

---

# **Aspen Plus**

## **Getting Started**

## **Modeling Petroleum**

## **Processes**

*Version 11.1*



## **Part Number: Aspen Plus 11.1 September 2001**

Copyright (c) 1981-2001 by Aspen Technology, Inc. All rights reserved.

Aspen Plus<sup>®</sup>, Aspen Properties<sup>®</sup>, Aspen Engineering Suite<sup>™</sup>, AspenTech<sup>®</sup>, ModelManager<sup>™</sup>, the aspen leaf logo and Plantelligence are trademarks or registered trademarks of Aspen Technology, Inc., Cambridge, MA.

BATCHFRAC<sup>™</sup> and RATEFRAC<sup>™</sup> are trademarks of Koch Engineering Company, Inc.

All other brand and product names are trademarks or registered trademarks of their respective companies.

This manual is intended as a guide to using AspenTech's software. This documentation contains AspenTech proprietary and confidential information and may not be disclosed, used, or copied without the prior consent of AspenTech or as set forth in the applicable license agreement. Users are solely responsible for the proper use of the software and the application of the results obtained.

Although AspenTech has tested the software and reviewed the documentation, the sole warranty for the software may be found in the applicable license agreement between AspenTech and the user. **ASPENTECH MAKES NO WARRANTY OR REPRESENTATION, EITHER EXPRESSED OR IMPLIED, WITH RESPECT TO THIS DOCUMENTATION, ITS QUALITY, PERFORMANCE, MERCHANTABILITY, OR FITNESS FOR A PARTICULAR PURPOSE.**

### **Corporate**

Aspen Technology, Inc.  
Ten Canal Park  
Cambridge, MA 02141-2201  
USA  
Phone: (1) (617) 949-1021  
Toll Free: (1) (888) 996-7001  
Fax: (1) (617) 949-1724  
URL: <http://www.aspentech.com>

### **Division**

Design, Simulation and Optimization Systems  
Aspen Technology, Inc.  
Ten Canal Park  
Cambridge, MA 02141-2201  
USA  
Phone: (617) 949-1000  
Fax: (617) 949-1030

---

# Contents

<b>About This Manual</b>	<b>1-1</b>
Modeling Petroleum Processes with Aspen Plus .....	1-2
Simulations in this Book .....	1-3
Using Backup Files .....	1-3
Related Documentation .....	1-4
Getting Started Guides .....	1-4
Installation Guide .....	1-4
User Guide.....	1-4
Technical Support .....	1-4
World Wide Web .....	1-4
Online Technical Support Center .....	1-4
Contacting Customer Support .....	1-5
Hours .....	1-5
Phone .....	1-5
Fax .....	1-6
E-mail .....	1-6
 <b>Blending Crude and Petroleum Fractions</b>	 <b>2-1</b>
Petroleum Process Flowsheet .....	2-2
Starting Aspen Plus .....	2-4
To Start Aspen Plus .....	2-4
To Select the Template Option.....	2-4
To Specify the Application Type and Run Type .....	2-4
Specifying Title and Global Options .....	2-5
Specifying Components .....	2-7
Entering Assay Data .....	2-8
Specify the Distillation Curve for OIL-1 .....	2-8
Specify the Light Ends Data for OIL-1 .....	2-9
Specify the API Gravity Data for OIL-1 .....	2-9
Specify the Distillation Curve for OIL-2 .....	2-10
Specify the Light Ends Data for OIL-2 .....	2-11
Specify the API Gravity Data for OIL-2 .....	2-11
Blending the Oils .....	2-13
Generating Pseudocomponents .....	2-14
Running the Simulation .....	2-15
Examining Results .....	2-15

Plotting Distillation Curves.....	2-16
To Locate the Curve Data for OIL-1 .....	2-16
To Plot the TBP Curve.....	2-17
To Label This Curve for OIL-1 .....	2-17
To Plot the TBP Curve for OIL-2.....	2-18
To Add This Curve to the Plot Containing the Curve for OIL-1 .....	2-18
To Label This Curve for OIL-2 .....	2-18
To Plot the BLEND Curve.....	2-19
To Add This Curve to the Plot.....	2-19
To Label This Curve for MIXOIL.....	2-19
To Combine Y-Axes .....	2-20
To Rename the Plot.....	2-20
Creating a Backup File.....	2-21
Exiting Aspen Plus.....	2-21

## **Adding a Preflash Tower 3-1**

Preflash Tower Flowsheet.....	3-2
Starting Aspen Plus .....	3-3
Opening an Existing Run .....	3-3
Opening the Assay Data Analysis Simulation Created in Chapter 2 .....	3-3
If Your Saved File blend.bkp Does Not Appear .....	3-3
If You Did Not Create blend.bkp .....	3-3
Saving a Run Under a New Name.....	3-4
Changing the Run Type.....	3-4
Turning Off Automatic Naming of Blocks and Streams.....	3-4
Selecting a Distillation Model.....	3-5
Defining the Graphical Simulation Flowsheet.....	3-6
To Choose a Different Icon for the Model .....	3-6
Place the Preflash Tower on the Flowsheet.....	3-6
Place the Streams on the Flowsheet .....	3-7
Specifying Properties .....	3-7
To Select the BK10 Property Method .....	3-8
Entering Stream Data .....	3-9
Specifying the Preflash Unit.....	3-10
Specify Column Configuration .....	3-10
Specify Stream Locations.....	3-11
Specify Column Pressure Profile .....	3-11
Specify Condenser Temperature .....	3-12
Specify Furnace Configuration.....	3-12
Create Design Specification .....	3-12
Modifying Pseudocomponents.....	3-13
Running the Simulation.....	3-14
Examining Simulation Results.....	3-15
To Examine Summary Results.....	3-15
To Examine Distillate Rate Results .....	3-15

To Examine the Temperature Profile .....	3-16
Plotting Results .....	3-17
To Plot the Temperature Profile.....	3-17
To Plot TBP Curves for PREFLASH Feed and Product Streams .....	3-17
To Select CDU-FEED and MIXCRUDE as Dependent Variables .....	3-18
Exiting Aspen Plus.....	3-19

## **Adding an Atmospheric Crude Distillation Unit 4-1**

Atmospheric Crude Distillation Unit Flowsheet.....	4-2
Starting Aspen Plus .....	4-4
Opening an Existing Run .....	4-4
If You Saved the Preflash Flowsheet Simulation Created in Chapter 3.....	4-4
If Your Saved File preflash.bkp Does Not Appear .....	4-4
Opening the Backup File preflash.bkp in the Examples Folder.....	4-4
Saving a Run Under a New Name.....	4-5
To Update the Title for This Run.....	4-5
Adding the Crude Distillation Unit to the Flowsheet.....	4-5
To Make Space for the New Tower .....	4-5
To Place and Connect the Distillation Unit on the Flowsheet .....	4-5
To Connect to the Preflash Block Already on the Flowsheet .....	4-6
To Connect Other Streams .....	4-6
Specifying Steam Feeds to the Tower .....	4-7
Specifying the Atmospheric Tower.....	4-9
Specify the Main Column.....	4-9
Specify the Side Strippers .....	4-11
Specify the Pumparounds.....	4-13
Enter Design Specifications .....	4-14
Running the Simulation.....	4-18
Examining Simulation Results.....	4-18
To Examine Furnace Duty Results.....	4-18
To Examine Pumparound Return Temperatures .....	4-19
Examining the Product Streams .....	4-19
To Review the Stream Results for the Crude Tower Products.....	4-19
To Examine the Results for the Five Product Streams.....	4-20
To Examine the 5% and 95% Cutpoints and Product Yields.....	4-20
Plotting Distillation Curves.....	4-20
To Plot the Curves for the Product Streams.....	4-20
Exiting Aspen Plus.....	4-22

## **Adding A Vacuum Unit 5-1**

Vacuum Unit Flowsheet.....	5-2
Starting Aspen Plus .....	5-3
Opening an Existing Run .....	5-3
If You Saved the Crude Flowsheet from Chapter 4 .....	5-3
If Your Saved File crude.bkp Does Not Appear .....	5-3

Opening the Backup File preflash.bkp in the Examples Folder.....	5-3
Saving a Run Under a New Name.....	5-4
To Update the Title for This Run.....	5-4
Adding a Vacuum Tower to the Flowsheet.....	5-5
To Make Space for the New Tower.....	5-5
To Place and Connect the Distillation Unit on the Flowsheet.....	5-5
To Connect the Distillation Unit to the Crude Block.....	5-5
Specifying the Steam Feed.....	5-7
Specifying the Vacuum Unit.....	5-8
Specify the Basic Column Configuration.....	5-8
Specify the Pumparounds.....	5-10
Enter Design Specification.....	5-11
Running the Simulation.....	5-12
Examining Simulation Results.....	5-12
To Examine Column Profiles.....	5-13
To Examine Pumparound Results.....	5-13
Plotting Distillation Curves.....	5-14
To Plot TBP Curves.....	5-14
Examining Stream Results.....	5-15
Exiting Aspen Plus.....	5-16
<b>Displaying Petroleum Stream Properties.....</b>	<b>6-1</b>
Starting Aspen Plus.....	6-2
Opening an Existing Run.....	6-2
If You Saved the Simulation Created in Chapter 5.....	6-2
If Your Saved File vacuum.bkp Does Not Appear.....	6-2
Opening the Backup File preflash.bkp in the Examples Folder.....	6-2
Saving a Run Under a New Name.....	6-3
To Update the Title for This Run.....	6-3
Adding Sulfur Content Data.....	6-3
To Enter Sulfur Content for OIL-1.....	6-3
To Enter Sulfur Content for OIL-2.....	6-4
Selecting Options for the Sulfur Property.....	6-5
Adding the Sulfur Property.....	6-6
To Create the Property Set.....	6-6
To Add the Property Set to the Stream Property List.....	6-8
Running the Simulation.....	6-9
Examining Results.....	6-9
Leaving Aspen Plus.....	6-10
<b>Connecting to the Aspen Plus Simulation Engine.....</b>	<b>7-1</b>

---

# About This Manual

Aspen Plus is a powerful tool that allows you to easily model petroleum processes. Use this book to learn the basics of how to build and run a process model of a simplified crude fractionation train. After completing the five simulations in this guide, you will be able to perform basic simulations of petroleum processes.

This manual assumes you have:

- Installed the Aspen Plus simulation engine and user interface.
- Completed the tutorial simulations in *Getting Started Building and Running a Process Model*, so that you are familiar with the basics of how to use Aspen Plus.

# Modeling Petroleum Processes with Aspen Plus

Petroleum refining processes are highly complex and integrated. They have unique characteristics that set them apart from other chemical processes, including:

- Process feedstocks, which consist of complex and wide-boiling mixtures of hydrocarbons, whose exact compositions are unknown
- Highly-coupled and heat-integrated fractionation units, used to separate feedstocks into a variety of products with different specifications
- Open steam and cooling water for stripping and heat recovery, giving rise to the presence of two liquid phases throughout the refining process
- Degree of separation specified in terms of distillation temperatures, gaps, overlaps, and other properties
- Product specifications given in terms of stream properties such as flash point, pour point, sulfur content, metal contents, and octane number

Aspen Plus provides special features to handle the unique characteristics of petroleum refining applications. With Aspen Plus, develop a simulation model of your petroleum process, then use this model to study alternative modes of operation or optimize the existing operation.

This manual guides you through some of the key petroleum features in Aspen Plus. You can use these features with the wide range of other Aspen Plus capabilities, such as estimating and regressing physical properties, fitting plant data to simulation models, and optimizing your process.



## Simulations in this Book

This book guides you in building a flowsheet that describes a crude fractionation train in a petroleum refinery. Each chapter contains a tutorial simulation that builds one block of the flowsheet. Complete the entire flowsheet by following each chapter in succession.

This book includes the following hands-on simulations:

Complete this chapter	To learn how to
2 Blending Crude and Petroleum Fractions	Define components, enter assay data for two crude oils, and blend the crude oils into a single process feed.
3 Adding a Preflash Tower	Model a preflash tower.
4 Adding an Atmospheric Crude Distillation Unit	Model a crude atmospheric unit.
5 Adding a Vacuum Unit	Model a vacuum unit.
6 Displaying Petroleum Properties	Use petroleum properties with your simulation model to obtain information about the process.

## Using Backup Files

We recommend that you perform all simulations sequentially in order to build the entire model. However, skip chapters and work on the simulation of your choice, using backup files containing simulation data.

Aspen Plus provides backup files containing all problem specifications and results for each tutorial simulation. If you skip a simulation, you might need to load a backup file to supply missing data. The chapter describes how to do this. If you perform each tutorial simulation in order, use backup files to compare your results.

## Related Documentation

In addition to this document, a number of other documents are provided to help users learn and use Aspen Plus. The documentation set consists of the following:

### Getting Started Guides

*Aspen Plus Getting Started Building and Running a Process Model*

*Aspen Plus Getting Started Modeling Processes with Solids*

*Aspen Plus Getting Started Modeling Processes with Electrolytes*

*Aspen Plus Getting Started Using Equation-Oriented Modeling*

*Aspen Plus Getting Started Customizing Unit Operation Models*

### Installation Guide User Guide

*AES Installation Manual*

*Aspen Plus User Guide*

## Technical Support

### World Wide Web

For additional information about AspenTech products and services, visit the AspenTech World Wide Web homepage on the Internet at:

<http://www.aspentech.com/>

### Online Technical Support Center

AspenTech customers with a valid license and software maintenance agreement can register to access the **Online Technical Support** Center at:

<http://support.aspentech.com/>

This web support site allows you to:

- Access current product documentation
- Search for tech tips, solutions and frequently asked questions (FAQs)
- Search for and download application examples
- Submit and track technical issues
- Send suggestions
- Report product defects
- Review lists of known deficiencies and defects

Registered users can also subscribe to our Technical Support e-Bulletins. These e-Bulletins are used to proactively alert users to important technical support information such as:

- Technical advisories
- Product updates and Service Pack announcements

## Contacting Customer Support

Customer support is also available by phone, fax, and email for customers with a current support contract for this product. For the most up-to-date phone listings, please see the Online Technical Support Center at <http://support.aspentech.com>.

## Hours

Support Centers	Operating Hours
North America	8:00 – 20:00 Eastern Time
South America	9:00 – 17:00 Local time
Europe	8:30 – 18:00 Central European time
Asia and Pacific Region	9:00 – 17:30 Local time

## Phone

Support Centers	Phone Numbers	
North America	1-888-996-7100	Toll-free from U.S., Canada, Mexico
	1-281-584-4357	North America Support Center
	(52) (5) 536-2809	Mexico Support Center
South America	(54) (11) 4361-7220	Argentina Support Center
	(55) (11) 5012-0321	Brazil Support Center
	(0800) 333-0125	Toll-free to U.S. from Argentina
	(000) (814) 550-4084	Toll-free to U.S. from Brazil
	8001-2410	Toll-free to U.S. from Venezuela
Europe	(32) (2) 701-95-55	European Support Center
	Country specific toll-free numbers:	
	Belgium	(0800) 40-687
	Denmark	8088-3652
	Finland	(0) (800) 1-19127
	France	(0805) 11-0054
	Ireland	(1) (800) 930-024
	Netherlands	(0800) 023-2511
	Norway	(800) 13817
	Spain	(900) 951846
	Sweden	(0200) 895-284
	Switzerland	(0800) 111-470
	UK	(0800) 376-7903
Asia and Pacific Region	(65) 395-39-00	Singapore
	(81) (3) 3262-1743	Tokyo

**Fax**

Support Centers	Fax Numbers
North America	1-617-949-1724 (Cambridge, MA) 1-281-584-1807 (Houston, TX: both Engineering and Manufacturing Suite) 1-281-584-5442 (Houston, TX: eSupply Chain Suite) 1-281-584-4329 (Houston, TX: Advanced Control Suite) 1-301-424-4647 (Rockville, MD) 1-908-516-9550 (New Providence, NJ) 1-425-492-2388 (Seattle, WA)
South America	(54) (11) 4361-7220 (Argentina) (55) (11) 5012-4442 (Brazil)
Europe	(32) (2) 701-94-45
Asia and Pacific Region	(65) 395-39-50 (Singapore) (81) (3) 3262-1744 (Tokyo)

**E-mail**

Support Centers	E-mail
North America	support@aspentech.com (Engineering Suite) atmdsupport@aspentech.com (Aspen ICARUS products) mimi.support@aspentech.com (Aspen MIMI products) pims.support@aspentech.com (Aspen PIMS products) aspenretail.support@aspentech.com (Aspen Retail products) acs.support@aspentech.com (Advanced Control products) AMS.Support@aspentech.com (Manufacturing Suite) Gabriela.Torres@aspentech.com (Mexico)
South America	info@tecnosolution.com.ar (Argentina) tecnosp@aspentech.com (Brazil)
Europe	atesupport@aspentech.com (Engineering Suite) AMS.Support@aspentech.com (All other suites) cimview@aspentech.com (CIMVIEW products)
Asia and Pacific Region	atasupport@aspentech.com (Singapore: Engineering Suite) SG_Support@aspentech.com (Singapore: All other suites) atjsupport@aspentech.com (Tokyo: Engineering Suite) TK_Support@aspentech.com (Tokyo: All other suites)

---

# Blending Crude and Petroleum Fractions

This tutorial contains five simulations to guide you through the process of building a model of a crude fractionation train typically found in a petroleum refinery.

In this first simulation, begin building the model by defining the crude feed to the process, using the Aspen Plus Assay Data Analysis Run Type. Use this run type to analyze petroleum crudes before running a flowsheet simulation.

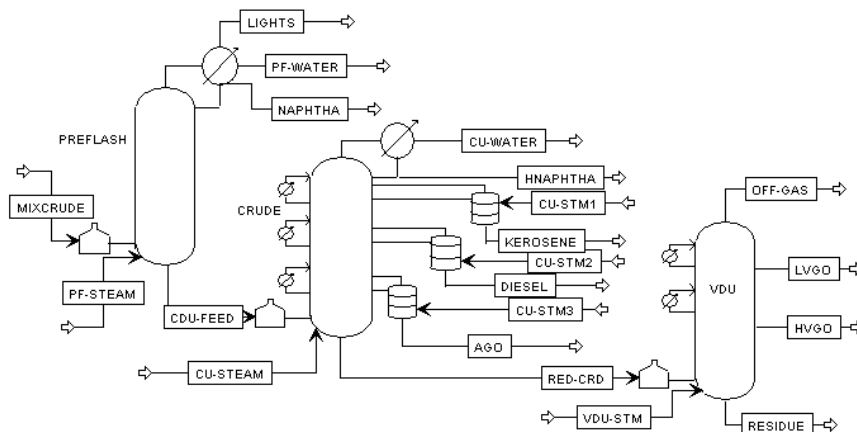
In this simulation you will:

- Define components
- Enter assay data for two crude oils
- Blend the crude oils to produce the crude feed
- Generate pseudocomponents for the blend
- Run the Assay Data Analysis calculations
- Examine results

Allow about 45 minutes to complete this simulation.

