.843416	0
9	Junction_3	Junction		1008.959	84.82224	2938.743	2687.134	251.6084	28.34726	10549.25	8.561769	0.311105	6.201101
10	Junction_4	Junction		474.9088	61.94341	6721.208	6095.912	625.2964	54.85296	8998.319	9.303333	1.390267	7.630885
11	Junction 1	Junction		1045.48	88.90203	2441.181	2200.001	241.1792	17.10997	7777.255	9.879613	0.1798045	7.131513
12	Junction 2	Junction		1001.876	84.48725	3784.451	3410.562	373.8891	26.52481	7777.255	9.879613	0.2894245	7.160905
13	Gas_Sales	Sink		800	112.8748	0	0	0	49.55957		0	0.8917709	0
14	Oil_Storage	Sink		250	58.31444	7281.533	7281.533	0	3.149002	432.4641	0	0.0551968	0.0002704715
15	Water_Treatment	Sink		160	58.67066	748.5409	0	748.5409	6.210025E-07		100	4.327931E-09	100
16	3PS 1	Three phase separator		472.6779	62.03906	0	0	0	49.56338		0	1.398004	8.333915
17	Well_1	Well		1048.258	88.98719	2441.181	2200.001	241.1792	17.10997	7777.255	9.879613	0.1792512	7.128045
18	Well_2	Well		1004.934	88.04487	2504.962	2257.481	247.4805	17.55701	7777.255	9.879613	0.1935089	7.189909
19	Well_3	Well		1012.393	88.3918	1778.293	1641.312	136.9816	20.21559	12316.72	7.702981	0.225397	5.711925

27. You now determine the drop in Gas production in the event of a compressor shutdown. This scenario assumes that there is a bypass line around the compressor that allows gas production to be maintained if the compressor goes down.

28. To simulate this scenario, right-click the Compressor and select **Deactivate**.

29. Re-run the network simulation task and compare your results with the answer in the table.

Results	
Gas sales (without compressor) (mmscf/d)	


Network simulation | Engine console | Node/Branch results | Profile results | Output summary | Output details

Display mode: Node Branch

Type filter: All

Select columns... Expand all

	Name	Type	Branch	Pressure (out)	Temperature (out)	ST liquid rate	ST Oil rate	ST Water rate	ST Gas rate	ST GOR	ST WCUT	FL Gas rate (...	FL WCUT
				psi	degF	STB/d	STB/d	STB/d	mmscf/d	SCF/STB	%	mmcf/d	%
1	Completion	Completion	Well_3	3093.738	139.9495	1705.773	1574.377	131.3953	19.39117	12316.72	7.702981	0.08304484	100
2	Completion	Completion	Well_1	2892.998	129.9393	2314.764	2086.074	228.6897	16.22393	7777.255	9.879613	0.0408984	4.267589
3	Completion	Completion	Well_2	2892.809	129.9376	2377.471	2142.586	234.8849	16.66344	7777.255	9.879613	0.04201433	4.268053
4	GL-2	Flowline		800	112.9747	0	0	0	45.18686		0	0.813942	0
5	OL-1	Flowline		250	64.86879	6703.813	6703.813	0	5.222783	779.0765	0	0.1628569	0.000694788
6	WL-1	Flowline		160	65.28792	715.2268	0	715.2268	1.174743E-06		100	4.814548E-09	100
7	HeatExchanger1	Heat exchanger		847.0422	120	0	0	0	45.18686		0	0.7778583	0
8	Junction_3	Junction		1205.448	90.03442	2803.34	2563.529	239.8106	27.0822	10564.42	8.554462	0.2407917	5.942198
9	Junction_4	Junction		864.0766	75.30731	6391.521	5797.197	594.3247	52.22231	9008.201	9.298643	0.6693166	6.876199
10	Junction1	Junction		1232.081	93.58168	2314.764	2086.074	228.6897	16.22393	7777.255	9.879613	0.1397504	6.847911
11	Junction2	Junction		1199.07	88.93917	3594.877	3239.717	355.1599	25.0952	7777.255	9.879613	0.2207351	6.862364
12	Gas_Sales	Sink		800	112.973	0	0	0	45.18687		0	0.8139384	0
13	Oil_Storage	Sink		250	64.87421	6703.813	6703.813	0	5.222783	779.0765	0	0.1628618	0.000693608
14	Water_Treatment	Sink		160	65.28792	712.0231	0	712.0231	1.169481E-06		100	4.792982E-09	100
15	3PS 1	Three phase separator		863.0091	75.44082	0	0	0	45.18602		0	0.6701356	4.648944
16	Well_1	Well		1234.188	93.63774	2314.764	2086.074	228.6897	16.22393	7777.255	9.879613	0.139458	6.84444
17	Well_2	Well		1201.358	93.15737	2377.471	2142.586	234.8849	16.66344	7777.255	9.879613	0.1482755	6.902305
18	Well_3	Well		1208.075	94.55054	1705.773	1574.377	131.3953	19.39117	12316.72	7.702981	0.1764913	5.491624