# Petroleum Process Flowsheet

The following illustration shows the petroleum process flowsheet you will work with throughout this book:



## *Problem Definition: Crude Fractionation Train*

The process consists of the following steps:

- 1 The process feed (MIXCRUDE), consisting of a blend of two crude oils (OIL-1 and OIL-2; see Tables 2.1 and 2.2), goes to the preflash furnace.
- 2 The preflash tower (PREFLASH) removes light gases and some naphtha from the partially vaporized feed.
- 3 Preflash bottoms (CDU-FEED) are further processed in the crude distillation unit (CDU). The CDU consists of a crude unit furnace and an atmospheric tower. First, the crude unit furnace partially vaporizes the bottoms from the preflash. Then the atmospheric tower separates the preflash bottoms into five cuts:
  - Heavy naphtha (HNAPHTHA)
  - Kerosene (KEROSENE)
  - Diesel (DIESEL)
  - Atmospheric gas oil (AGO)
  - Reduced crude (RED-CRD)

- 4 Reduced crude goes to the vacuum distillation unit (VDU) for further fractionation under vacuum conditions. The VDU consists of a vacuum unit furnace and vacuum tower. The vacuum tower produces the following additional cuts:
- Overhead (OFF-GAS)
  - Light vacuum gas oil (LVGO)
  - Heavy vacuum gas oil (HVGO)
  - Asphaltic residue (RESIDUE)

*Table 2.1*  
*OIL-1: (API = 31.4)*

TBP Distillation		Light Ends Analysis		API Gravity Curve	
Liq. Vol. %	Temp. (F)	Component	Liq. Vol. Frac.	Mid. Vol. %	Gravity
6.8	130.0	Methane	0.001	5.0	90.0
10.0	180.0	Ethane	0.0015	10.0	68.0
30.0	418.0	Propane	0.009	15.0	59.7
50.0	650.0	Isobutane	0.004	20.0	52.0
62.0	800.0	N-Butane	0.016	30.0	42.0
70.0	903.0	2-Methyl-Butane	0.012	40.0	35.0
76.0	1000.0	N-Pentane	0.017	45.0	32.0
90.0	1255.0			50.0	28.5
				60.0	23.0
				70.0	18.0
				80.0	13.5

Table 2.2  
OIL-2: (API = 34.8)

TBP Distillation		Light Ends Analysis		API Gravity Curve	
Liq. Vol. %	Temp. (F)	Component	Liq. Vol. Frac.	Mid. Vol. %	Gravity
6.5	120.0	Water	0.001	2.0	150.0
10.0	200.0	Methane	0.002	5.0	95.0
20.0	300.0	Ethane	0.005	10.0	65.0
30.0	400.0	Propane	0.005	20.0	45.0
40.0	470.0	Isobutane	0.01	30.0	40.0
50.0	550.0	N-Butane	0.01	40.0	38.0
60.0	650.0	2-Methyl-Butane	0.005	50.0	33.0
70.0	750.0	N-Pentane	0.025	60.0	30.0
80.0	850.0			70.0	25.0
90.0	1100.0			80.0	20.0
95.0	1300.0			90.0	15.0
98.0	1475.0			95.0	10.0
100.0	1670.0			98.0	5.0

## Starting Aspen Plus

### To Start Aspen Plus

- 1 From your desktop, select **Start** and then select **Programs**.
- 2 Select **AspenTech | Aspen Engineering Suite | Aspen Plus 11.1 | Aspen Plus User Interface**.

The **Aspen Plus Startup** dialog box appears. **Aspen Plus** displays a dialog box whenever you must enter information or make a selection before proceeding. In this simulation, use an **Aspen Plus** template.


### To Select the Template Option

- 3 Select the **Template** option and click **OK**.

The **New** dialog box appears.

Specify the Application Type and the Run Type for the new run. **Aspen Plus** uses the Application Type you choose to automatically set various defaults appropriate to your application. For this simulation, use the Petroleum with English Units template and the Assay Data Analysis Run Type.

### To Specify the Application Type and Run Type

- 4 Select the **Petroleum with English Units** template.
- 5 Click  in the Run Type field to display the available run types.



- 6 Select **Assay Data Analysis** to specify an Assay Data Analysis/PCS run.
- 7 Click **OK** to apply these options.

---

**Note:** If the **Connect to Engine** dialog box appears, see Chapter 7.

---

Wait a few seconds while Aspen Plus starts.

The Aspen Plus graphical workspace is shaded because you do not use a graphical simulation flowsheet in specifying an Assay Data Analysis run.

Use the Aspen Plus expert guidance Next function to take you to the next required input specification.

- 8 In the toolbar, click .

The **Setup | Specifications | Global** sheet appears.

## Specifying Title and Global Options

The **Setup | Specifications | Global** sheet displays the defaults Aspen Plus uses for other forms. Use this sheet to:

- Specify a title for your simulation
- Review the global settings that were set when you selected the Petroleum with English Units application type

---

**Note:** The run type Assay Data Analysis is already selected. You do not need to change this field.

---

It is always good practice to describe your simulation by entering a title for the simulation.

- 1 In the Title field, enter *Getting Started With Petroleum -- Simulation 1*.

Since you chose the Petroleum with English Units application type when you started this simulation, Aspen Plus has set the following global defaults for input specifications:

- ENGPETRO units (English Engineering units appropriate for Petroleum applications, such as volume flow in barrels per day)
- StdVol flow basis for all flow inputs
- Free-water calculations

The sheet is complete, and all the defaults set by the Petroleum with English Units application type are appropriate for this simulation:

Global Description Accounting Diagnostics

Title: Getting Started With Petroleum -- Simulation 1

Units of measurement

Input data: ENGPET

Output results: ENGPET

Global settings

Run type: Assay Data Analysis

Input mode: Steady-State

Stream class: CONVEN


Flow basis: StdVol

Ambient pressure: 14.69595 psi

Ambient temp.: 50 F

Valid phases:

☒ Use free water calculations

- 2 Click  to continue.

The **Components | Specifications | Selection** sheet appears.


## Specifying Components

Use the **Components Specifications Selection** sheet to specify components for the simulation.

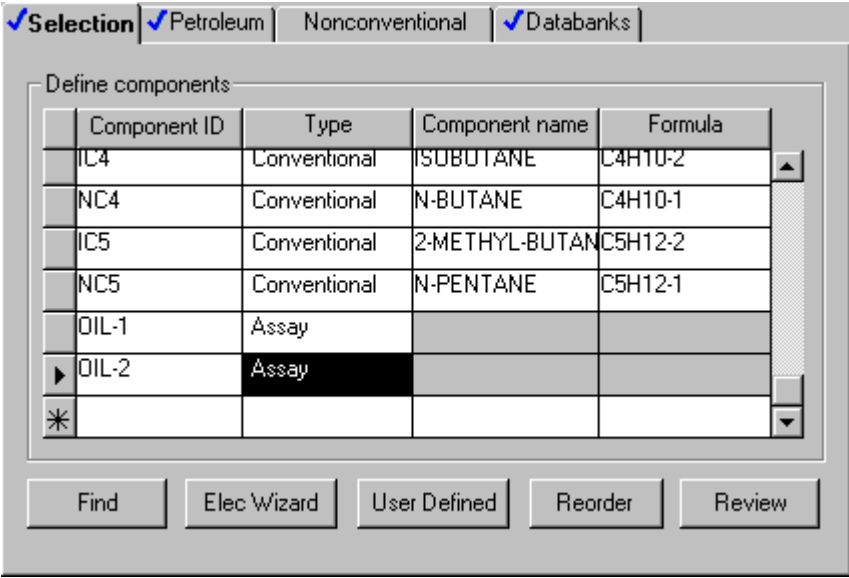
- 1 Enter the following component IDs and component names:

Component ID	Component Name
H2O	WATER
C1	METHANE
C2	ETHANE
C3	PROPANE
IC4	ISOBUTANE
NC4	N-BUTANE
IC5	2-METHYL-BUTANE
NC5	N-PENTANE


You will also define two crude assays, OIL-1 and OIL-2.

- 2 In the next Component ID field, enter OIL-1.
- 3 In the Type field for OIL-1, click  and select Assay.
- 4 In the next Component ID field, enter OIL-2 and identify it as an Assay, in the same way as you did for OIL-1.

The **Components Specifications** sheet is complete:



Component ID	Type	Component name	Formula
IC4	Conventional	ISOBUTANE	C4H10-2
NC4	Conventional	N-BUTANE	C4H10-1
IC5	Conventional	2-METHYL-BUTANE	C5H12-2
NC5	Conventional	N-PENTANE	C5H12-1
OIL-1	Assay		
OIL-2	Assay		
*			


- 5 Click  to continue.

The **Components | Assay/Blend | OIL-1 | Basic Data | Dist Curve** sheet appears.

## Entering Assay Data

On the **Dist Curve** sheet, enter the laboratory assay data for OIL-1 (Table 2.1).

### Specify the Distillation Curve for OIL-1

- 1 In the **Distillation curve type** field, click  and select *True boiling point (liquid volume basis)*.

---

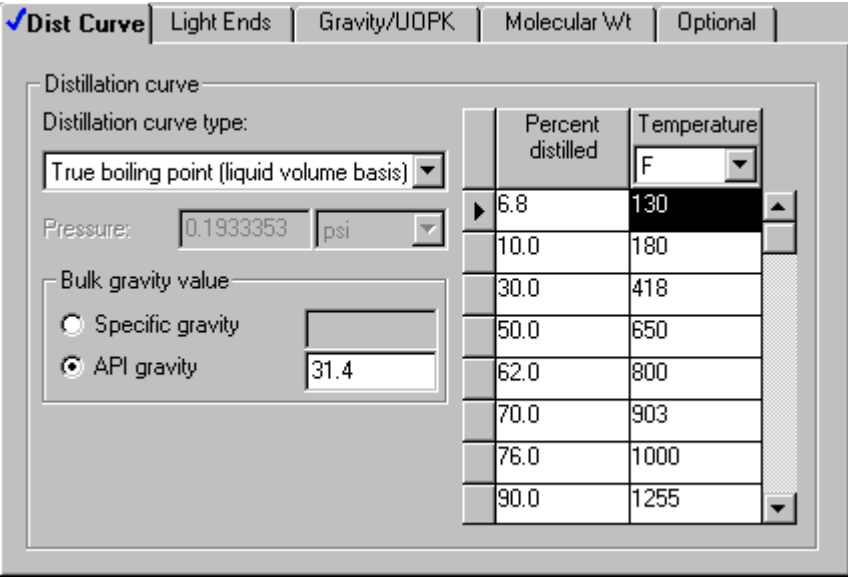
**Note:** Select True boiling point to indicate that you are entering data for a TBP curve on a standard liquid volume basis.

---

- 2 In the **Bulk gravity value** area, select the *API gravity* option.
- 3 In the **API gravity** field, enter 31.4.
- 4 In the **Percent Distilled** and **Temperature** columns, enter the following TBP distillation values for OIL-1:

Percent Distilled	Temperature F
6.8	130
10.0	180
30.0	418
50.0	650
62.0	800
70.0	903
76.0	1000
90.0	1255

The **Components | Assay/Blend | OIL-1 | Basic Data | Dist Curve** sheet is complete:



Distillation curve

Distillation curve type:  
 True boiling point (liquid volume basis) ▼

Pressure: 0.1933353 psi ▼

Bulk gravity value  
☐ Specific gravity  
☒ API gravity 31.4

Percent distilled	Temperature
6.8	130
10.0	180
30.0	418
50.0	650
62.0	800
70.0	903
76.0	1000
90.0	1255

## Specify the Light Ends Data for OIL-1

- 5 Select the **Light Ends** sheet.  
Use this sheet to enter light ends analysis data, given as a fraction of the assay mixture.
- 6 In the **Component** and **Fraction** (StdVol) fields, enter the following light ends analysis data:

Component	Fraction StdVol
C1	0.001
C2	0.0015
C3	0.009
IC4	0.004
NC4	0.016
IC5	0.012
NC5	0.017

The **OIL-1 | Basic Data | Light Ends** sheet is complete:

Light ends fraction:

Light ends analysis

Component	Fraction StdVol	Gravity	Molecular weight
C1	0.001		
C2	0.0015		
C3	0.009		
IC4	0.004		
NC4	0.016		
IC5	0.012		
NC5	0.017		

## Specify the API Gravity Data for OIL-1

- 7 Select the **Gravity/UOPK** sheet.  
The **OIL-1 | Basic Data | Gravity/UOPK** sheet appears.
- 8 In the **Type** field, select the *API gravity* option.

- 9 In the **Mid percent distilled** and **API gravity** fields, enter the following API gravity curve data:

Mid Percent Distilled	API Gravity
5.0	90.0
10.0	68.0
15.0	59.7
20.0	52.0
30.0	42.0
40.0	35.0
45.0	32.0
50.0	28.5
60.0	23.0
70.0	18.0
80.0	13.5

The **OIL-1 | Basic Data | Gravity/UOPK** sheet is complete:

☒ Dist Curve  
 ☒ Light Ends  
 ☒ Gravity/UOPK  
 Molecular Wt  
 Optional

Type

☐ Specific gravity  
 ☒ API gravity  
 ☐ UOPK

API gravity curve data

Bulk value: 31.4

	Mid percent distilled	API gravity
	50.0	28.5
	60.0	23
	70.0	18
	80.0	13.5
*		

- 10 Click to continue.

The **OIL-2 | Basic Data | Dist Curve** sheet appears.

### Specify the Distillation Curve for OIL-2

- 1 In the **Distillation Curve Type** field, click and select *True boiling point (liquid volume basis)*.
- 2 In the **Bulk gravity value** field, select the *API gravity* option.
- 3 In the **API gravity** field, enter 34.8.

- 4 In the **Percent distilled** and **Temperature** fields, enter the following TBP distillation values:

Percent Distilled	Temperature F
6.5	120.0
10.0	200.0
20.0	300.0
30.0	400.0
40.0	470.0
50.0	550.0
60.0	650.0
70.0	750.0
80.0	850.0
90.0	1100.0
95.0	1300.0
98.0	1475.0
100.0	1670.0

The **OIL-2 | Basic Data | Dist Curve** sheet is complete.

**Specify the Light  
Ends Data for OIL-2**

- 5 Select the **Light Ends** sheet.

The **OIL-2 | Basic Data | Light Ends** sheet appears.

- 6 In the **Component** and **Fraction (StdVol)** fields, enter the following light ends analysis data:

Component	Fraction StdVol
H2O	0.001
C1	0.002
C2	0.005
C3	0.005
IC4	0.01
NC4	0.01
IC5	0.005
NC5	0.025

The **OIL-2 | Basic Data | Light Ends** sheet is complete.

**Specify the API  
Gravity Data for OIL-2**

- 7 Select the **Gravity/UOPK** sheet.


The **OIL-2 | Basic Data | Gravity/UOPK** sheet appears.

- 8 In the **Type** field, select the *API gravity* option.

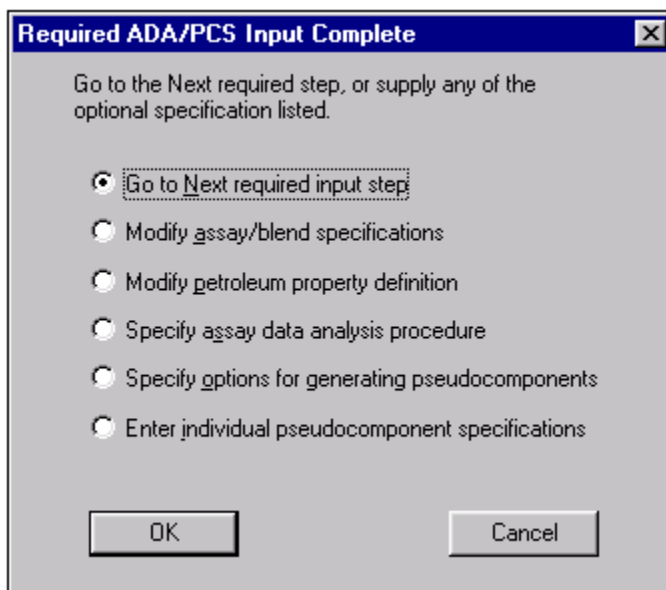
- 9 In the **Mid percent distilled** and **API gravity** fields, enter the following API gravity curve data:

Mid percent Distilled	API gravity
2.0	150.0
5.0	95.0
10.0	65.0
20.0	45.0
30.0	40.0
40.0	38.0
50.0	33.0
60.0	30.0
70.0	25.0
80.0	20.0
90.0	15.0
95.0	10.0
98.0	5.0

The **OIL-2 | Basic Data | Gravity/UOPK** sheet is complete.

- 10 Click  to continue.

The **Required ADA/PCS Input Complete** dialog box appears.





## Blending the Oils


- 1 On the **Required ADA/PCS Input Complete** dialog box, select *Modify assay/blend specifications* and click **OK**.

The **Components | Assay/Blend** object manager appears.

- 2 Click **New**.


The **Create new ID** dialog box appears.

- 3 In the **Enter ID** field, enter MIXOIL.

- 4 In the **Select type** field, click  and select BLEND.

- 5 Click **OK**.

The **Components | Assay/Blend | MIXOIL | Mixture | Specifications** sheet appears. Use this sheet to define the fractions for each crude oil in the blend.

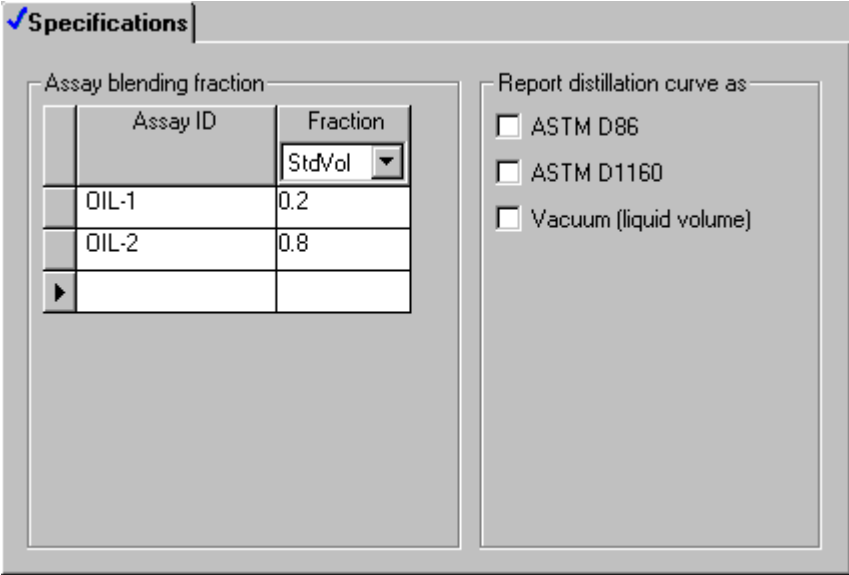
- 6 In the **Assay ID** column, click  and select OIL-1.

- 7 In the **Fraction (StdVol)** field, enter 0.20.

- 8 In the next row of the **Assay ID** column, click  and select OIL-2.

- 9 In the **Fraction (StdVol)** field, enter 0.80.

The **MIXOIL | Mixture | Specifications** sheet is complete:



**Specifications**

Assay blending fraction


Assay ID	Fraction
OIL-1	0.2
OIL-2	0.8

Report distillation curve as

☐ ASTM D86

☐ ASTM D1160

☐ Vacuum (liquid volume)

- 10 Click  to continue.

The **Required ADA/PCS Input Complete** dialog box appears.

## Generating Pseudocomponents

By default, Aspen Plus generates a set of pseudocomponents for your simulation based on all the assays and blends in your simulation. However, in this simulation, the pseudocomponents should be based on the blend only.

- 1 From the **Required ADA/PCS Input Complete** dialog box, select *Specify options for generating pseudocomponents* and click **OK**.

The **Petro Characterization | Generation** object manager appears.

- 2 Click **New**.

The **Create new ID** dialog box appears.

- 3 In the **Enter ID** field, enter CRUDE and click **OK**.

The **Petro Characterization | Generation | CRUDE | Specifications** sheet appears.


- 4 In the **Assay/Blend ID** field, click  and select MIXOIL.

---

**Note:** The **Weighting factor** field automatically updates with the proper value based on the Assay/Blend ID you selected.

---


The **Specifications** sheet is complete.



	Assay/Blend ID	Weighting factor
▶	MIXOIL	1
*		

Pseudocomponent property options

Property method: ASPEN

- 5 Click  to continue.

The **Required ADA/PCS Input Complete** dialog box appears.

- 6 From the **Required ADA/PCS Input Complete** dialog box, select the default, *Go to Next required input step* and click **OK**.

The **Required Properties Input Complete** dialog box appears.

- 7 Click **OK**.

The **Required Assay Data Analysis/Pseudocomponent Generation Input Complete** dialog box appears.

## Running the Simulation

You have entered the data and specifications for this simulation, and can now run the Assay Data Analysis.

- 1 In the **Required Assay Data Analysis/Pseudocomponent Generation Input Complete** dialog box, click **OK**.

The **Control Panel** window appears. The Control Panel allows you to monitor and interact with the Aspen Plus simulation calculations.

As Aspen Plus performs the analysis, status messages display in the Control Panel. The message *Results Available* appears in the status bar at the bottom of the main window.

- 2 Examine the results of your analysis.

## Examining Results

An ADA run calculates a wide variety of properties for the assays, blends, and generated pseudocomponents.

For this simulation, review pseudocomponent properties and TBP curves for the oils and the blend.

- 1 From the Control Panel, click .

The **Results Summary | Run Status | Summary** sheet appears. It indicates that calculations were completed normally. The main properties for each pseudocomponent appear on the **Components | Petro Characterization | Results | Summary** sheet.

- 2 Use the Data Browser menu tree to navigate to the **Components | Petro Characterization | Results** form. The **Summary** sheet appears.

- 3 Use the horizontal and vertical scrollbars at the bottom and right of the sheet to scroll through the component properties.

Summary		Petro Properties	Viscosity	
Pseudocomponents	Normal BP	API Gravity	Specific gravity	Molecular weight
	F			
▶ PC120F	120.089471	81.8332787	0.66328142	77.091571
PC138F	137.70481	77.4601986	0.67716245	81.196339
PC163F	162.680744	72.0614456	0.69512181	87.235420
PC188F	187.729121	67.2545235	0.71193348	93.528022
PC213F	212.976267	62.4253273	0.7296623	103.32907
PC238F	238.262494	57.3528678	0.74926053	109.02837
PC263F	263.191275	51.9331906	0.77139802	114.46458

## Plotting Distillation Curves

Plot the TBP curves for the blend and each crude oil in a single plot. The results for each assay or blend appear on the **Components | Assay/Blend | assay/blend name | Results** sheet.

### To Locate the Curve Data for OIL-1

- 1 Use the Data Browser menu tree to navigate to the **Components | Assay/Blend | OIL-1 | Results** form. The **Light Ends Analysis** sheet appears.

Light Ends Analysis		Pseudocomp Breakdown	Curves
Specific gravity:	0.58213152	Density:	36.2497261 lb/cuft
API gravity:	111.572219	Molecular wt:	59.4808141
Component	Volume percent of assay	Weight percent of assay	Mole fraction
C1	0.1	0.03457145	0.03161354
C2	0.15	0.06145202	0.02998087
C3	0.9	0.5256157	0.17486383
IC4	0.4	0.2591946	0.06542002
NC4	1.6	1.07641199	0.27168351
Total	6.05	4.05453574	1

- 2 Select the **Curves** sheet.

The **Curves** sheet for OIL-1 appears.

Percent distilled	True boiling pt (liquid volume)
	F
Pres: PSI	14.6959488
0	-46.525138
5	94.7012928
10	180
30	418

### To Plot the TBP Curve

- 3 Select the **Percent distilled** column.
- 4 From the Aspen Plus menu bar, select **Plot | X-Axis Variable**.
- 5 Select the **True boiling pt. (liquid volume)** column.
- 6 From the Aspen Plus menu bar, select **Plot | Y-Axis Variable**.
- 7 From the Aspen Plus menu bar, select **Plot | Display Plot**.

The **Components Assay/Blend Oil-1 Results Curves - Plot** window appears.

### To Label This Curve for OIL-1

- 8 Double-click the legend box.  
The **Plot Legend Box** dialog box appears.
- 9 Select *True boiling pt (vol)*. True boiling pt (vol) appears in the **Legend Text** field.
- 10 In the **Legend Text** field, delete *True boiling pt (vol)* and enter *TBP for OIL-1*.
- 11 Click **Replace**, then click **OK**.

## To Plot the TBP Curve for OIL-2

Now that you have plotted and labeled one curve, you can add the other curves to the plot.

- 1 From the Data Browser, open the **OIL-2 | Results** form, and select the **Curves** sheet.

The **Curves** sheet for OIL-2 appears.

Percent distilled	True boiling pt (liquid volume)
	F
Pres: PSI	14.6959488
0	-93.585454
5	78.8137355
10	200
30	400

## To Add This Curve to the Plot Containing the Curve for OIL-1

- 2 Select the **Percent distilled** column.
- 3 From the Aspen Plus menu bar, select **Plot | X-Axis Variable**.
- 4 Select the **True boiling pt. (liquid volume)** column.
- 5 From the Aspen Plus menu bar, select **Plot | Y-Axis Variable**.
- 6 From the Aspen Plus menu bar, select **Plot | Add New Curve**. The **Plot Window List** dialog box appears.
- 7 Select *Components Assay/Blend OIL-1 Results Curves*, and click **OK**.

The OIL-2 curve appears on the plot.

## To Label This Curve for OIL-2

- 8 Double-click the legend box. The **Plot Legend Box** dialog box appears.
- 9 Select *True boiling pt (vol)*. True boiling pt (vol) appears in the **Legend Text** field.
- 10 In the **Legend Text** field, delete *True boiling pt (vol)* and enter *TBP for OIL-2*.
- 11 Click **Replace**, then click **OK**.

## To Plot the BLEND Curve

- 1 In the Data Browser, open the **MIXOIL | Results** form, and select the **Curves** sheet.

The **MIXOIL | Results | Curves** sheet appears.

Percent distilled	True boiling pt (liquid volume)
0	-93.585454
5	82.6984804
10	194.999523
30	402.747953

## To Add This Curve to the Plot

- 2 Select the **Percent distilled** column.
- 3 From the Aspen Plus menu bar, select **Plot | X-Axis Variable**.
- 4 Select the **True boiling pt. (liquid volume)** column.
- 5 From the Aspen Plus menu bar, select **Plot | Y-Axis Variable**.
- 6 From the Aspen Plus menu bar, select **Plot | Add New Curve**.  
The **Plot Window List** dialog box appears.

- 7 Select *Components Assay/Blend OIL-1 Results Curves*, and click **OK**.

The MIXOIL curve appears on the plot.

- 8 Double-click the legend box.  
The **Plot Legend Box** dialog box appears.

## To Label This Curve for MIXOIL

- 9 Select *True boiling pt (vol)*.
- 10 In the **Legend Text** field, delete *True boiling pt (vol)* and enter *TBP for Blend*.
- 11 Click **Replace**, then click **OK**.

Now you have all three curves in one plot window, but there are some things you can do to make the plot look nicer.

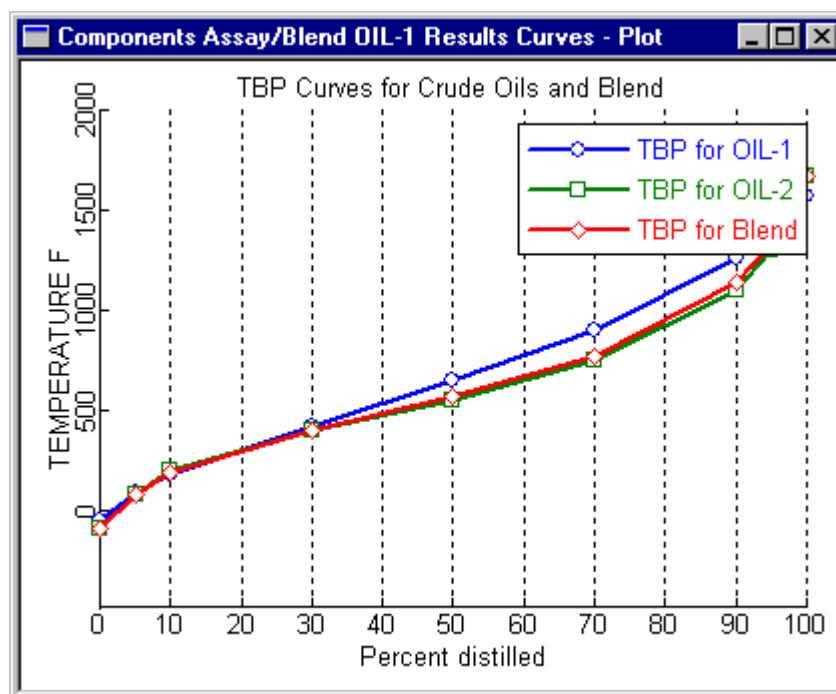
### To Combine Y-Axes

- 1 Double-click on the plot.  
The **PfsPlot 11.1 Control Properties** dialog box appears.
- 2 Select the **AxisMap** sheet.
- 3 Click **All in One** and then click **OK**.

### To Rename the Plot

- 4 Double-click the *Components Assay/Blend OIL-1 Results Curves* title.  
The **Plot Text Setting** dialog box appears.
- 5 In the Title text field, delete the text and enter *TBP Curves for Crude Oils and Blend*.
- 6 Click **OK**.


As the completed plot shows, the TBP curve for the blend is between the curves for each oil. Curve locations depend on the proportions of the two oils you define for the blend:





## Creating a Backup File

Once satisfied with your Assay Data Analysis results, save this simulation as a backup file. This allows you to use the developed assay data analysis in another Aspen Plus simulation.

- 1 From the Aspen Plus menu bar, select **File | Save As**.  
The **Save As** dialog box appears.
- 2 In the **Save as type** field, click  and select *Aspen Plus Backup Files (\*.bkp)*.
- 3 In the **File name** field, enter BLEND and click **Save**.

---

**Note:** Your working directory appears in the **Save in** field.

---

The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.

- 4 Click **No**.

## Exiting Aspen Plus

- 1 From the **Aspen Plus** menu bar, select **File | Exit**.  
The **Aspen Plus** dialog box appears.
- 2 Click **Yes** to save the simulation.  
The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.
- 3 Click **No**.

---

**Note:** The Chapter 3 simulation uses this run as the starting point.

---



---

# Adding a Preflash Tower

In this simulation, add a preflash tower to the simulation and specify options for pseudocomponent generation.

You will:

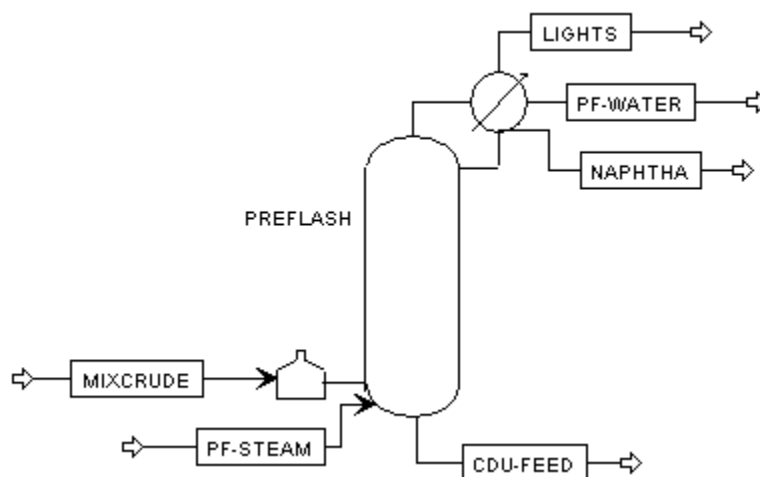
- Define the flowsheet graphically
- Specify properties, feed streams, and preflash tower
- Modify cuts specifications for pseudocomponents generation
- Run the simulation
- Examine simulation results

Allow about 45 minutes to complete this simulation.

## Preflash Tower Flowsheet

Figure 3.1 shows the process flowsheet for this simulation. The process feed, consisting of the oil blend, goes first to the preflash furnace where it is partially vaporized. The partially vaporized feed then enters the preflash tower. Model the tower and the furnace simultaneously with a single PetroFrac block.

Steam feeds to the bottom of the tower. The tower produces a wide naphtha cut as a distillate product.



**Figure 3.1** Flowsheet for Preflash Tower

Simulate the tower with 10 theoretical stages, no reboiler, and a partial condenser. The condenser operates at 170 F and 39.7 psia, with a pressure drop of 2 psi. The tower pressure drop is 3 psi.

The tower is stripped with open steam in the bottom. The steam stream is at 400 F and 60 psia, and has a flow rate of 5,000 lb/hr. The furnace operates at a pressure of 50 psia and a temperature of 450 F. The distillate rate is estimated at 15,000 bbl/day. Its value is manipulated to produce a wide naphtha cut with an ASTM 95% temperature of 375 F.

## Starting Aspen Plus

- 1 From your desktop, select **Start | Programs**.
- 2 Select **AspenTech | Aspen Engineering Suite | Aspen Plus 11.1 | Aspen Plus User Interface**.

The **Aspen Plus Startup** dialog box appears. **Aspen Plus** displays a dialog box whenever you must enter information or make a selection before proceeding. In this simulation, use an **Aspen Plus** template.

## Opening an Existing Run


### Opening the Assay Data Analysis Simulation Created in Chapter 2

- 1 In the **Aspen Plus Startup** dialog box, select the **Open an Existing Simulation** option.
- 2 In the list, select **blend.bkp** and click **OK**.

### If Your Saved File **blend.bkp** Does Not Appear

- 1 In the list on the **Aspen Plus Startup** dialog box, double-click **More Files**.  
The **Open** dialog box appears.
- 2 Navigate to the directory containing your saved **blend.bkp** file.
- 3 Select **blend.bkp** from the list of files, and click **Open**.

### If You Did Not Create **blend.bkp**

- If you did not create the assay data analysis simulation in Chapter 2, open the backup file **blend.bkp** in the **Examples** folder.
- 1 In the list on the **Aspen Plus Startup** dialog box, double-click **More Files**.  
The **Open** dialog box appears.
  - 2 Click .

By default, the Favorites list contains five folders that are provided with Aspen Plus.

  - 3 Double-click the **Examples** folder.
  - 4 Select the **blend.bkp** file, and click **Open**.

---


**Note:** If the **Connect to Engine** dialog box appears, see Chapter 7.

---

The **Aspen Plus** window appears.


## Saving a Run Under a New Name

Before you create a new run adding a preflash tower, create and save a copy of BLEND with a new Run ID, PREFLASH. Then make modifications under this new Run ID.

- 1 From the Aspen Plus menu bar, select **File | Save As**.  
The **Save As** dialog box appears.
- 2 Select the directory where you want to save the simulation.
- 3 In the **File name** field, enter PREFLASH.
- 4 In the **Save as type** field, click  and select Aspen Plus Backup Files (\*.bkp).
- 5 Click **Save** to save the simulation and continue.  
The **Aspen Plus** dialog box appears, asking if you want to save the simulation as an Aspen Plus Document file.
- 6 Click **No**.  
The current Run ID, PREFLASH, appears in the title bar.

## Changing the Run Type

In Chapter 2, the BLEND run you performed was an Assay Data Analysis run. In this simulation, perform a flowsheet simulation. First, you need to switch to the Flowsheet run type.

- 1 In the Data Browser, open the **Setup | Specifications | Global** sheet.
- 2 In the **Run type** field, click  and select *Flowsheet*.
- 3 In the **Title** field, enter *Getting Started With Petroleum -- Simulation 2*.

The graphics workspace is now active and ready for you to build the flowsheet. No more changes to this sheet are necessary.

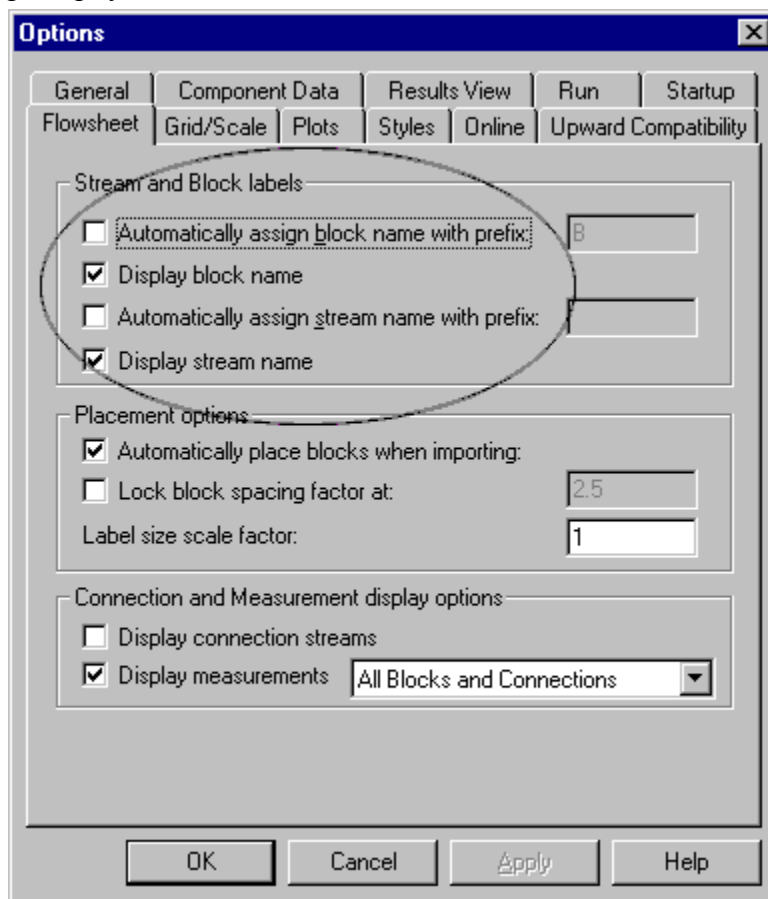
## Turning Off Automatic Naming of Blocks and Streams

Since you will enter your own block and stream IDs, turn off the default options to create block and stream IDs automatically.