Exercise 2 Screen the network for erosion issues

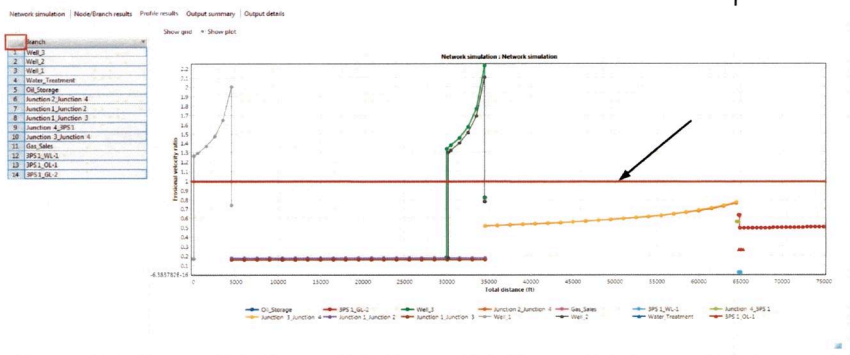
In this case study, your goal is to screen the network for possible erosion limit violations and identify branches that exceed the erosion velocity limit.

1. To re-activate the compressor, right- click and select **Activate**.
2. Navigate to *Home* > *Simulation settings* > *Erosion/Corrosion*.
3. Confirm that the API 14e erosion model is selected and the default Erosional velocity constant (C value) of 100 is being used.
4. Run the network simulation task.
5. On the **Profile results** tab, change the Y-axis variable to display Erosion velocity ratio. The erosional velocity ratio (EVR) is calculated as:

$$EVR = \frac{Fluid\ velocity}{Erosion\ velocity\ calculated\ by\ the\ Erosion\ model}$$

If EVR > 1, there is an erosion risk.

6. Display the EVR plot for all branches.The arrow points to the EVR limit for all branches.



7. Identify the branches where the EVR exceeds one and record your answers.

Results	
Branches where EVR > 1	

Why do these branches have the highest EVR?

(These are the well branches. They have the highest flowing pressures and temperatures, hence highest fluid velocities. They also have smaller pipe diameter sizes than the surface flow lines).

8. Determine possible solutions to get rid of the erosion issues and implement them in the network.

Review questions



- How many boundary conditions must be supplied by the user in a **PIPESIM** network model?
- What are the rules for pressure and flow rate specification in a **PIPESIM** Network model?

Summary

In this module, you learned about:

- building a model of the network
- specifying the network boundary conditions
- solving the network and establishing the deliverability
- identifying the network branched for erosion limits

NOTES

NOTES

Module 8 Simple network model on the GIS map

The Geographic Information System (GIS) map capability introduced in **PIPESIM 2013** provides an additional option to build networks. In total, there are three methods available to create a network in **PIPESIM**.

- build on the logical network canvas
- build directly on the GIS map
- import a shape file to automatically create the network

Building the network on the logical network canvas is the method that was used for the Subsea Tieback and Looped Network exercises. After you build the network in the logical view, you can superimpose it on a GIS map by launching the GIS map and selecting a target area. You reposition the equipment and pipeline routes to replicate the actual layout.

The second option, building the network model directly on the GIS map, is the focus of this module.

The third (and more advanced option) is to import a shape file to automatically create the network. This is a feature that was introduced in **PIPESIM 2014** and is not covered in this training. For more details about this option, refer to the topic *Creating a Network Model from a GIS Shapefile Automatically*, and refer to two tutorials available in the **PIPESIM** Help.

- Automatic Creation of a Network Model on the GIS Map
- Investigation of the Use of Inline Heating for Wax Mitigation

This module discusses how to build a pipeline network directly on the GIS map. This feature allows you to build a model that more closely represents real-world conditions. The exact pipeline terrain profiles follow the real route on the earth and they can be automatically collected by **PIPESIM**. The captured elevation data is used by **PIPESIM** for more accurate pressure drop calculations.



Learning objectives

After completing this module, you will know how to:

- add a GIS map to the model
- draw pipelines following terrain features
- collect and display elevation data for the pipelines using GIS maps



Lesson 1 Pipeline modeling

Accurate modeling of pipelines in many areas of the world (on land and subsea) requires knowledge of the elevation changes along the pipeline route; historically, collecting that data has not been a trivial exercise. In recent years, however, GIS web services have been developed to make geo-referenced map images available directly to users or for access by software, such as **PIPESIM**.



Exercise 1 Build the network model on a map

1. Create a new network-centric workspace.
2. On the **Home** tab, click the **GIS map** icon to launch the GIS map.

   A GIS map provided by Esri automatically appears if you have Internet access and there are no firewalls preventing access to the Esri site. The default map is the Esri World Satellite map and it is one of several Esri maps built into **PIPESIM**. You have the option to set up connections to other Esri maps or to other map services, such as Bing and even corporate GIS map services.