- 1 From the **Aspen Plus** menu bar, select **Tools | Options**.  
The **Options** dialog box appears.

- 2 Select the **Flowsheet** tab.
- 3 Clear the checkboxes for the **Automatically assign block name with prefix** and the **Automatically assign stream name with prefix** options.

Since you have deselected these options, Aspen Plus will prompt you for block and stream names.



- 4 Click **OK** to apply these changes.

You are ready to build your flowsheet. Figure 3.1 shows the process flow diagram for this simulation. To review how to define a flowsheet graphically, see *Getting Started Building and Running a Process Model*, Chapter 3.

## Selecting a Distillation Model

Aspen Plus offers several distillation models. To determine which is appropriate for the preflash tower, use prompts and online help.

- 1 Close the Data Browser. You are now in the **Process Flowsheet** window.
- 2 From the **Model Library**, select the **Columns** tab.

The list of available distillation columns appears.

- 3 Select different models within the Model Library and read the descriptions in the Status bar.

From the prompts, it appears that PetroFrac is appropriate for this petroleum problem. Obtain more extensive information on PetroFrac by reading the online help:

- 4 Select the PetroFrac model.
- 5 Press **F1** on the keyboard.

The **Aspen Plus Help** window appears, with detailed information about the PetroFrac model.

- 6 Review this information to confirm that PetroFrac is the best choice, then close the help window.


## Defining the Graphical Simulation Flowsheet

Now you will begin to build the process flowsheet by adding the preflash distillation unit.

- 1 From the Model Library, select the **Columns** tab.
- 2 From the **Columns** tab, select PetroFrac (the unit operation model to place in your process flowsheet).



**To Choose a Different Icon for the Model**

- 3 Click  next to the PetroFrac block icon to display all the PetroFrac model icons.
- 4 Move the mouse over the icons to see a name (or label) for each.
- 5 Locate the PetroFrac icon PREFL1F in the first row of icons, and select it.
- 6 From the **Columns** tab, select the PetroFrac icon and drag it to the **Process Flowsheet** window.
- 7 Release the icon in the desired location within the flowsheet. The **Input** dialog box appears.
- 8 In the **Enter the Block ID** field, enter PREFLASH and click **OK**.

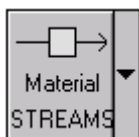
**Place the Preflash Tower on the Flowsheet**



## Place the Streams on the Flowsheet


Connect and name the streams, as shown in Figure 3.1. Use the port names shown below to connect to the appropriate port:

Stream	Port
MIXCRUDE	Main Column Feed
PF-STEAM	Main Column Feed
LIGHTS	Vapor Distillate from Main Column
PF-WATER	Condenser Water Decant for Main Column
NAPHTHA	Liquid Distillate from Main Column
CDU-FEED	Bottoms Product from Main Column



- 9 Click .

When you move the mouse to the flowsheet, the stream ports appear.



- 10 Select the **Main Column Feed** port and select it to create a new stream.
- 11 Point to a blank part of the window where you want the feed to originate and click.
- The **Input** dialog box appears.
- 12 In the **Enter the Stream ID** field, enter PF-STEAM and click **OK**.
- 13 For the MIXCRUDE stream, click the feed port and then drag the mouse to the desired location in front of the feed furnace before releasing the button.
- 14 Create the remaining streams in similar fashion.
- 15 When all streams are complete, click  to continue.
- The **Flowsheet Complete** dialog box appears.
- 16 Click **OK**.

The **Properties | Specifications | Global** sheet appears.

## Specifying Properties

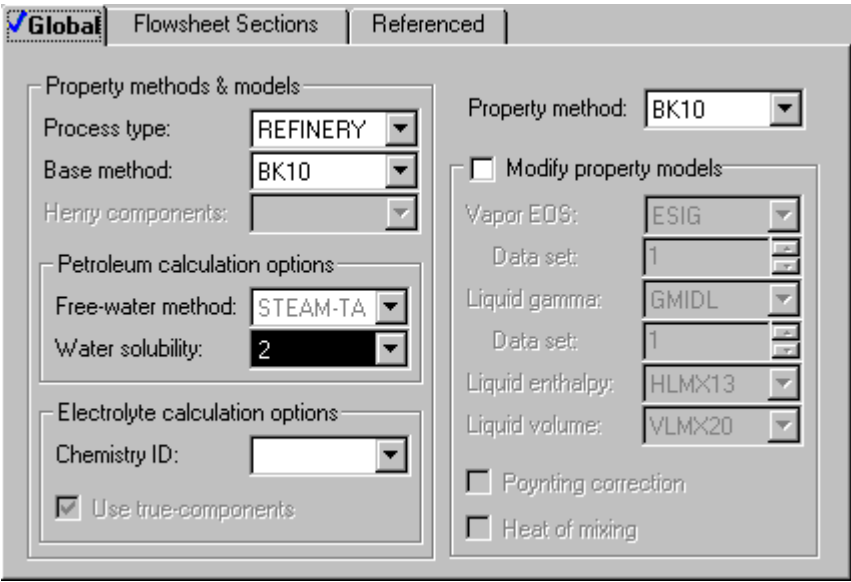
Use the **Properties | Specifications | Global** sheet to select the thermodynamic methods used to calculate properties such as K-values, enthalpy, and density. Property methods in Aspen Plus are arranged according to process types. The BK10 property method (Braun K-10 method) is appropriate for most refining applications involving heavy petroleum fractions and low pressures.

## To Select the BK10 Property Method


- 1 In the **Process type** field, click  and select REFINERY.
- 2 In the **Base method** field,  and select BK10.

Since you selected the Petroleum with English Units application type in Chapter 2, Aspen Plus selected the appropriate defaults for petroleum applications. Steam Tables (STEAM-TA) is the default method for calculating free-water phase properties, as shown in the **Free-water method** field. Aspen Plus also selected Method 2 for the water solubility default, as shown in the **Water solubility** field. The Water solubility option controls the method for calculating the solubility and K-value of water in the hydrocarbon phase.

You do not need to change anything else on this sheet.



The screenshot shows the 'Property methods & models' dialog box in Aspen Plus. The 'Process type' is set to 'REFINERY'. The 'Base method' is set to 'BK10'. The 'Free-water method' is set to 'STEAM-TA'. The 'Water solubility' is set to '2'. The 'Vapor EOS' is set to 'ESIG'. The 'Liquid gamma' is set to 'GM1L'. The 'Liquid enthalpy' is set to 'HLMX13'. The 'Liquid volume' is set to 'VLMX20'. The 'Use true-components' checkbox is checked. The 'Modify property models' checkbox is unchecked.

- 3 Click  to continue.

The **Required Properties Input Complete** dialog box appears. Correct representation of physical properties is essential to process modeling. For many simulations, the only physical property specification you must provide is to select an option set. The **Required Properties Input Complete** dialog box is a reminder that the Aspen Plus physical property system has many optional capabilities that you can use to increase the accuracy of physical property calculations.

- 4 Click **OK**.

The **Streams | MIXCRUDE | Input | Specifications** sheet appears.

## Entering Stream Data

MIXCRUDE is the crude oil blend feed stream to the tower. Enter condition and flow rates for the MIXCRUDE stream.

- 1 In the **Temperature** field, enter 200.  
The default units F (Fahrenheit) are correct for this simulation.
- 2 In the **Pressure** field, enter 60.  
The default units (psi) are correct for this simulation.
- 3 In the **Composition** section, use the scrollbar to access the MIXOIL component.
- 4 In the **Value** column of the MIXOIL component, enter 100000.  
The **Streams | MIXCRUDE | Input** form is complete:

Substream name: **MIXED** Ref Temperature

State variables:

Temperature: 200 F

Pressure: 60 psi

Total flow: Stdvol bbl/day

Solvent:

Composition:

Component	Value
C2	
C3	
IC4	
NC4	
IC5	
NC5	
MIXOIL	100000
<b>Total:</b>	<b>100000</b>


- 5 Click to continue.  
The **PF-STEAM | Input | Specifications** sheet appears. PF-STEAM is the steam feed to the tower.
- 6 In the **Temperature** field, enter 400 F.
- 7 In the **Pressure** field, enter 60 psi.
- 8 In the **Composition** section, click and select Mass-Flow.
- 9 In the **Value** column of the H2O component, enter 5000 lb/hr.  
The **PF-STEAM | Input | Specifications** sheet is complete.
- 10 Click to continue.

## Specifying the Preflash Unit

Use the **Blocks | PREFLASH | Setup | Configuration** sheet to enter configuration and operating specifications for the column, including:

- Number of theoretical stages
- Valid phases in the column
- Distillate vapor fraction

### Specify Column Configuration

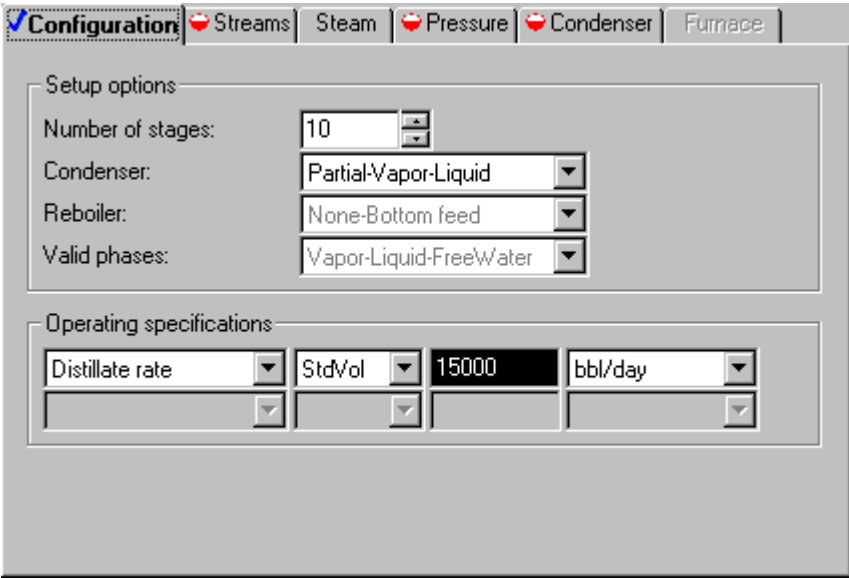
- 1 In the **Number of stages** field, enter 10 for ten theoretical stages.
- 2 In the **Condenser** field, click  and select *Partial-Vapor-Liquid*. This indicates that there is a partial condenser with both vapor and liquid distillate.

In the **Reboiler** field, the default *None-Bottom feed* option is appropriate, since the column has no reboiler and there is a bottom steam feed.

- 3 In the **Distillate rate** field, enter 15000 to specify an estimate for the distillate rate of 15,000 bbl/day.


The distillate rate will be manipulated to achieve a specification on the ASTM 95% temperature for the naphtha product.

The **PREFLASH | Setup | Configuration** sheet is complete:



Configuration sheet showing setup options and operating specifications:



- Setup options:**
  - Number of stages: 10
  - Condenser: Partial-Vapor-Liquid
  - Reboiler: None-Bottom feed
  - Valid phases: Vapor-Liquid-FreeWater
- Operating specifications:**
  - Distillate rate: 15000 bbl/day

- 4 Click  to continue.

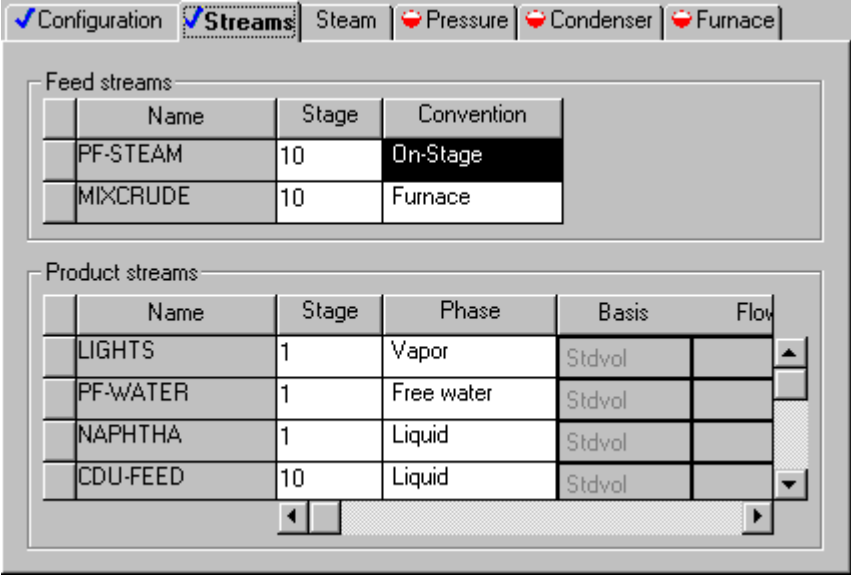
## Specify Stream Locations

The **PREFLASH | Setup | Streams** sheet appears.

Enter feed locations and conventions for the PF-STEAM and MIXCRUDE streams.

- 5 In the **Stage** column for PF-STEAM, enter 10.
- 6 In the **Convention** column for PF-STEAM, click  and select On-Stage.
- 7 In the **Stage** column for MIXCRUDE, enter 10.
- 8 In the **Convention** column for MIXCRUDE, click  and select *Furnace* to indicate that a furnace is attached to the tower on stage 10. The feed goes to the furnace first.

The **PREFLASH | Setup | Streams** sheet is complete:




**Feed streams**

Name	Stage	Convention
PF-STEAM	10	On-Stage
MIXCRUDE	10	Furnace

**Product streams**

Name	Stage	Phase	Basis	Flow
LIGHTS	1	Vapor	Stdvol	
PF-WATER	1	Free water	Stdvol	
NAPHTHA	1	Liquid	Stdvol	
CDU-FEED	10	Liquid	Stdvol	

- 9 Click  to continue.


## Specify Column Pressure Profile

The **PREFLASH | Setup | Pressure** sheet appears. On this sheet, enter the column pressure profile, or specify a top stage pressure and a pressure drop or the rest of the column.

- 10 In the **View** field, click  and select *Top/Bottom*.
- 11 Enter the following pressure profile data:

<b>Top Stage / Condenser Pressure</b>	39.7 psi
<b>Stage 2 Pressure</b>	41.7 psi
<b>Bottom Stage Pressure</b>	44.7 psi

The **PREFLASH Setup Pressure** sheet is complete.

- 12 Click  to continue.

## Specify Condenser Temperature

The **PREFLASH | Setup | Condenser** sheet appears.

13 Select the **Temperature** option.

14 In the **Temperature** field, enter a value of 170 F.

The **PREFLASH | Setup | Condenser** sheet is complete.

15 Click  to continue.

## Specify Furnace Configuration

The **PREFLASH | Setup | Furnace** sheet appears.

16 Select the **Single stage flash** option to specify the furnace type.

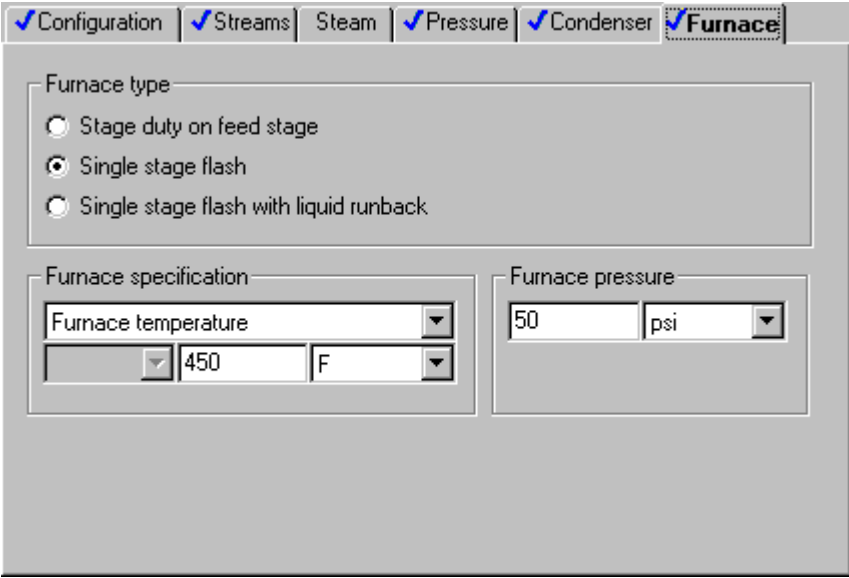
17 In the **Furnace specification** field, click  and select *Furnace temperature*.

18 In the **Furnace temperature** field, enter 450 F.

19 In the **Furnace pressure** field, enter 50 psi.

These specifications model the furnace as a single-stage flash at the specified conditions.

The **PREFLASH | Setup | Furnace** sheet is complete:



20 Click  to continue.

The **Required Input Complete** dialog box appears.

21 Click **Cancel**.






## Create Design Specification

Next, you need to set up a design specification for the product quality of the naphtha stream, using the ASTM 95% temperature. Manipulate the distillate rate to achieve this target.

1 From the Data Browser, select the **PREFLASH | Design Specs** folder.


The **Design Specs** object manager appears.

- 2 Click **New**.  
The **Create new ID** dialog box appears.
- 3 In the **Create new ID** dialog box, click **OK** to accept the default ID of 1.  
The **Design Specs | 1 | Specifications** sheet appears.
- 4 Enter the following information for the naphtha product design specification:
 

<b>Type</b>	ASTM D86 temperature (dry, liquid volume basis)
<b>Target</b>	375 F
<b>Liquid %</b>	95
- 5 Click  to continue.  
The **Design Specs | 1 | Feed/Product Streams** sheet appears.
- 6 From the **Available streams** column, select the NAPHTHA stream and click  to move the stream to the **Selected stream** column.
- 7 Click  to continue.  
The **Design Specs | 1 | Vary** sheet appears.
- 8 In the **Type** field, click  and select *Distillate flow rate*.
- 9 Click  to continue.  
The **Required Input Complete** dialog box appears.
- 10 Click **Cancel**.

## Modifying Pseudocomponents

In Chapter 2, Aspen Plus generated all pseudocomponents using default options. In this simulation, modify the number of pseudocomponents generated for different temperature ranges.

- 1 From the Data Browser, select the **Components | Petro Characterization | Generation** folder.  
The **Generation** object manager appears.
- 2 Select the CRUDE line and then click **Edit**.  
The **CRUDE | Specifications** sheet appears.
- 3 Select the **Cuts** sheet.
- 4 In the **Option for cut specifications** field, click  and select *Range and increments*.

- Enter the following temperature ranges and increments to generate pseudocomponents for this problem. Note that **Lower temperature** is filled in automatically for rows beyond the first.

Lower temperature	Upper temperature	Increment
100	800	25
800	1200	50
1200	1400	100
1400	1640	120

The **CRUDE** | **Cuts** sheet is complete:

Specifications **Cuts** Naming Options

Pseudocomponent options:

Option for cut specifications: Range and increments

Pseudocomponent generation option: F

Range and increment pseudocomponent data:

	Lower temperature	Upper temperature	Number of cuts	Increment
	F	F		F
	800	1200		50
	1200	1400		100
	1400	1640		120
*	1640			

- Click to continue.

The **Required ADA/PCS Input Complete** dialog box appears.

- Click **OK**.

The **Required Input Complete** dialog box appears.

## Running the Simulation



- From the **Required Input Complete** dialog box, click **OK**.

The **Control Panel** appears.

As the run proceeds, status messages appear in the Control Panel. It takes several seconds for Aspen Plus to process input specifications and perform the simulation. When the calculations finish, the message *Results Available* appears in the status bar at the bottom of the main window.

- Use the vertical scroll bar to see the messages.



- 3 When the *Results Available* message appears in the toolbar, click  at the top right corner of the Control Panel to close it.
- 4 From the Data Browser, click  to close the Data Browser.
- 5 Examine the results of your run.

## Examining Simulation Results

Aspen Plus produces a wide variety of simulation results. In this simulation examine:

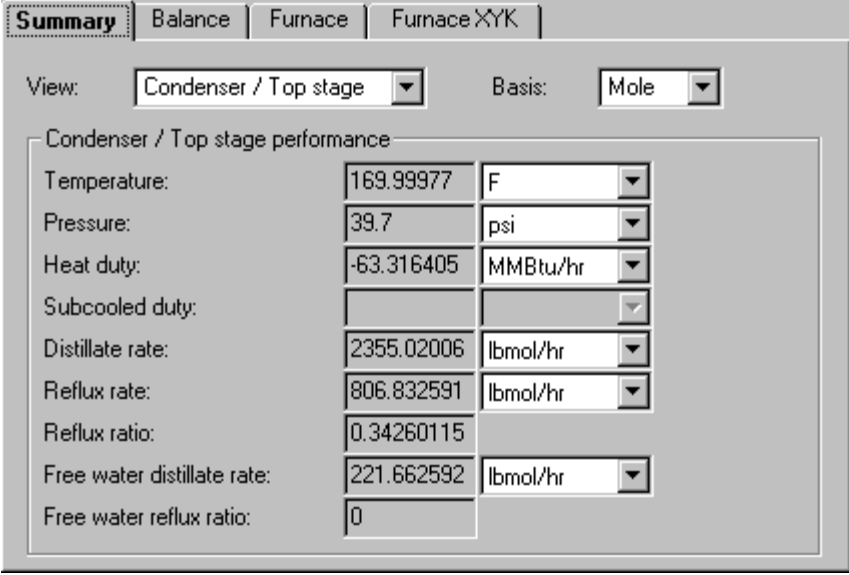
- Condenser duty
- Distillate rate
- Temperature profile
- Crude TBP curves

Condenser duty results appear on the **PREFLASH Results Summary** sheet.

### To Examine Summary Results

- 1 From the **Process Flowsheet** window, select the PREFLASH icon.
- 2 Right-click to display the menu and select Results.

The **PREFLASH | Results Summary | Summary** sheet appears, providing a summary of all condenser and bottom stage results.



The screenshot shows the 'Summary' sheet of the PREFLASH simulation results. The 'View' dropdown is set to 'Condenser / Top stage' and the 'Basis' dropdown is set to 'Mole'. The 'Condenser / Top stage performance' section displays the following data:

Temperature:	169.99977	F
Pressure:	39.7	psi
Heat duty:	-63.316405	MMBtu/hr
Subcooled duty:		
Distillate rate:	2355.02006	lbmol/hr
Reflux rate:	806.832591	lbmol/hr
Reflux ratio:	0.34260115	
Free water distillate rate:	221.662592	lbmol/hr
Free water reflux ratio:	0	

### To Examine Distillate Rate Results

Distillate rate results appear on the **PREFLASH | Design Specs | 1 | Results** sheet.

- 3 From the Data Browser, click the + beside the **Design Specs** folder and then select **1**.

The **Design Specs | 1 | Specifications** sheet appears.

4 Select the **Results** sheet.

The **Results** sheet appears, providing the distillate rate value that Aspen Plus calculated to satisfy the design specification.

The screenshot shows the 'Design specification' and 'Adjusted variable' sections of the Aspen Plus interface. The 'Design specification' section includes fields for Type (D86T), Target (375), Error (0.00065691), and Qualifiers (STREAM: NAPHTHA, LVPCT: 9.50000000D+01). The 'Adjusted variable' section includes fields for Type (DISTILLATE RATE OF MAIN COLUMN) and Final value (20232.1362 bbl/day).

The temperature profile for the column appears on the **PREFLASH | Profiles | TPFQ** sheet.

**To Examine the Temperature Profile**

5 From the Data Browser, select the **Blocks | PREFLASH | Profiles**.

The **PREFLASH | Profiles | TPFQ** sheet appears.

The screenshot shows the 'TPFQ' window with the 'Profiles' tab selected. The 'View' is set to 'Summary' and the 'Basis' is set to 'Mole'. The table displays the following data:

Stage	Temperature	Pressure	Heat duty	Liquid flow	Vapor flow
	F	psi	MMBtu/l	lbmol/hr	lbmol/hr
1	169.99977	39.7	-63.316405	2552.5596	609.2930
2	340.307096	41.7	0	770.446444	3383.515
3	375.977119	42.075	0	727.600198	3347.129
4	388.672941	42.45	0	688.659601	3304.282
5	395.212462	42.825	0	654.948594	3265.342
6	399.722448	43.2	0	621.288792	3221.621

## Plotting Results


### To Plot the Temperature Profile

- 1 From the **Aspen Plus** menu bar, select **Plot | Plot Wizard**.

The **Plot Wizard Step 1** dialog box appears.

- 2 Click .

The **Plot Wizard Step 2** dialog box appears.

- 3 Select the **Temp** plot type and click .

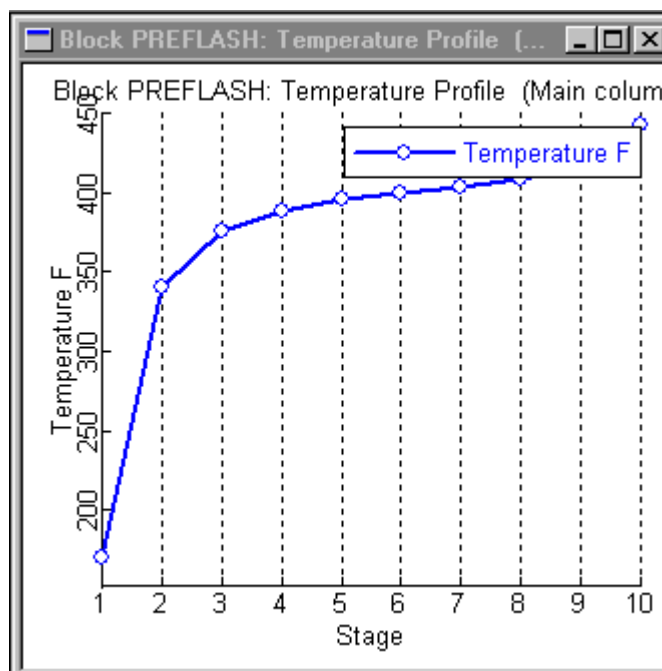
The **Plot Wizard Step 3** dialog box appears.

- 4 Click .

The **Plot Wizard Step 4** dialog box appears.

- 5 Click **Finish**.

The temperature profile plot appears.



- 6 Follow the same steps to display any other column profile.

You may want to close some forms to make space for the new plots.

### To Plot TBP Curves for PREFLASH Feed and Product Streams

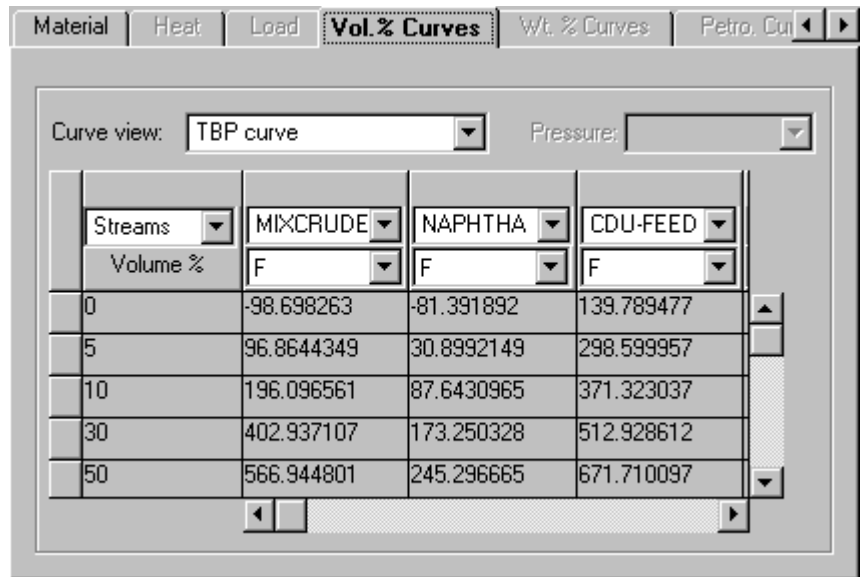
The true boiling point curves appear on the **Stream Results | Vol.% Curves** sheet.

- 1 From the Data Browser, select the **PREFLASH | Stream Results** form.

The **PREFLASH | Stream Results | Material** sheet appears.

- 2 Select the **Vol.% Curves** sheet.

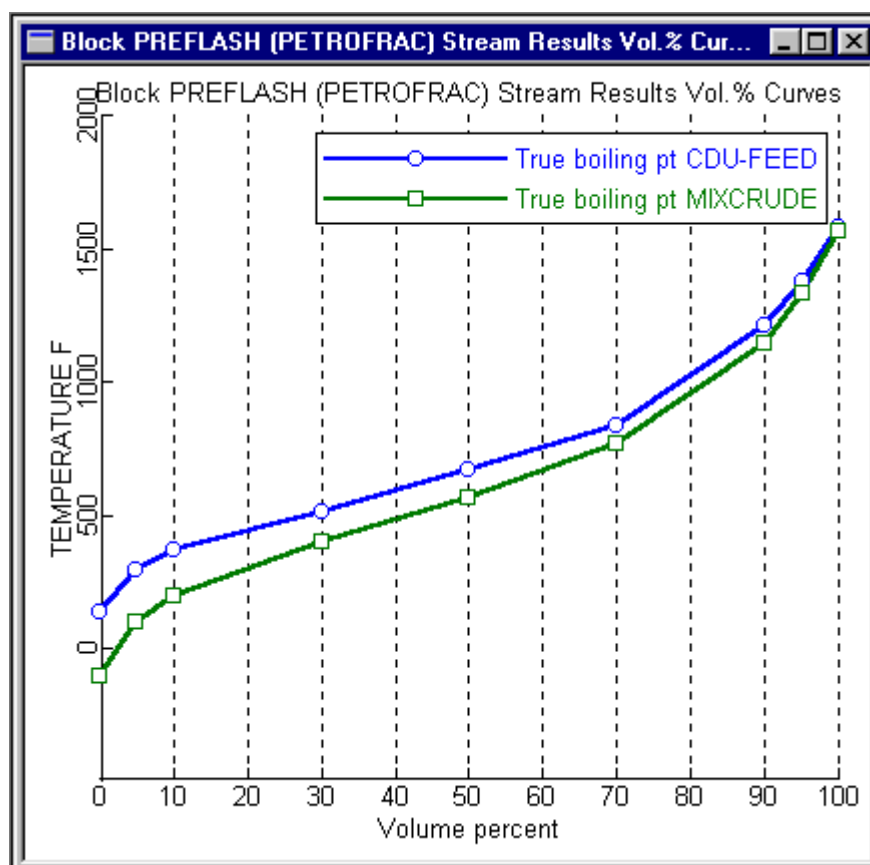
The **Stream Results Vol.% Curves** sheet appears.



### To Select CDU-FEED and MIXCRUDE as Dependent Variables

- 3 Click at the top of the **CDU-FEED** column to select the column.
- 4 Use the horizontal scrollbar, if necessary, to locate the **MIXCRUDE** column.
- 5 At the same time, press Ctrl and select the **MIXCRUDE** column to select the column.  
At this time, both the MIXCRUDE and CDU-FEED columns should be highlighted.
- 6 From the Aspen Plus menu bar, select **Plot | Y-Axis Variable**.
- 7 Select the **Volume %** column.
- 8 From the Aspen Plus menu bar, select **Plot | X-Axis Variable**.
- 9 From the Aspen Plus menu bar, select **Plot | Display Plot**.

10 Aspen Plus displays the following plot:



As expected, the preflash tower removes a significant portion of the light material from the process feed. This reduces the load for the subsequent atmospheric crude unit and increases overall process capacity.

11 When finished viewing the plot, close the plot window.

## Exiting Aspen Plus

1 From the **Aspen Plus** menu bar, select **File | Exit**.

The **Aspen Plus** dialog box appears.

2 Click **Yes** to save the simulation.

The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.

3 Click **No**.

The Chapter 4 simulation uses this run as the starting point.



---

# Adding an Atmospheric Crude Distillation Unit

In this simulation, add an atmospheric crude distillation unit to the simulation you ran in Chapter 3.

You will:

- Add the atmospheric crude distillation unit to the flowsheet.
- Specify additional feed streams.
- Specify the atmospheric crude distillation unit.
- Run the simulation.
- Examine the results.

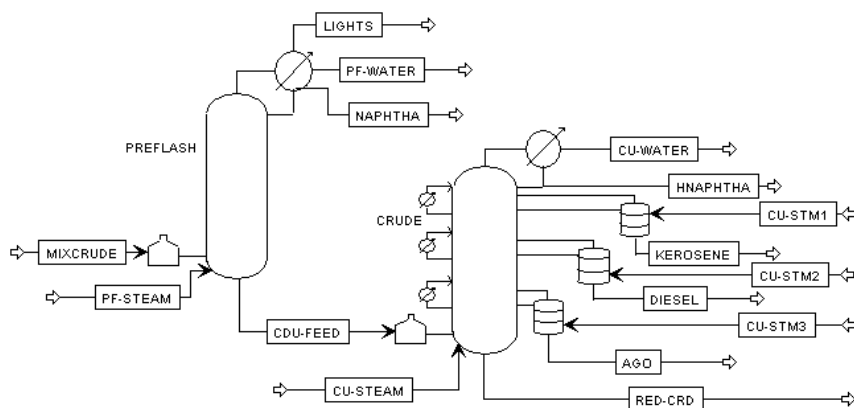
Allow about 45 minutes to complete this simulation.

## Atmospheric Crude Distillation Unit Flowsheet

Figure 4.1 shows the process flowsheet you are developing in this simulation. The topped crude from the preflash tower goes first to the crude furnace, then to the atmospheric tower. Model the crude furnace and the atmospheric tower units together, using a single PetroFrac block. This approach enables you to adjust furnace variables to achieve certain tower specifications, such as the overflash.

The tower has:

- A total condenser
- Three coupled side strippers
- Two pumparound circuits



**Figure 4.1** Process Flowsheet: The Atmospheric Crude Distillation Unit

PetroFrac simulates the furnace and the atmospheric tower simultaneously, as a single unit. The furnace operates at a pressure of 24.18 psia and provides an overflash of 3% in the tower. The furnace outlet enters the atmospheric tower on stage 22 of the main fractionator.

The main fractionator is modeled with 25 equilibrium stages. The heavy naphtha product flow is estimated at 13,000 bbl/day, and is manipulated to achieve an ASTM 95% temperature of 375 F. The condenser operates at 15.7 psia with a pressure drop of 5 psi. The tower pressure drop is 4 psi.



The main fractionator has 2 pumparound circuits. The following table summarizes the pumparound locations and specifications:

Pumparound	Location	Specifications
1	From stage 8 to 6	Flow: 49,000 bbl/day Duty: -40 MMBTU/hr
2	From stage 14 to 13	Flow: 11,000 bbl/day Duty: -15 MMBTU/hr

The crude distillation unit has three sidestrippers. Their locations and operating specifications are:

Stripper	Location	Specifications
KEROSENE	Liquid draw from stage 6 Vapor return to 5	Product rate: 11,700 bbl/day Steam stripping (CU-STM1) 4 equilibrium stages
DIESEL	Liquid draw from stage 13 Vapor return to 12	Product rate: 16,500 bbl/day (estimate) Steam stripping (CU-STM2) 3 equilibrium stages
AGO	Liquid draw from stage 18 Vapor return to 17	Product rate: 8,500 bbl/day Steam stripping (CU-STM3) 2 equilibrium stages

The diesel flow rate is an estimate only. Manipulate it to achieve an ASTM 95% temperature of 640 F for the diesel product.

All sidestrippers and the main fractionator use steam for stripping. The following table summarizes the steam flows and conditions:

Stream	Location	Conditions and Flow
CU-STEAM	Main tower	400 F, 60 psia, 12,000 lb/hr
CU-STM1	Kerosene stripper	400 F, 60 psia, 3,300 lb/hr
CU-STM2	Diesel stripper	400 F, 60 psia, 1,000 lb/hr
CU-STM3	AGO stripper	400 F, 60 psia, 800 lb/hr

## Starting Aspen Plus

- 1 From your desktop, select Start and then select Programs.
- 2 Select AspenTech, then Aspen Plus 11.0, then Aspen Plus User Interface.

The **Aspen Plus Startup** dialog box appears. Aspen Plus displays a dialog box whenever you must enter information or make a selection before proceeding.

## Opening an Existing Run

**If You Saved the Preflash Flowsheet Simulation Created in Chapter 3**

- 1 In the **Aspen Plus Startup** dialog box, select the radio button next to Open an Existing Simulation.
- 2 In the list, select preflash.bkp and click **OK**.

**If Your Saved File preflash.bkp Does Not Appear**

- 1 In the list box, double-click More Files....  
The **Open** dialog box appears.
- 2 Navigate to the directory containing your saved preflash.bkp.
- 3 Select preflash.bkp from the list of files, and click **Open**.

---

**Note:** If you did not create the assay data analysis simulation in Chapter 3, you can open the backup file preflash.bkp in the **Examples** folder.

---

**Opening the Backup File preflash.bkp in the Examples Folder**

- 1 From the **Aspen Plus Startup** dialog box, select Open an Existing Simulation.
- 2 Click **OK**.  
The **Open** dialog box appears.

- 3 Click .