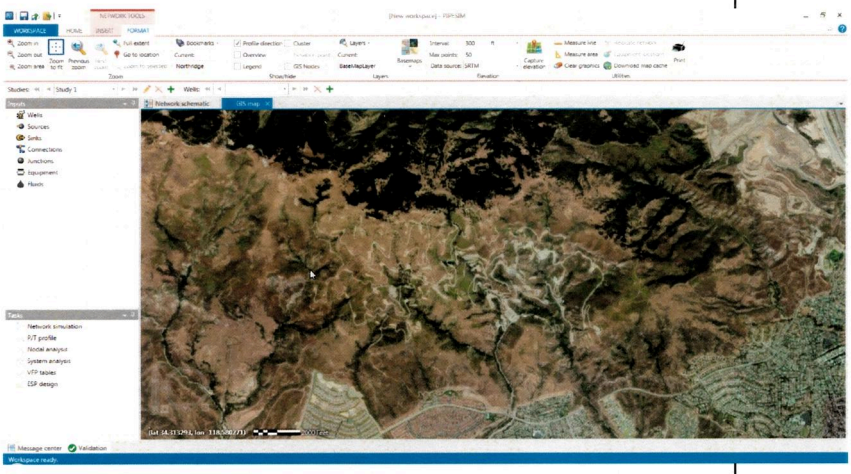
   For details, refer to topics in the **PIPESIM** Help: *Choosing a Basemap*, *Using Map Services*, and *Adding Bing Basemaps*.

3. With the GIS map open, click the **Format** tab to display the expanded functionality available for GIS networks.



NOTE: There are several ways to zoom into a location on the map to build the network. You can manually zoom into an area, import a shape file, or open a map cache of a limited map area. After zooming in on the area of interest, save the location to the workspace as a Bookmark so you can reuse it. Bookmarks are workspace-specific and can be repeatedly used, but only in the workspace in which they are saved. **PIPESIM** also has built-in bookmarks.

4. Locate the *Bookmarks* list on the **Format** tab and select Northridge from the drop-down list. The map zooms into the Northridge area.

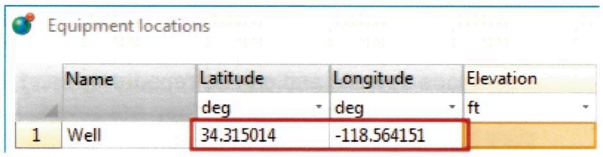




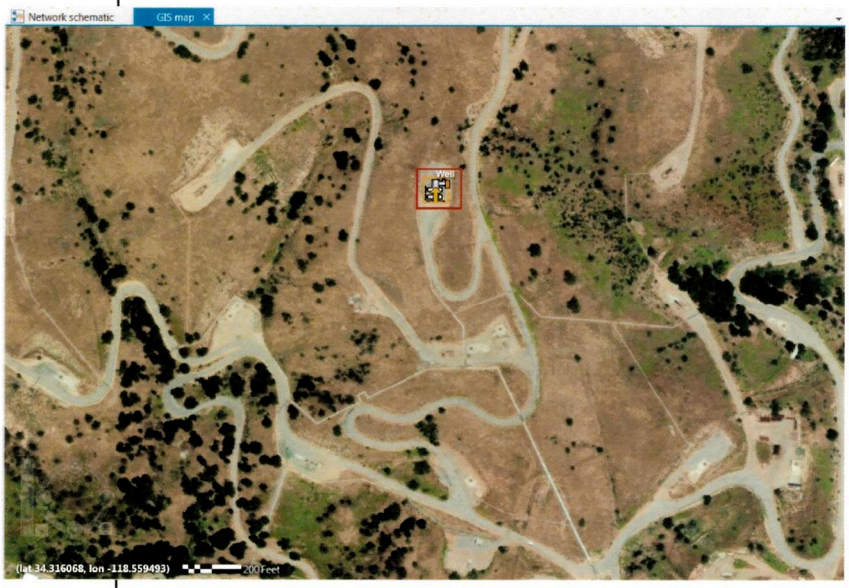
TIP: To pan the map (move left/right and up/down), hold the Ctrl key and click the mouse, then move left/right and up/down.

5. Go to the **Insert** tab and insert a well anywhere on the map.
6. Select the Simple vertical template for it, when prompted.

7. To position the well at an exact geographic location:
   a. Go to the **Format** tab and click the **Equipment Locations** icon.
   b. Enter the Lat-Long coordinates for the well, as shown in the figure.



8. Use the **Zoom area** feature on the **Format** tab to zoom in on an areal extent around a well that resembles the one depicted in the figure.



9. As closely as possible, insert additional objects in the locations that are depicted in the figure. (The exact locations are unimportant.)
   - A second well in the southeast (use the Simple Vertical template)
   - A choke near the first well
   - One junction
   - One sink



10. From the **Insert** tab, click the **Flowline** icon.

11. Draw the first flowline to connect the Choke 1 to Junction 1, following the path shown in the figure.

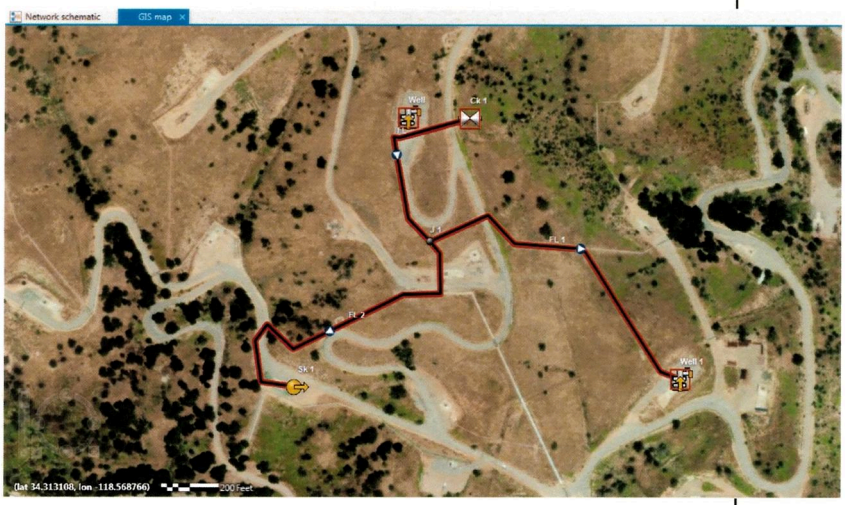
TIP: Trace the path of the existing flowline in the GIS map view that appears as a thin white line.



12. Draw the second flowline to connect Well-1 to Junction 1 to Well-1, again following as closely as possible the path shown in the figure. Again, trace the path of the thin white line, which represents an existing flowline on the GIS map.



13. Draw the third (final) flowline to connect Junction 1 to Sink 1.



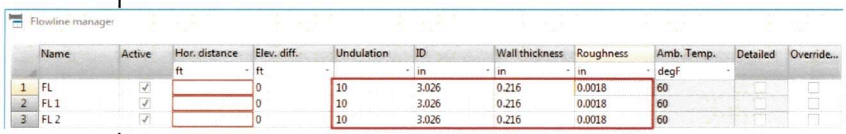
14. On the **Insert** tab, click the **Connector** icon.

15. Draw a connector from the first well to the Choke. Your map should match the map in the figure.



16. From the **Home** tab, select **Flowline Manager**.
17. Enter these values for all flow lines in the network:
    - Common flowline ID: 3.026 inch
    - Wall thickness: 0.216 inch
    - Pipe roughness: 0.0018 inch
18. To fill down a value in a column, for example ID of 3.026 inch, enter the value in the first row and press F3.

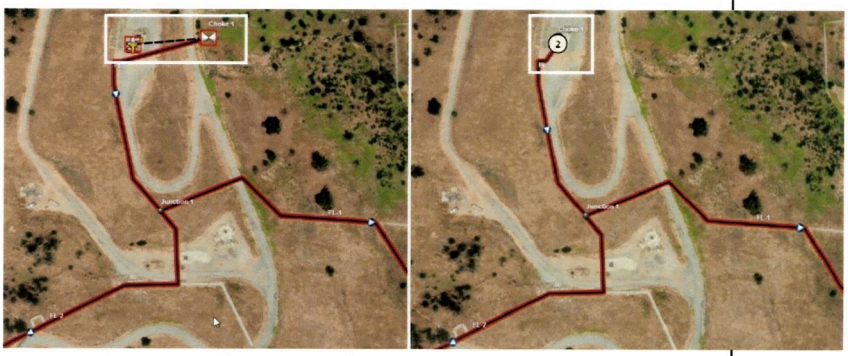
    Leave the *Hor. Distance* column blank; it is populated when you use the **PIPESIM** GIS Elevation Capture feature.



	Name	Active	Hor. distance	Elev. diff.	Undulation	ID	Wall thickness	Roughness	Amb. Temp.	Detailed	Override...
			ft	ft		in	in	in	degF		
1	FL	☑		0	10	3.026	0.216	0.0018	60	☐	☐
2	FL 1	☑		0	10	3.026	0.216	0.0018	60	☐	☐
3	FL 2	☑		0	10	3.026	0.216	0.0018	60	☐	☐

19. On the **Format** tab, select the *Cluster* check box. This action groups together all the items joined by connectors into a single node that represents the actual physical location of these objects.

    The number in the node (circle) indicates the number of objects in the cluster. The unclustered (original) and clustered views display on the left and right, respectively.



20. Capture the elevations.

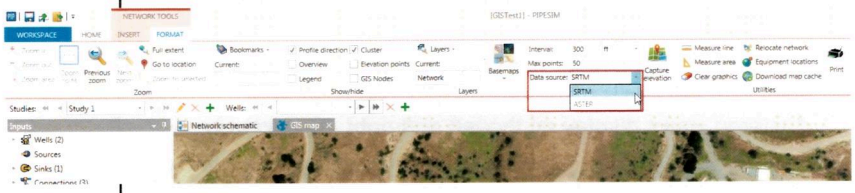
    a. Select the *Elevation points* box in the Show/hide group of the **Format** tab. Red circles have been added to the flow lines. The circles indicate the positions along the flow lines where the points were captured from the map.

    b. To capture elevations at more locations along the flow lines, go to the **Format** tab and change the interval from the default value of 300 ft to `60` ft in the *Interval* box of the Elevation group.

       The number of red circles (elevation points) along the flow lines increases.

    c. On the **Format** tab, in the Elevation group, there are two elevation data sources available: SRTM and ASTER. Leave the default SRTM service selected.