---

**Note:** By default, the Favorites list contains five folders that are provided with Aspen Plus.

---

- 4 Double-click the **Examples** folder.
- 5 Select preflash.bkp and click **Open**.

---


**Note:** If the **Connect to Engine** dialog box appears, see Chapter 7.

---

The **Aspen Plus** window appears. Aspen Plus displays the graphical flowsheet for the PREFLASH run.

## Saving a Run Under a New Name

Before creating a new run adding the atmospheric crude unit, create and save a copy of PREFLASH with a new Run ID, CRUDE. Then make modifications under this new Run ID.

- 1 From the Aspen Plus menu bar, select **File | Save As**.  
The **Save As** dialog box appears.
- 2 Select the directory where you want to save the simulation.
- 3 In the **File name** field, enter CRUDE.
- 4 In the **Save as type** field, click  and select *Aspen Plus Backup Files (\*.bkp)*.
- 5 Click **Save** to save the simulation and continue.  
The **Aspen Plus** dialog box appears, asking if you want to save the simulation as an Aspen Plus Document file.
- 6 Click **No**.

The current Run ID, CRUDE now appears in the title bar.

### To Update the Title for This Run

- 1 From the Aspen Plus menu bar, select **Data | Setup**.  
The **Setup Specifications** sheet appears.
- 2 In the **Title** field, enter *Getting Started with Petroleum -- Simulation 3*.
- 3 Close the Data Browser window.  
The graphical workspace is now active and ready for you to add to the existing flowsheet.

## Adding the Crude Distillation Unit to the Flowsheet

Add the crude distillation unit to the flowsheet.


### To Make Space for the New Tower

- 1 Click and drag around the entire flowsheet.  
This creates a region around the entire flowsheet.
- 2 Click and hold the left mouse button within the region and drag the flowsheet to the left.  
This makes room for the crude distillation unit, as shown in Figure 4.1.

### To Place and Connect the Distillation Unit on the Flowsheet

- 1 Click the left mouse button outside the region to deactivate it.
- 2 From the Model Library, select the **Columns** tab.  
The list of available distillation columns appears.



- 3 Click  next to the PetroFrac block icon.  
This displays all the icons available for the PetroFrac model.
- 4 Move the mouse over the list of icons to see a name (or label) for each.  
In this exercise, select the CDU10F PetroFrac icon.
- 5 Press the mouse button on the CDU10F PetroFrac icon, and drag the icon to the **Process Flowsheet** window.
- 6 Release the icon in the desired location.  
The **Input** dialog box appears.
- 7 In the Enter the Block ID field, enter CRUDE and click **OK**.

#### To Connect to the Preflash Block Already on the Flowsheet

- 1 Select the CDU-FEED stream.
- 2 Right-click and from the menu select Reconnect Destination.
- 3 Point to the **Main Column Feed** port at the bottom of the column.
- 4 Click to connect the CDU-FEED stream to the bottom feed port and drag it to the desired location in front of the feed furnace before releasing the mouse button.

#### To Connect Other Streams

- 1 If necessary, select **View | Zoom | Zoom Full** from the Aspen Plus menu bar to display the entire flowsheet.
- 2 Connect, move, and name the streams as shown in Figure 4.1, using the names and ports in the table below.

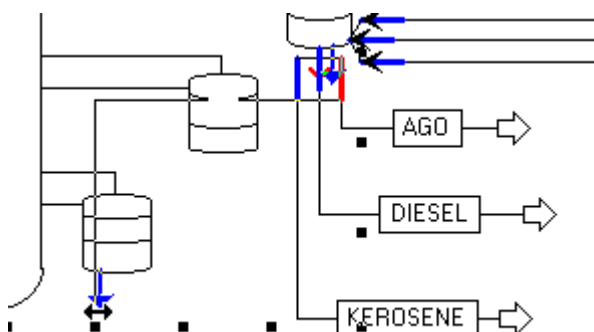
Stream ID	Port Name
CDU-FEED	Main Column Feed (already connected)
CU-STEAM	Main Column Feed
CU-STM1	Stripper Steam Feeds
CU-STM2	Stripper Steam Feeds
CU-STM3	Stripper Steam Feeds
CU-WATER	Condenser Water Decant for Main Column
HNAPHTHA	Liquid Distillate from Main Column
KEROSENE	Bottoms Product from Stripper
DIESEL	Bottoms Product from Stripper
AGO	Bottoms Product from Stripper
RED-CRD	Bottoms Product from Main Column

---

**Note:** In order to make the streams appear as in Figure 4.1, rotate some feed icons and move some ports.

---

- To rotate an icon, select it, then right-click and select Rotate Icon from the menu. Then select the appropriate rotation from the submenu.
- To move a port, first connect the stream to the port in the port's current location. Then select the block. All the connected streams will show a blue or red highlighted line where they connect to the block. Click on this line for the stream you wish to move, and drag it to its new location. Zoom in (as above for selecting Zoom Full) if it is difficult to differentiate streams. In the figure below, the user is moving the port for stream AGO.



You have finished adding the distillation unit to the flowsheet.

- 3 Click .

The **Flowsheet Complete** dialog box appears.

- 4 Click **OK**.

The **Streams | CU-STEAM | Input | Specifications** sheet appears.

## Specifying Steam Feeds to the Tower

Use the **CU-STEAM | Input | Specifications** sheet to enter the specifications for the steam feed to the main column.

- 1 Enter the following state variable and component flow specifications:

<b>Temperature</b>	400 F
<b>Pressure</b>	60 psi
<b>Composition</b>	Mass-Flow lb/hr
<b>Value for H2O</b>	12000 lb/hr

The **CU-STEAM | Input | Specifications** sheet is complete.

Substream name: **MIXED** Ref Temperature

State variables:

Temperature: 400 F

Pressure: 60 psi

Total flow: Stdvol, bbl/day

Solvent:

Composition:

Mass-Flow lb/hr

Component	Value
H2O	12000
C1	
C2	
C3	
IC4	
NC4	
IC5	
Total:	12000

2 Click .

The **CU-STM1 Input** sheet appears.

3 Enter the following state variable and component flow specifications:

**Temperature** 400 F

**Pressure** 60 psi

**Composition** Mass-Flow lb/hr

**Value for H2O** 3300 lb/hr

4 Click .

The **CU-STM2 Input** sheet appears.

5 Enter the following state variable and component flow specifications:

**Temperature** 400 F

**Pressure** 60 psi

**Composition** Mass-Flow lb/hr

**Value for H2O** 1000 lb/hr

6 Click .

The **CU-STM3 Input** sheet appears.

- 7 Enter the following state variable and component flow specifications:

<b>Temperature</b>	400 F
<b>Pressure</b>	60 psi
<b>Composition</b>	Mass-Flow lb/hr
<b>Value for H2O</b>	800 lb/hr

You have finished entering steam feed specifications.

- 8 Click .

The **Blocks | CRUDE | Setup | Configuration** sheet appears.

## Specify the Main Column

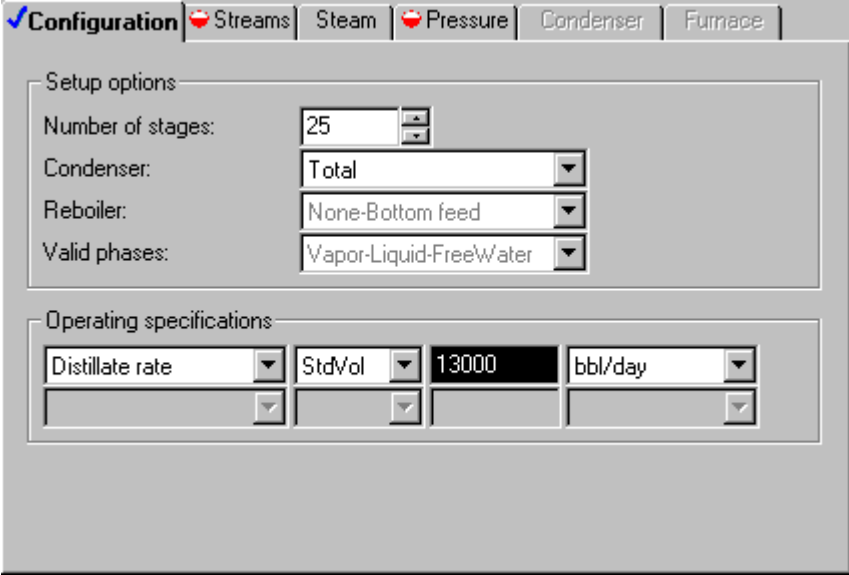
## Specifying the Atmospheric Tower

Use the **CRUDE | Setup | Configuration** sheet to specify the main column configuration.

- 1 Enter the following block configuration information:

<b>Number of stages</b>	25
<b>Condenser</b>	Total
<b>Reboiler</b>	None-Bottom Feed
<b>Distillate rate</b>	13000 bbl/day

The distillate flow rate is an estimate. This value will be manipulated to achieve the desired ASTM 95% temperature for the heavy naphtha stream.



The screenshot shows the 'Configuration' sheet of a software interface. At the top, there are tabs: 'Configuration' (selected with a checkmark), 'Streams', 'Steam', 'Pressure', 'Condenser', and 'Furnace'. Below the tabs, there are two main sections: 'Setup options' and 'Operating specifications'. In the 'Setup options' section, there are four rows: 'Number of stages' with a value of 25, 'Condenser' with a dropdown menu set to 'Total', 'Reboiler' with a dropdown menu set to 'None-Bottom feed', and 'Valid phases' with a dropdown menu set to 'Vapor-Liquid-FreeWater'. In the 'Operating specifications' section, there are four rows: 'Distillate rate' with a dropdown menu set to 'StdVol', a value of 13000, and a unit dropdown set to 'bbl/day'. Below these are four empty rows with dropdown menus.

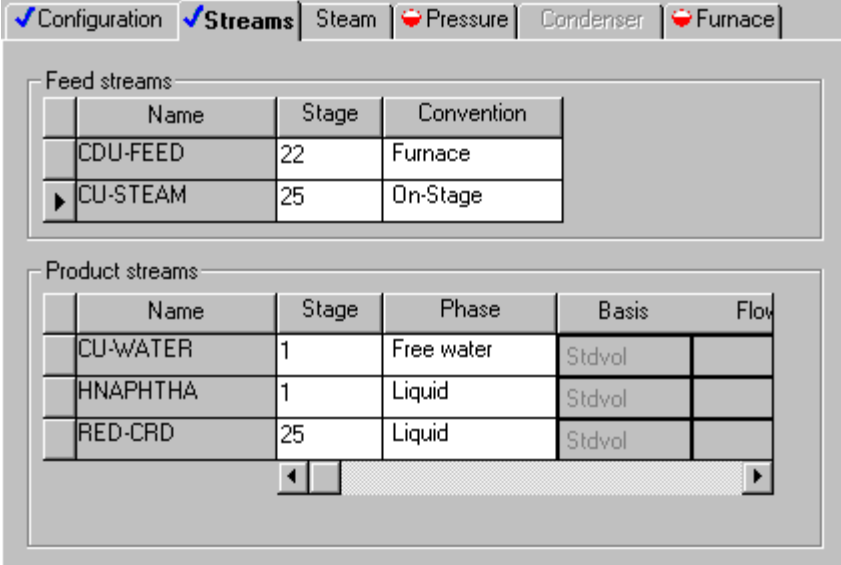
- 2 Click .

The **Blocks | CRUDE | Setup | Streams** sheet appears.

- 3 Enter the following feed locations and conventions:

	Stage	Convention
CDU-FEED	22	Furnace
CU-STEAM	25	On-Stage

The Furnace feed convention attaches a furnace to stage 22. This furnace is solved simultaneously with the main fractionation column and strippers.



Feed streams		
Name	Stage	Convention
CDU-FEED	22	Furnace
CU-STEAM	25	On-Stage

Product streams				
Name	Stage	Phase	Basis	Flow
CU-WATER	1	Free water	Stdvol	
HNAPHTHA	1	Liquid	Stdvol	
RED-CRD	25	Liquid	Stdvol	

- 4 Click .

The **Blocks | CRUDE | Setup | Pressure** sheet appears.

- 5 Enter the following pressures using the *Top/Bottom View*:

**Stage 1 / Condenser pressure** 15.7 psi

**Stage 2 Pressure** 20.7 psi

**Bottom stage pressure** 24.7 psi

- 6 Click .

The **Blocks | CRUDE | Setup | Furnace** sheet appears.

- 7 Select the **Single stage flash** option.

- 8 From the **Furnace specification** list, select Fractional overflash.

- 9 Enter a StdVol **Fractional overflash** of 0.03 and a Furnace pressure of 24.18 psi.



The **Blocks | CRUDE | Setup | Furnace** sheet is now complete:

The screenshot shows the Aspen Plus 'Furnace' configuration window. The 'Furnace' tab is active. The 'Furnace type' section has three radio buttons: 'Stage duty on feed stage', 'Single stage flash' (which is selected), and 'Single stage flash with liquid runback'. The 'Furnace specification' section has a dropdown menu set to 'Fractional overflash' and a text box containing '0.03'. The 'Furnace pressure' section has a text box containing '24.18' and a unit dropdown set to 'psi'.

You have completed specifications for the main column.

10 Click .

---

**Note:** Because you attached feed and product streams for three side strippers in the graphical flowsheet, Aspen Plus automatically configured this block with three side strippers: S-1, S-2, and S-3.

---

## Specify the Side Strippers

The **Blocks | CRUDE | Strippers | S-1 | Setup | Configuration** sheet appears.

1 Enter the following specifications:

<b>Number of stages</b>	4
<b>Stripper product</b>	KEROSENE
<b>Liquid draw</b>	6
<b>Overhead return</b>	5
<b>Stripping steam</b>	CU-STM1
<b>Bottom product flow</b>	11700 bbl/day

The **S-1 | Setup | Configuration** sheet is complete:

2 Click .

The **S-2 | Setup | Configuration** sheet appears.

3 Enter the following specifications:

<b>Number of stages</b>	3
<b>Stripper product</b>	DIESEL
<b>Liquid draw</b>	13
<b>Overhead return</b>	12
<b>Stripping steam</b>	CU-STM2
<b>Bottom product flow</b>	16500 bbl/day

The **Blocks CRUDE Strippers S-2 Setup** sheet is complete:

4 Click .

The **Blocks CRUDE Strippers S-3 Setup** sheet appears.

5 Enter the following specifications:

<b>Number of stages</b>	2
<b>Stripper product</b>	AGO
<b>Liquid draw</b>	18
<b>Overhead return</b>	17
<b>Stripping steam</b>	CU-STM3
<b>Bottom product flow</b>	8500 bbl/day

## Specify the Pumparounds

The **Blocks CRUDE Strippers S-3 Setup** sheet is complete:

- 6 Click .

The **Required Input Complete** dialog box appears.

- 7 Click **Cancel**.

Now supply specifications for Pumparound 1 and Pumparound 2.

- 1 In the Data Browser, select the **Blocks | CRUDE | Pumparounds** folder.

The **Pumparounds** object manager appears.

- 2 Click **New**.

- 3 In the **Create new ID** dialog box, click **OK** to accept the default ID, P-1.

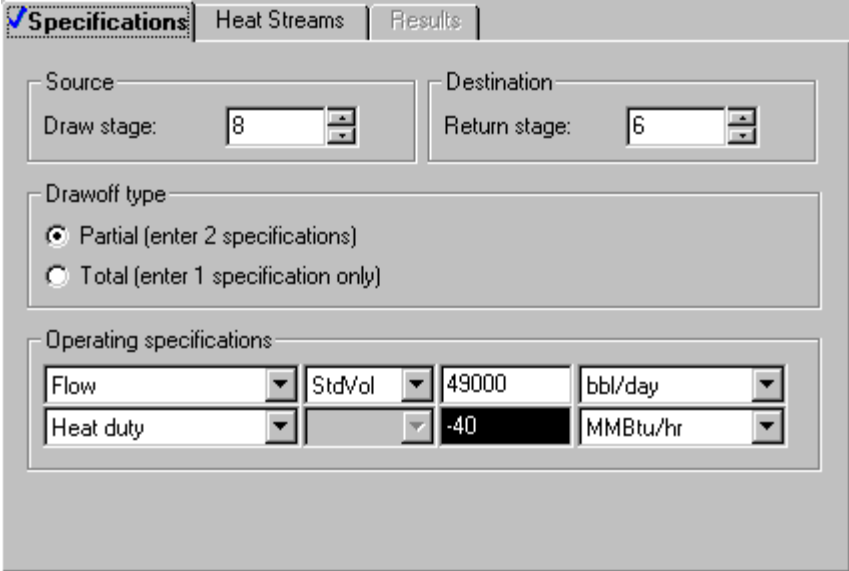
The **Pumparounds | P-1 | Specifications** sheet appears.

- 4 Enter the following specifications for P-1:

<b>Draw stage</b>	8
<b>Return stage</b>	6
<b>Drawoff type</b>	Partial
<b>StdVol Flow</b>	49000 bbl/day
<b>Heat duty</b>	-40.0 MMBtu/hr

To specify the heat duty, you will need to change *Temperature* to *Heat duty* for the second operating specification.

The **Pumparounds P-1 Specifications** sheet is complete:



Specifications			
Source		Destination	
Draw stage:	8	Return stage:	6
Drawoff type			
<input checked="" type="radio"/> Partial (enter 2 specifications) <input type="radio"/> Total (enter 1 specification only)			
Operating specifications			
Flow	StdVol	49000	bbl/day
Heat duty		-40	MMBtu/hr

- 5 Select the **Blocks | CRUDE | Pumparounds** folder to reopen the Pumparounds object manager.

## Enter Design Specifications

- 6 Click **New**.
- 7 In the **Create new ID** dialog box, click **OK** to accept the default ID, P-2.

The **Pumparounds | P-2 | Specifications** sheet appears.

- 8 Enter the following specifications for P-2:

<b>Draw stage</b>	14
<b>Return stage</b>	13
<b>Drawoff type</b>	Partial
<b>StdVol Flow</b>	11000 bbl/day
<b>Heat duty</b>	-15.0 MMBtu/hr

- 9 Click .

The **Required Input Complete** dialog box appears.

- 10 Click **Cancel**.

You have completed all required PetroFrac specifications.

However, in this simulation, specify two design specifications, the ASTM 95% temperature for HNAPHTHA and for DIESEL.

- 1 From the Data Browser, select the **Blocks | CRUDE | Design Specs** folder.


The **Design Specs** object manager appears.

- 2 Click **New**.

- 3 In the **Create new ID** dialog box, click **OK** to accept the default ID, 1.

The **Blocks | CRUDE | Design Specs | 1 | Specifications** sheet appears.

Enter specifications to have PetroFrac manipulate the distillate rate to achieve an ASTM 95% temperature of 375 F for HNAPHTHA:

- 4 In the **Type** field, click  and select *ASTM D86 temperature (dry, liquid volume basis)*.