For more detail on the data sources, refer to the **PIPESIM** Help topic: *ASTER and SRTM Elevation Data Sources*.

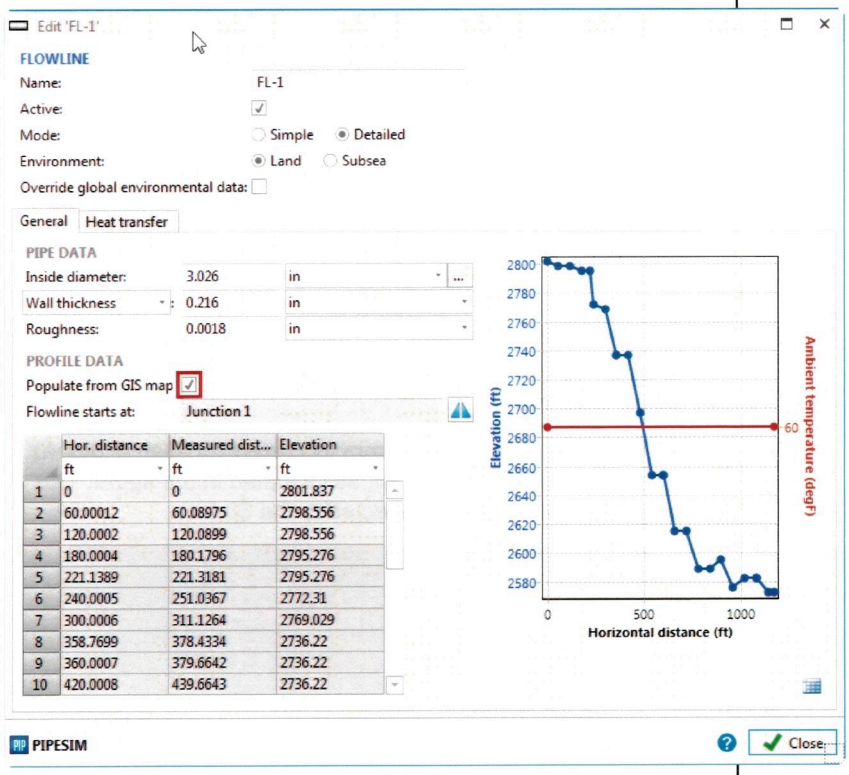


d. Click **Capture elevation**. The elevation points change to green if the operation was successful. The flow lines are no longer invalid and the map should match the figure.



21. (Optional) Select the *GIS nodes* box. This setting causes red square boxes to display at the end points of each flowline segment that correspond to a change in direction.

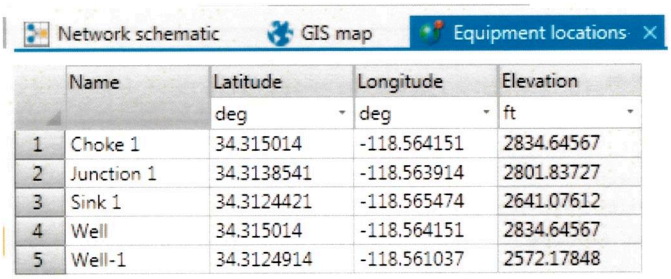
22. Double-click any flowline to view the geometry profile data. The *Populate from GIS map* box is selected because the profile data was captured from the map.



NOTE: If you have profile data for the flow lines from another source, clear the **Populate from GIS map** box. Copy this data and paste it into the grid. This data is termed the *logical* profile data to differentiate it from the GIS profile data that was captured from the map.

The profile data from both sources is stored. To run the network simulation with the logical or GIS profile data, select or clear the *Populate from GIS map* box before the run.

23. (Optional) To view the GIS map locations for the various equipment (chokes, junctions, well, etc.), click **Equipment locations** in the Utilities group on the **Format** tab.



	Name	Latitude	Longitude	Elevation
		deg	deg	ft
1	Choke 1	34.315014	-118.564151	2834.64567
2	Junction 1	34.3138541	-118.563914	2801.83727
3	Sink 1	34.3124421	-118.565474	2641.07612
4	Well	34.315014	-118.564151	2834.64567
5	Well-1	34.3124914	-118.561037	2572.17848

24. You can reposition any object on the GIS map. In this step, you reposition the Sink.

    On the **Format** tab, click **Zoom out** once to view a slightly larger areal extent.

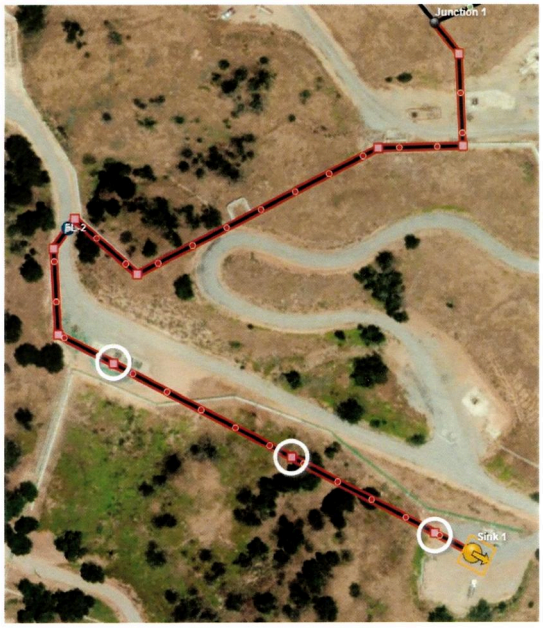
25. On the map, drag the sink from its current location (Point A) to the new location (Point B), as indicated in the figure. This re-orients the flowline connected to the Sink.



26. The flowline connected to the sink is now invalid. (It is red, and the dots on it that represent the elevation points have changed to red.) You repositioned the sink, rendering the previously-captured elevation data invalid, so you must recapture the elevation.
27. Before you do that, further alter the path for this flowline.



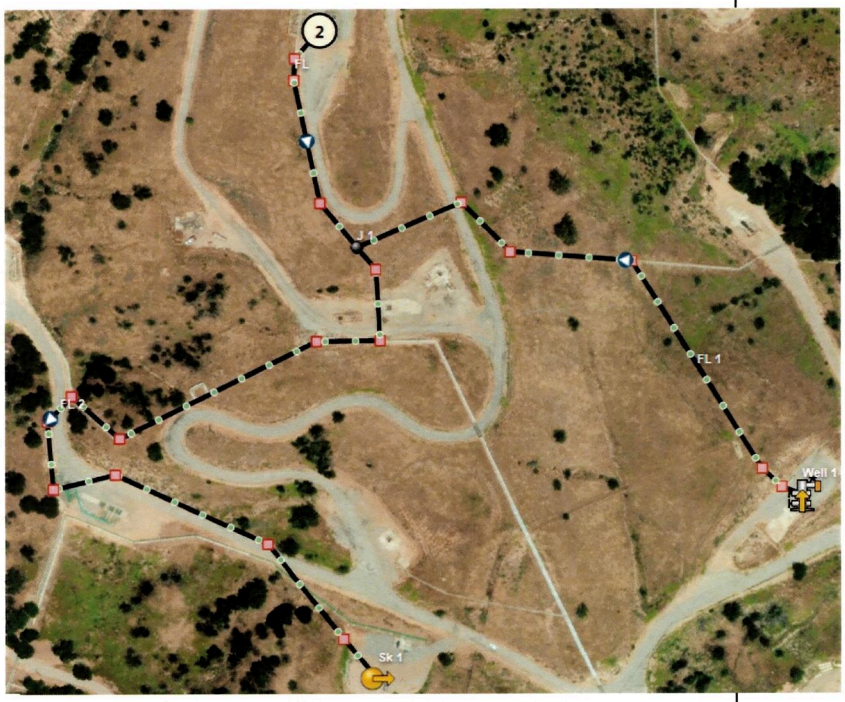
28. Alter the path for the flowline connected to the sink. From the **Insert** tab, click the **Add node** icon and add three intermediate nodes to the flowline at the locations indicated in the figure.



29. Move the first two intermediate nodes upwards to the new positions indicated on the map in the figure that follows.
30. Move the Sink downwards until the flowline trajectory resembles the trajectory in the figure. To match the trajectory, move the nodes and the sink in the directions indicated by the arrows.

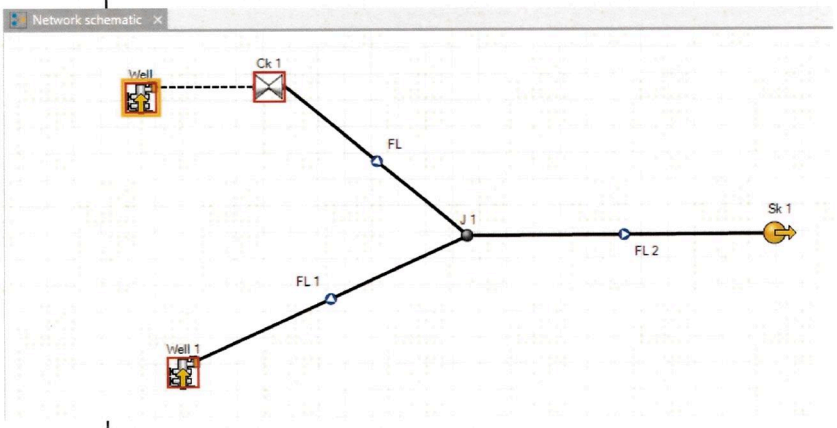


The overall network should match the network map in the figure.

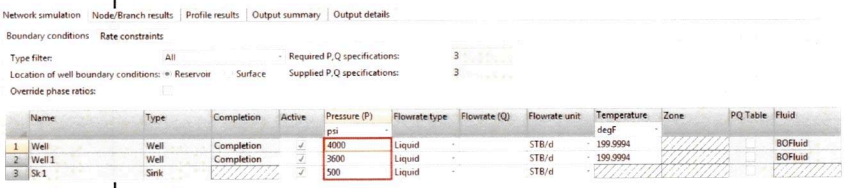


31. From the **Format** tab, click **Capture elevations** to recapture the elevations, then save your workspace.
32. In the **Inputs** pane on the left, expand the *Equipment* list, then double-click the Choke to enter a bean size of 1 inch.
33. Close the choke editor.
34. Exit the GIS map by closing it. You now are in the logical network view.