- 5 Enter the following specifications:

<b>Target</b>	375 F
<b>Liquid %</b>	95

Specifications Components Feed/Product Streams Vary Results


Design specification  
Type: ASTM D86 temperature (dry, liquid volume basis)

Specification  
Target: 375 F

Liquid %: 95

6 Click .

The **Blocks | CRUDE | Design Specs | 1 | Feed/Product Streams** sheet appears.

7 From the **Available streams** column, select the HNAPHTHA stream and click  to move the stream to the **Selected stream** column.

Specifications Components Feed/Product Streams Vary Results

Product streams

Available streams

CU-WATER
RED-CRD
KEROSENE
DIESEL

> >> < <<

Selected stream

HNAPHTHA
----------

Feed/Product streams as base streams

8 Click .

The **Blocks | CRUDE | Design Specs | 1 | Vary** sheet appears.

9 In the **Type** field, click  and select *Distillate flow rate*.

10 From the Data Browser, select the **Blocks | CRUDE | Design Specs** folder to reopen the object manager.

For the second design specification, enter specifications to have PetroFrac manipulate the DIESEL flow rate to reach an ASTM 95% temperature of 640 F.

The **Design Specs Object Manager** appears.

**11** Click **New**.

**12** In the **Create new ID** dialog box, click **OK** to accept the default ID, 2.

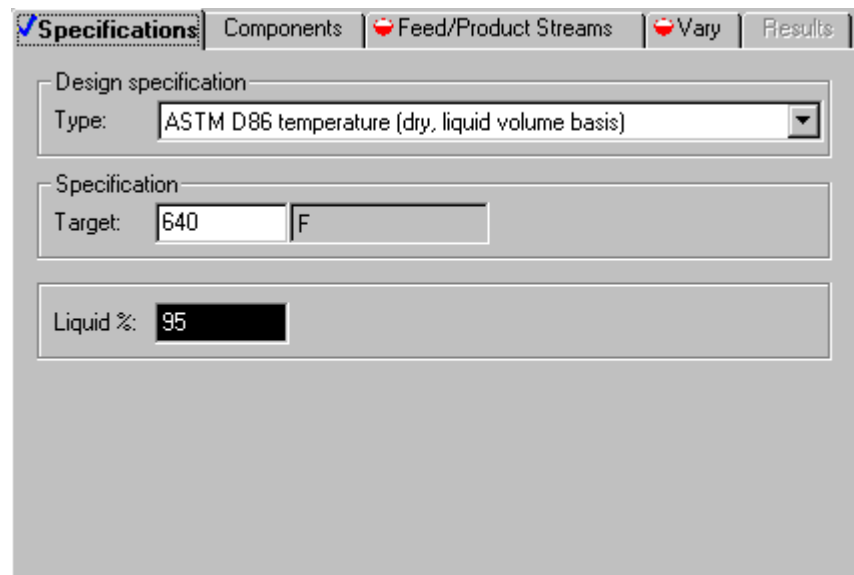
The **Blocks | CRUDE | Design Specs | 2 | Specifications** sheet appears.

**13** Enter the following specifications:

**Type**        ASTM D86 temperature (dry, liquid volume basis)

**Target**       640 F


**Liquid %**    95

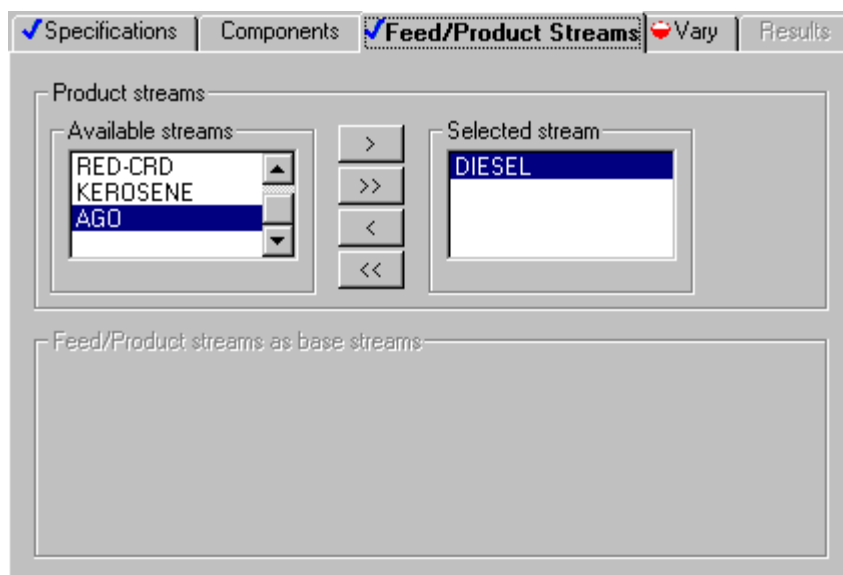


The screenshot shows the 'Specifications' tab of the 'Design Specs Object Manager' dialog box. The 'Design specification' section has a dropdown menu set to 'ASTM D86 temperature (dry, liquid volume basis)'. The 'Specification' section has a 'Target' field with '640' and a unit dropdown set to 'F'. The 'Liquid %' field is set to '95'.

**14** Click .

The **Blocks | CRUDE | Design Specs | 2 | Feed/Product Streams** sheet appears.

**15** From the **Available streams** column, select the DIESEL stream and click  to move the stream to the **Selected stream** column.



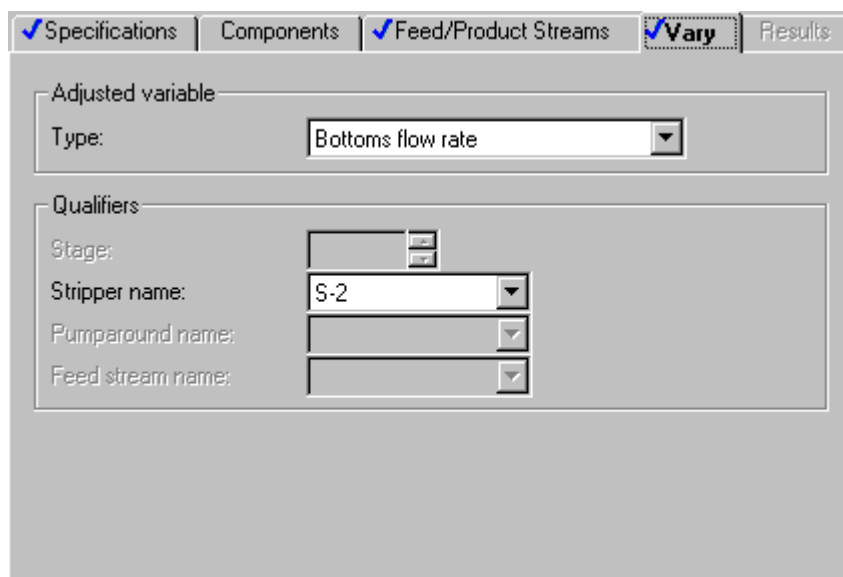
16 Click .

The **Blocks | CRUDE | Design Specs | 2 | Vary** sheet appears.

17 In the **Type** field, click  and select *Bottoms flow rate*.

18 In the **Stripper name** field, click  and select S-2.

The **Block CRUDE Design Specs 2 Vary** sheet is complete:



You have entered all the specifications for the atmospheric crude tower, without any initial profile estimates for tower temperatures, flow rates, or compositions. Aspen Plus can converge the column without estimates for these variables.



19 In the **Aspen Plus** window, click .

## Running the Simulation

- 1 In the **Required Input Complete** dialog box, click **OK**.

The **Control Panel** appears.

As the run proceeds, status messages appear in the Control Panel. It takes several seconds for Aspen Plus to process input specifications and perform the simulation. When the calculations finish, the message *Results Available* appears in the status bar at the bottom of the main window.

- 2 Use the vertical scrollbar to see the messages.
- 3 When the *Results Available* message appears in the toolbar, click  at the top right corner of the Control Panel to close it.
- 4 Click  at the top right corner of the Data Browser to close it.
- 5 Examine the results of your run.

## Examining Simulation Results

This run produces a wide variety of simulation results. In this simulation examine:

- Furnace duty
- Pumparound return temperatures
- Stream results for the atmospheric tower products
- ASTM curves for the tower products

Furnace duty results appear on the **CRUDE | Results Summary | Furnace** sheet.

### To Examine Furnace Duty Results

- 1 From the **Process Flowsheet** window, select the CRUDE icon.
- 2 Right-click to display the menu and select **Results**.  
The **Blocks | CRUDE | Results Summary | Summary** sheet appears.
- 3 Select the **Furnace** sheet.



A summary of important furnace results appears:

The screenshot shows a software window titled 'Furnace' with tabs for 'Summary', 'Balance', 'Furnace', and 'Furnace XYK'. The 'Furnace' tab is active. Below the tabs, there is a 'Basis:' dropdown menu set to 'Mole'. A section titled 'Furnace summary' contains a table of process parameters:

Temperature:	673.730773	F
Pressure:	24.18	psi
Heat duty:	199.769679	MMBtu/hr
Feed flow:	3869.27353	lbmol/hr
Vapor flow:	2565.57676	lbmol/hr
Liquid flow:	1303.69677	lbmol/hr

### To Examine Pumparound Return Temperatures

- 4 From the Data Browser, select the **CRUDE | Pumparounds** folder.
- 5 Select the **Results** sheet.

The **Pumparounds | Results** sheet appears:

The screenshot shows a software window titled 'Results' with tabs for 'Object Manager', 'Summary', and 'Results'. The 'Results' tab is active. Below the tabs, there is a section titled 'Pumparound results summary' containing a table with data for two pumparounds, P-1 and P-2:

Pumparound ID:		P-1	P-2
Return stage:		6	13
Temperature:	F	337.094803	389.254177
Pressure:	psi	21.7434783	22.7869565
Vapor fraction:		0	0
Heat duty:	MMBtu/hr	-40	-15
Mole flow:	lbmol/hr	3364.68332	577.370813
Mass flow:	lb/hr	593005.596	138509.677
StdVol flow:	bbl/day	48999.9954	11000.0275

## Examining the Product Streams



### To Review the Stream Results for the Crude Tower Products

- 1 From the Data Browser, select the **Blocks | CRUDE | Stream Results** form.

The **Blocks | CRUDE | Stream Results | Material** sheet appears.

## To Examine the Results for the Five Product Streams

## To Examine the 5% and 95% Cutpoints and Product Yields

- 2 In the first column of results, click  and select HNAPHTHA.
- 3 In the next four columns, click  and select KEROSENE, DIESEL, AGO, and RED-CRD.
- 4 Use the vertical scrollbar to locate **\*\*\* DRY TOTAL\*\*\*** and have it display in the top line.

From these results, you can see that the tower produces a large quantity of the diesel cut. By specifying a high ASTM 95% temperature (375 F), you also maximize the amount of heavy naphtha produced.

Stream results for the crude tower products appear on the **Blocks CRUDE Stream Results Material** sheet:


Material Heat Load Vol.% Curves Wt.% Curves Petro. Curves Poly. Curves						
Display:	Streams	Format:	PETRO_E	Stream Table		
	HNAPHTHA	KEROSENE	DIESEL	AGO	RED-CRD	
*** DRY TOTAL ***						
Liq Vol 60F bbl/day	7831.8	11693.0	17246.2	8499.9	34545.8	
API Gravity	50.3	39.2	33.4	27.0	16.5	
Gravity 60F	0.778	0.829	0.858	0.893	0.956	
Watson UOP-K	11.7	11.6	11.7	11.7	11.6	
TBP Curve F						
0 %	27.3	309.9	366.6	433.1	587.4	
5 %	138.3	363.0	449.1	557.6	686.3	
10 %	192.2	380.8	473.6	605.2	730.9	
30 %	264.2	414.1	519.7	672.1	827.4	
50 %	298.1	436.8	554.7	707.4	955.4	
70 %	332.9	458.6	594.2	743.3	1132.5	
90 %	370.4	490.0	646.0	788.1	1405.5	
95 %	387.3	501.0	667.0	804.2	1512.0	
100 %	417.5	533.7	697.7	861.4	1612.4	

- 5 Scroll and examine stream information for different products.

## Plotting Distillation Curves

Next, examine and plot the ASTM curves for the products. The ASTM curves are on the **Stream Results | Vol.% Curves** sheet.

## To Plot the Curves for the Product Streams

- 6 Select the **Vol.% Curves** sheet.  
The **CRUDE | Stream Results | Vol.% Curves** sheet appears.
- 7 In the **Curve view** field, click  and select *ASTM D86 curve*.
- 8 Select the **Volume %** column.
- 9 From the Aspen Plus menu bar, select **Plot | X-Axis Variable**.

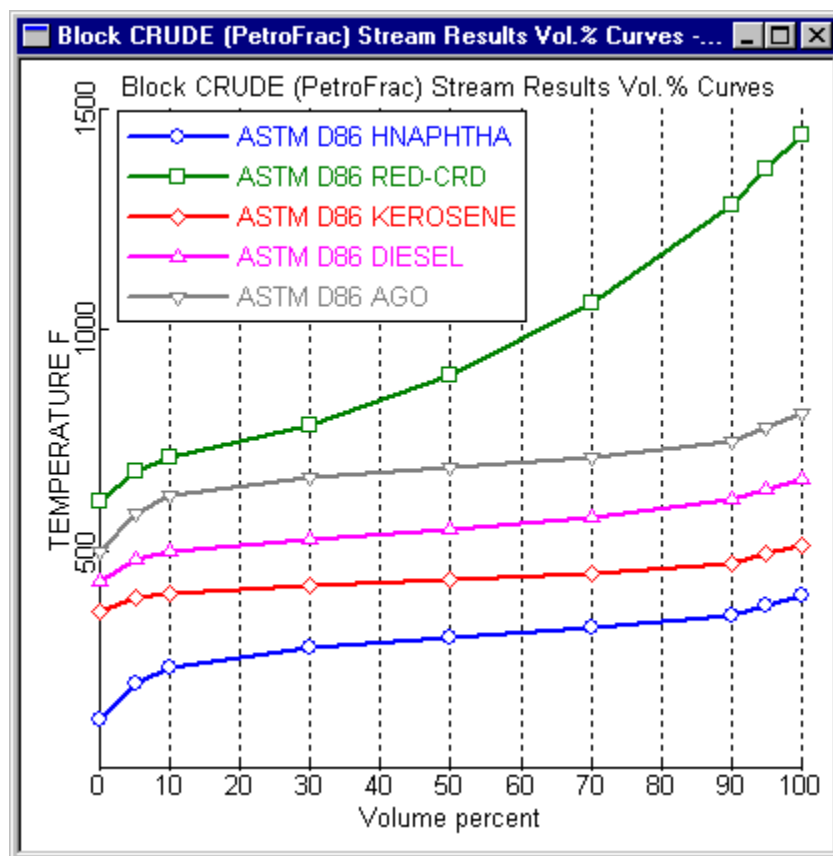
- 10 Select the **HNAPHTHA** column.
- 11 At the same time, press Ctrl and select the RED-CRD, KEROSENE, DIESEL, and AGO columns to select the columns.

---

**Note:** At this time, the HNAPHTHA, RED-CRD, KEROSENE, DIESEL, and AGO columns should be highlighted.

---

- 12 From the Aspen Plus menu bar, select **Plot | Y-Axis Variable**.
- 13 From the Aspen Plus menu bar, select **Plot | Display Plot**.
- 14 Resize the plot and drag the legend box as desired.



- 15 When finished viewing the plot, close the plot window.

At this point, simulate other modes of operation, such as maximizing kerosene production to satisfy a higher demand for jet fuel. Do this by:

- Decreasing the ASTM 95% temperature for heavy naphtha to 325 F
- Increasing the ASTM 95% temperature for kerosene to about 520 F
- Introduce another design specification by manipulating the kerosene flow rate

The product flow rates resulting from these specifications are:

	<b>ASTM-D86 5% Temperature F</b>	<b>ASTM-D86 95% Temperature F</b>	<b>Product Flow- Rate (m bbl/day)</b>
HNAPHTHA	166	325	4.7
KEROSENE	374	520	20.0
DIESEL	502	640	10.9
AGO	578	772	8.5
RED-CRD	674	1358	35.6

The results show that the kerosene production goes up at the expense of heavy naphtha and diesel.

## Exiting Aspen Plus

- 1 From the **Aspen Plus** menu bar, select **File | Exit**.

The **Aspen Plus** dialog box appears.

- 2 Click **Yes** to save the simulation.

The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.

- 3 Click **No**.

The Chapter 5 simulation uses this run as the starting point.

---

# Adding A Vacuum Unit

In this simulation, build on the simulation from the previous simulation and install a vacuum unit to complete the process model.

You will:

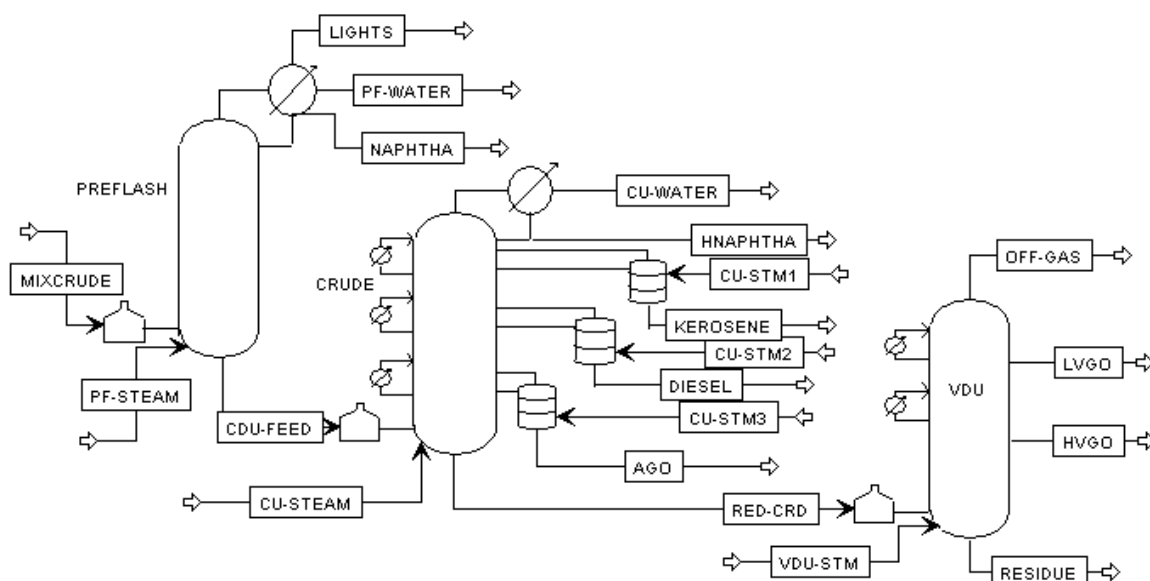
- Add a vacuum tower, steam feed, and product streams to the flowsheet
- Specify the steam feed
- Specify the vacuum unit
- Run the simulation
- Examine the results

Allow about 30 minutes to complete this simulation.