35. Rearrange the equipment and flow lines, if needed. (This does not affect the equipment locations in the GIS map view.)



36. Go to the **Fluid manager** on the **Home** tab and verify that Black oil is the selected option.
37. Launch the **Fluid manager**.
38. Click the green plus sign + to create a new fluid.
39. When prompted, select the Dry gas template from the drop-down list and click **OK**.
40. Click the **Fluid mapping** tab.
41. To map both wells to the new black oil fluid you created, select it from the drop-down list.
42. Exit the **Fluid manager** and save your workspace.
43. Launch the Network Simulation task and specify the boundary conditions shown in the figure.



44. Run the task and record your results in the table.

TIP: On the **Node/Branch results** tab, click **Expand all** to extract some of the results.



Results	
Property	**Value**
Gas flow rate at Sink (mmscf/d)	
Gas flow rate from Well (mmscf/d)	
Gas flow rate from Well-1 (mmscf/d)	
Differential pressure across Choke (psi)	
Outlet Pressure from the Junction (psi)	



NOTE: Your answers do not exactly match the answers in the Answer key because of slight differences in the way flowlines are laid out in the GIS map view.

Review question



Why is it important to have elevational data available when simulating a pipeline?

Summary

In this module, you learned about:

- adding a map to a model
- drawing pipelines following terrain features
- collecting and displaying elevation data for the pipeline

NOTES

Appendix A PIPESIM 2014 Fundamentals: Answer key

This appendix contains the answers and/or results from exercises for you to compare to your findings.

Module 2: Simple pipeline tutorial

Lesson 1: Single-phase flow calculations

Exercise 1: Model a water pipeline with hand calculations

Water velocity	~ 7.6 ft/sec
Reynold's number	~ 153,400; turbulent flow
Friction fFactor	~ 0.01984
dP(friction)	~ 616 psi
dP(elevation)	~ 442 psi
dP(Total)	1,058 psi
Outlet pressure	142 psia

Exercise 2: Model a water pipeline with hand calculations and PIPESIM

Result	Hand calculation	PIPESIM
Liquid Velocity (ft/s)	~ 7.6	~ 7.6
$\Delta P_{frictional}$ (psi)	~ 616.0	~ 611.0
$\Delta P_{elevational}$ (psi)	~ 442.0	~ 443.0
ΔP_{total} (psi)	~ 1,058.0	~ 1,054.0
Outlet pressure (psia)	~ 142.0	~ 146.0

Exercise 6: Calculate a gas pipeline flow capacity

Gas flow rate	10.7 mmscf/d

Module 3: Oil well performance analysis

Lesson 1: Nodal analysis

Exercise 2: Perform a nodal analysis

Operating point – flow rate	8,840 STB/d
Operating point – BHP	2,495 psi
AOFP	21,311 STB/d

Lesson 2: Pressure/temperature profile

Exercise 1: Generate a pressure/temperature profile

Production rate	8,840 STB/d
Flowing BHP	2,495 psi
Flowing WHT	138 degF
Depth at which gas appears	Between 7,082 ft and 7,199 ft

Lesson 3: Black oil fluid calibration

Exercise 1: Calibrate PVT data

Results		
Wellhead Pressure = 300 psi	**Uncalibrated**	**Calibrated**
Production Rate, STB/d	8,840	8,171
Flowing BHP, psi	2,495	2,579
Flowing WHT, degF	138	134
AOFP, STB/d	21,311	21,320
Depth where gas appears	Between 7,082 ft and 7,199 ft	Between 6,848 ft and 6,965 ft

Lesson 4: Inflow performance matching

Exercise 1: Sensitize on the well PI to match performance

Matched Productivity Index	9,375 STB/d/psi
New AOFP	24,984 STB/d
Previous AOFP	21,320 STB/d

Lesson 5: Well performance analysis

Exercise 1: Run a system analysis task

Critical water cut	71%

Lesson 6: Artificial lift

Exercise 1: Evaluate gas lift performance

Water cut, %	Optimum gas injection rate, mmscf/d	Liq. Prod. Rate, STB/d
70	6	10,38110381

Water cut, %	Available gas injection rate, mmscf/d	Liq. Prod. Rate, STB/d
70	3	9,3069306

Exercise 2: Evaluate ESP performance

ESP Design Results (Water cut = 70%)	
ESP Model	TE11000
No. of stages	68.0
Efficiency, %	72.7
Power, hp	255.0
Head, ft	2,485.0
Differential pressure, psi	1,017.0
Fluid temperature rise, degF	1.4

Exercise 3: Model multiple completions

Self-lifting results	
Liquid Rate (STB/d)	9,593
Gas Rate (upper zone) (mmscf/d)	3.45

Lesson 7: Downhole chokes

Exercise 1: Model a downhole choke

Results	
Choke bean size required to constrain gas rate to 4 mmscf/d (inches)	1.05
Liquid rate (STB/d)	8,200

Module 4: Gas well performance

Lesson 2: Gas well deliverability

Exercise 1: Calculate gas well deliverability

Results	
Gas Rate (mmscf/d)	17.881
Flowing bottomhole pressure (psi)	1,810.9
Bottomhole temperature (degF)	245.3
Wellhead temperature (degF)	175.6

Exercise 2: Calibrate the inflow model using multipoint test data

Results	Well PI	Calibrated back pressure
Gas Rate (mmscf/d)	17.881	14.696
Flowing bottomhole pressure (psi)	1,810.9	1,656
Bottomhole temperature (degF)	245.3	242.6
Wellhead temperature (degF)	175.6	170.4

Back pressure equation	
Parameter C (mmscf/d/psi^2n)	7.98 E-7
Parameter n	1

Lesson 3: Erosion prediction

Exercise 1: Select a tubing size

Results	
Selected tubing ID (inches)	3.476
Gas rate (mmscf/d)	14.7
Flowing bottomhole pressure (psi)	1,655.5
Flowing bottomhole temperature (degF)	242.6
Flowing wellhead temperature, degF	170.4
Max Erosional velocity ratio	0.9847

Lesson 4: Choke modeling

Exercise 1: Model a flowline and choke

Results	
Po = 710 psia	
Choke size (inches)	1.502925

Results	
Updated choke size = 1.50 inches	
Static reservoir pressure (psi)	4,600
Flowing bottomhole pressure (psi)	1,654.6
Flowing wellhead pressure (psi)	799.1
Flowing pressure immediately downstream of choke (psi)	712.3
Outlet pressure (psi)	710.6
Pressure losses across system	
Δ Reservoir (psi)	2,945.4
Δ Tubing (psi)	855.4
Δ Choke (psi)	86.9
Δ Flowline (psi)	1.7

Exercise 2: Predict future production rates

Results	
Reservoir pressure (psi)	**Gas rate (mmscf/d)**
4,600	14.696290
4,200	12.182110
3,800	9.852769
3,400	7.731681

Lesson 5: Liquid loading

Exercise 1: Determine the critical gas rate to prevent well loading

Critical gas flow rate (mmscf/d)	3.63

Module 5: Horizontal well design

Lesson 1: Inflow performance for horizontal completions

Exercise 1: Horizontal well performance

Gas production rate (mmscf/d)	47.55

Module 6: Subsea tieback design

Lesson 1: Flow assurance considerations

Exercise 1: Size the subsea tieback and riser

Minimum Tieback and Riser diameters that satisfy the minimum arrival pressure constraint of 400 psi for all flow rates (inches)	10.02

Results	
Property	**Value**
Selected Tieback and Riser ID based on 400 psi arrival pressure constraint (inches)	10.020
Max. erosional velocity ratio for selected ID	0.755
Minimum arrival pressure for selected ID (psi)	962.700
Maximum arrival pressure for selected ID (psi)	1267.200

Lesson 2: Hydrates

Exercise 1: Select tieback insulation thickness

Results	
Property	**Value**
Req. Insulation thickness (inch)	1

Exercise 2: Determine the methanol requirement

Results	
Property	**Value**
Req. Methanol Injection Volume (STB/d)	280

Lesson 3: Severe riser slugging

Exercise 1: Screen for severe riser slugging

	Results		
Severe slugging	**8,000 STB/d**	**14,000 STB/d**	**16,000 STB/d**
Severe slugging indicator	1.198	1.475	1.569

Lesson 4: Slug catcher sizing

Exercise 1: Size a slug catcher

	Results		
Slug catcher sizing	**8,000 STB/d**	**14,000 STB/d**	**16,000 STB/d**
1/1000 slug volume (bbl)	164.93	185.48	226.61
Sphere generated liquid volume (bbl)	456.08	424.50	413.90
Total liquid holdup (bbl)	875.15	765.08	728.16
Ramp-up volume (bbl)	875.15 – 728.16 = 146.99		
Design volume for slug catcher (bbl)	456.08 * 1.2 = 547.30		

Module 7: Looped gas gathering network

Lesson 1: Model a gathering network

Exercise 1: Model a pipeline network

Results	
Gas flow rate to Gas Sales (mmscf/d)	49.56
Oil flow rate going to Oil Storage (STB/d)	7,281.53
Water flow rate to treatment (STB/d)	748.54

Results	
Gas sales (without compressor) (mmscf/d)	45.19

Exercise 2: Screen the network for erosion issues

Results	
Branches where EVR > 1	Well_1, Well_2, and Well_3 Connected flow lines WFL-1, WFL-2, and WFL-3, respectively.

Module 8: Simple network model on the GIS map

Lesson 1: Pipeline modeling

Exercise 1: Build the network model on a map

Results	
Property	**Value**
Gas flow rate at Sink (mmscf/d)	45.07
Gas flow rate from Well (mmscf/d)	23.69
Gas flow rate from Well-1 (mmscf/d)	21.69
Differential pressure across choke (psi)	305.20
Outlet pressure from the junction (psi)	1,617.10

NOTES

NOTES