## Vacuum Unit Flowsheet

Figure 5.1 shows the process flowsheet for this simulation. It contains a vacuum unit that separates the reduced crude from the atmospheric tower into:

- Off-gas
- Light vacuum gas oil
- Heavy vacuum gas oil
- Residual vacuum oil



**Figure 5.1** Process Flowsheet Adding a Vacuum Unit

The vacuum tower has no condenser or reboiler. Stripping steam is fed to the bottom of the tower in stream VDU-STM at 400 F, 60 psia, 20,000 lb/hr.

The furnace operates at a pressure of 2.03 psia, and provides an overflash of 0.6%. The overflash is bypassed to the tower furnace.

The tower has two pumparound circuits. The following table shows their locations and specifications:

Pumparound	Location	Specifications
1	From stage 2 to 1	Flow: 20,000 bbl/day Duty: -28 MMBtu/hr (estimated)
2	From stage 4 to 3	Flow: 49,000 bbl/day Duty: -80 MMBtu/hr

The duty for the first pumparound is adjusted so that the top of the tower is at 150 F.

You will model the tower with six equilibrium stages. The light vacuum gas oil is taken out from stage 2 as a total draw. The flow is estimated at 8,000 bbl/day.

The second pumparound provides all the necessary reflux for the lower section of the tower. The heavy vacuum gas oil is withdrawn from stage 4 at 17,000 bbl/day.

## Starting Aspen Plus

- 1 From your desktop, select **Start** and then select **Programs**.
- 2 Select **AspenTech | Aspen Engineering Suite | Aspen Plus 11.1 | Aspen Plus User Interface**.

The **Aspen Plus Startup** dialog box appears.

## Opening an Existing Run

- 1 Select **Open an Existing Simulation**.
- 2 In the list, select **crude.bkp** and click **OK**.

**If You Saved the Crude Flowsheet from Chapter 4**

**If Your Saved File crude.bkp Does Not Appear**


- 1 In the list box, double-click **More Files**.  
The **Open** dialog box appears.
- 2 Navigate to the directory containing your saved **crude.bkp**.
- 3 Select **crude.bkp** from the list of files, and click **Open**.

---

**Note:** If you did not create the assay data analysis simulation in Chapter 4, you can open the backup file **crude.bkp** in the **Examples** folder.

---

**Opening the Backup File preflash.bkp in the Examples Folder**

- 1 In the list box, double-click **More Files**.
- 2 Click **OK**.  
The **Open** dialog box appears.
- 3 Click .
- 4 Double-click the **Examples** folder.
- 5 Select **crude.bkp** and click **Open**.

---

**Note:** If the **Connect to Engine** dialog box appears, see Chapter 7.

---

The **Aspen Plus** window appears. Aspen Plus displays the graphical flowsheet for the CRUDE run.

## Saving a Run Under a New Name


Before you create a new run adding the vacuum unit, create and save a copy of CRUDE with a new Run ID, VACUUM. Then make modifications under this new Run ID.

- 1 From the Aspen Plus menu bar, select **File | Save As**.

The **Save As** dialog box appears.

- 2 Select the directory where you want to save the simulation.

- 3 In the **File name** field, enter VACUUM.

- 4 In the **Save as type** field, click  and select *Aspen Plus Backup Files (\*.bkp)*.

- 5 Click **Save** to save the simulation and continue.

The **Aspen Plus** dialog box appears, asking if you want to save the simulation as an Aspen Plus Document file.

- 6 Click **No**.

The current Run ID, VACUUM now appears in the title bar.

### To Update the Title for This Run

- 1 From the Aspen Plus menu bar, select **Data | Setup**.

The **Setup | Specifications | Global** sheet appears.

- 2 In the **Title** field, enter Getting Started with Petroleum -- Simulation 4.

- 3 Close the Data Browser window.

The graphical workspace is now active and ready for you to add to the existing flowsheet.



## Adding a Vacuum Tower to the Flowsheet

You are ready to add the vacuum tower and steam feed to the flowsheet.

### To Make Space for the New Tower


- 1 From the Aspen Plus menu bar, select **View | Zoom | Zoom Full**.
- 2 Click and drag a region around the entire flowsheet.
- 3 Press and hold the left mouse button within the region and drag the flowsheet to the left.

Next, place and connect the vacuum unit on the flowsheet to create the process flowsheet in Figure 5.1. If you need to review how to create a graphical simulation flowsheet, see *Getting Started Building and Running a Process Model*, Chapter 3.

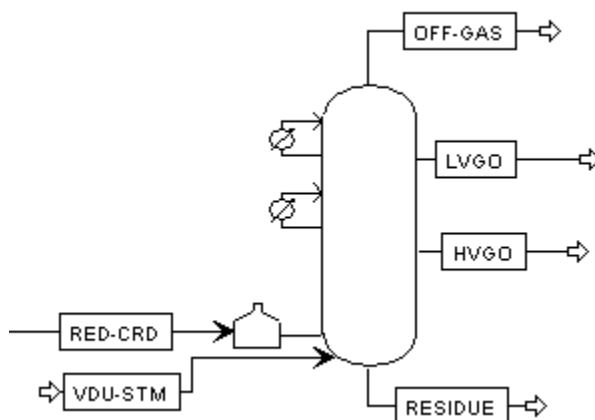
### To Place and Connect the Distillation Unit on the Flowsheet

- 1 From the Model Library, select the **Columns** tab.  
The list of available distillation columns appears.



- 2 Click  next to the PetroFrac block icon.
- 3 Move the mouse over the list of icons to see a name (or label) for each.
- 4 Select the VACUUM1F PetroFrac icon, and drag the icon to the **Process Flowsheet** window as shown in Figure 5.1.
- 5 Release the icon in the desired location.  
The **Input** dialog box appears.
- 6 In the **Enter the Block ID** field, enter VDU and click **OK**.
- 7 Select the RED-CRD stream.
- 8 Right-click and from the menu select Reconnect Destination.
- 9 Point to the **Main Column Feed** port at the bottom of the new column.
- 10 Click to connect the RED-CRD stream to the bottom feed port and drag it to the desired location in front of the feed furnace before releasing the mouse button.

### To Connect the Distillation Unit to the Crude Block



*Figure 5.2 The Vacuum Unit*

- 11** Connect, move, and name the other streams as shown in Figure 5.2.

Use the table below to help you connect to the appropriate port.

Stream ID	Port Name
RED-CRD	Main Column Feed (already connected)
VDU-STM	Main Column Feed
OFF-GAS	Vapor Distillate from Main Column
LVGO	Side Product from Main Column
HVGO	Side Product from Main Column
RESIDUE	Bottoms Product from Main Column

You have finished adding the distillation unit to the flowsheet.

- 12** Click .

The **Flowsheet Complete** dialog box appears.

- 13** Click **OK**.

The **VDU-STM | Input | Specifications** sheet appears.

## Specifying the Steam Feed

Use the **VDU-STM | Input | Specifications** sheet to enter the specifications for the steam feed to the main column.

- 1 Enter the following state variable and component flow specifications:

<b>Temperature</b>	400 F
<b>Pressure</b>	60 psi
<b>Composition</b>	Mass-Flow lb/hr
<b>Value for H2O</b>	20000 lb/hr

The **VDU-STM | Input | Specifications** sheet is complete.

Substream name: **MIXED** Ref Temperature

State variables:

Temperature: 400 F

Pressure: 60 psi

Total flow: Stdvol bbl/day

Solvent:

Composition:

Mass-Flow lb/hr

Component	Value
H2O	20000
C1	
C2	
C3	
IC4	
NC4	
IC5	
Total:	20000

- 2 Click .

The **Blocks | VDU | Setup | Configuration** sheet appears.

## Specify the Basic Column Configuration

# Specifying the Vacuum Unit

Use the **VDU | Setup | Configuration** sheet to specify the main column configuration.

- 1 Enter the following block configuration information:

<b>Number of stages</b>	6
<b>Condenser</b>	None-Top pumparound
<b>Reboiler</b>	None-Bottom feed

- 2 Click .

The **Blocks VDU Setup Streams** sheet appears.

- 3 Enter the following feed locations and connections:

	<b>State</b>	<b>Convention</b>
RED-CRD	6	Furnace
VDU-STM	6	On-Stage

The Furnace feed convention attaches a furnace to stage 6. This furnace is solved simultaneously with the vacuum tower.

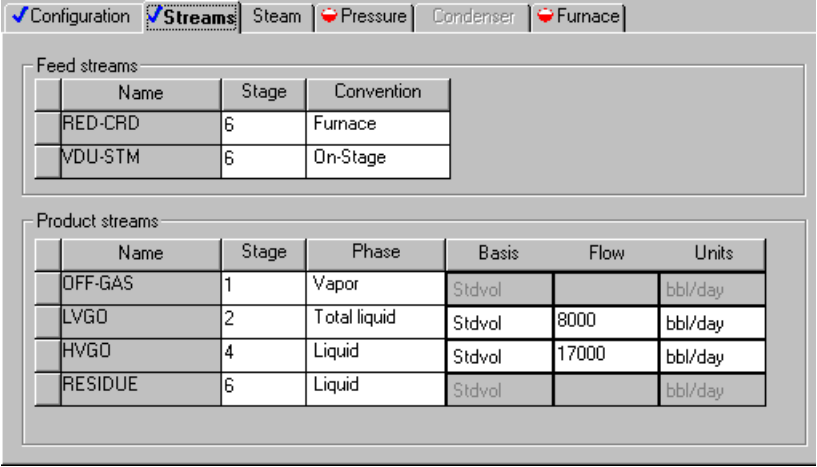
- 4 Enter the following product location, phase, and rate specifications:

	<b>Stage</b>	<b>Phase</b>	<b>Flow</b>
LVGO	2	Total liquid	8000 bbl/day (estimate)
HVGO	4	Liquid	17000 bbl/day

LVGO is a total liquid draw from stage 2.

**Note:** The flow rate you enter for LVGO is an estimate only.

The Blocks VDU Setup Streams sheet is complete:



The screenshot shows the 'Streams' tab of the VDU Setup interface. It contains two tables: 'Feed streams' and 'Product streams'.

Feed streams			
Name	Stage	Convention	
RED-CRD	6	Furnace	
VDU-STM	6	On-Stage	

Product streams						
Name	Stage	Phase	Basis	Flow	Units	
OFF-GAS	1	Vapor	Stdvol		bbl/day	
LVGO	2	Total liquid	Stdvol	8000	bbl/day	
HVGO	4	Liquid	Stdvol	17000	bbl/day	
RESIDUE	6	Liquid	Stdvol		bbl/day	

5 Click .

The **Blocks | VDU | Setup | Pressure** sheet appears.

6 In the **View** field, click  and select *Pressure profile*.

Specify the pressure profile by entering stage numbers and corresponding pressures on this sheet.

7 In the **Pressure profile** section, change the pressure units to *mmHg*.

8 Enter the following values:


Stage	Pressure (mmHg)
1	60
6	70

The **Blocks | VDU | Setup | Pressure** sheet is complete.

9 Click .

The **Blocks | VDU | Setup | Furnace** sheet appears.

10 In the **Furnace type** section, select the *Single stage flash with liquid runback* option.

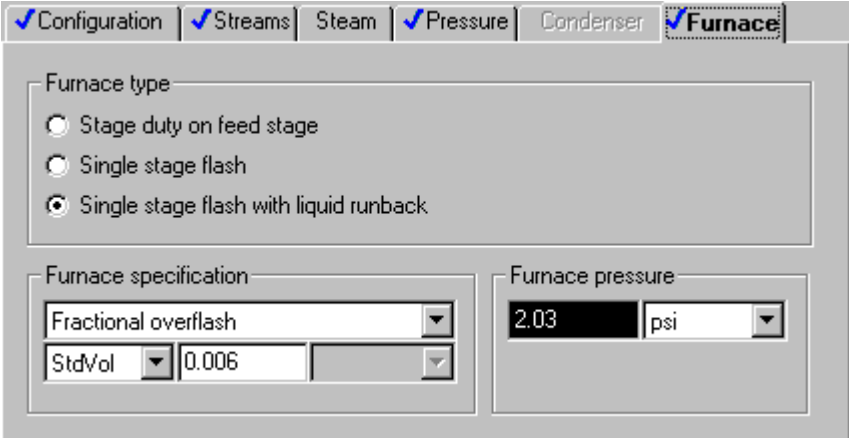
11 In the **Furnace specification** section, click  and select *Fractional overflash*.

12 In the **Fractional overflash** field, enter a StdVol of 0.006.

13 In the **Furnace pressure** field, enter 2.03 psi.

The single stage flash with liquid runback option models the furnace as a single-stage flash with the overflash from the flash zone now bypassed to the furnace.

The **Blocks VDU Setup Furnace** sheet is complete:



14 Click .

## Specify the Pumparounds

The **VDU | Pumparounds** object manager appears.

- 1 Click **New**.
- 2 In the **Create new ID** dialog box, click **OK** to accept the default ID P-1.

The **VDU | Pumparounds | P-1 | Specifications** sheet appears.

- 3 Enter the following specifications for P-1:

<b>Draw stage</b>	2
<b>Return stage</b>	1
<b>Drawoff type</b>	Partial
<b>StdVol Flow</b>	20000 bbl/day
<b>Heat duty</b>	-28.6 MMBtu/hr

Change *Temperature* to *Heat duty* in the second **Operating Specification** field to enter the heat duty.

The **VDU | Pumparounds | P-1 | Specifications** sheet is complete:

Source		Destination	
Draw stage:	2	Return stage:	1

Drawoff type

☒ Partial (enter 2 specifications)  
☐ Total (enter 1 specification only)

Operating specifications

Flow	StdVol	20000	bbl/day
Heat duty		-28.6	MMBtu/hr

- 4 From the Data Browser, select the **Pumparounds** folder.  
The **VDU | Pumparounds** object manager appears.
- 5 Click **New**.
- 6 In the **Create new ID** dialog box, click **OK** to accept the default ID P-2.

7 Enter the following specifications for P-2:

**Draw stage** 4  
**Return stage** 3  
**Drawoff type** Partial (enter 2 specifications)  
**StdVol Flow** 49000 bbl/day  
**Heat duty** -80.0 MMBtu/hr

The **VDU | Pumparounds | P-2 | Specifications** sheet is complete:

The screenshot shows the 'Specifications' tab of a dialog box. It has three sub-sections: 'Source' with 'Draw stage' set to 4; 'Destination' with 'Return stage' set to 3; and 'Drawoff type' with 'Partial (enter 2 specifications)' selected. The 'Operating specifications' section contains two rows: 'Flow' set to 'StdVol' with a value of 49000 and units of 'bbl/day', and 'Heat duty' set to a value of -80 and units of 'MMBtu/hr'.

8 Click .

The **Required Input Complete** dialog box appears.

You have completed all required PetroFrac specifications. However, in this simulation you have a design specification—the temperature specification for the top of the tower.




- 1 Click **Cancel**.
- 2 From the Data Browser, select the **Blocks | VDU | Design Specs** folder.

The **Blocks | VDU | Design Specs** object manager appears.


- 3 Click **New**.
- 4 In the **Create new ID** dialog box, click **OK** to accept the default ID 1.  
The **Blocks | VDU | Design Specs | 1 | Specifications** sheet appears.

## Enter Design Specification

- 5 Enter the following specifications:

<b>Type</b>	Stage temperature
<b>Target</b>	150 F
<b>Stage</b>	1
- 6 Click .  
The **Blocks | VDU | Design Specs | 1 | Vary** sheet appears.
- 7 In the **Type** field, click  and select *Pumparound duty*.
- 8 In the **Pumparound name** field, select P-1.  
The **Blocks | VDU | Design Specs | 1 | Vary** sheet is complete.
- 9 Close the **Data Browser** window.
- 10 In the **Aspen Plus** window, click .  
The **Required Input Complete** dialog box appears.

## Running the Simulation

- 1 In the **Required Input Complete** dialog box, click **OK**.  
The **Control Panel** appears.  
As the run proceeds, status messages appear in the Control Panel. It takes several seconds for Aspen Plus to process input specifications and perform the simulation. When the calculations finish, the message *Results Available* in the status bar at the bottom of the main window.
- 2 Use the vertical scrollbar to see the messages.
- 3 When the *Results Available* message appears in the toolbar, click  at the top right corner of the Control Panel to close it.
- 4 Examine the results of your run.

## Examining Simulation Results

This run produces a wide variety of simulation results. Examine simulation results for the vacuum tower that include:

- Column profiles
- Pumparound results
- TBP curves
- Stream results

Column profiles are on the **VDU | Profiles** form.



## To Examine Column Profiles

- 1 From the **Process Flowsheet** window, select the VDU icon.
- 2 Right-click to display the menu and select **Results**.  
The **Blocks | VDU | Results Summary | Summary** sheet appears.
- 3 From the Data Browser, select the **Blocks | VDU | Profiles** form.

The screenshot shows the 'TPFQ' window with the 'Profiles' tab selected. The 'View' dropdown is set to 'Summary' and the 'Basis' dropdown is set to 'Mole'. The 'Profiles' table displays data for 6 stages. The columns are Stage, Temperature, Pressure, Heat duty, Liquid flow, and Vapor flow. The units for Temperature, Pressure, Heat duty, Liquid flow, and Vapor flow are F, psi, MMBtu/l, lbmol/hr, and lbmol/hr, respectively.

Stage	Temperature	Pressure	Heat duty	Liquid flow	Vapor flow
	F	psi	MMBtu/l	lbmol/hr	lbmol/hr
1	150.000523	1.16020648	-25.972014	812.253373	1110.323
2	320.623505	1.19888003	0	1095.1345	1131.671
3	524.632182	1.23755358	-80	2181.28363	1414.552
4	603.653872	1.27622713	0	2084.13675	2146.602
5	781.846609	1.31490068	0	4.28229958	2049.455
6	791.046703	1.35357423	0	178.205631	1203.667

This sheet shows that the simulation calculated a bottoms temperature above 790 F. If this is higher than you want, you can reduce the tower pressure in a later simulation case. Lowering the tower pressure also affects the gas oil yield. The pumparound results are on the **VDU | Pumparounds | Results** sheet.

## To Examine Pumparound Results

- 4 From the Data Browser, select the **Blocks | VDU | Pumparounds** folder.
- 5 Select the **Results** sheet.

The **Blocks VDU Pumparounds Results** sheet appears:

Object Manager   Summary   <b>Results</b>			
Pumparound results summary			
Pumparound ID:		P-1	P-2
▶ Draw stage:		2	4
Return stage:		1	3
Temperature:	F	128.287024	419.360728
Pressure:	psi	1.19888003	1.27622713
Vapor fraction:		0	0
Heat duty:	MMBtu/hr	-25.972014	-80
Mole flow:	lbmol/hr	790.905173	1449.23391
Mass flow:	lb/hr	263664.418	675834.017
StdVol flow:	bbl/day	19999.9636	49000.087

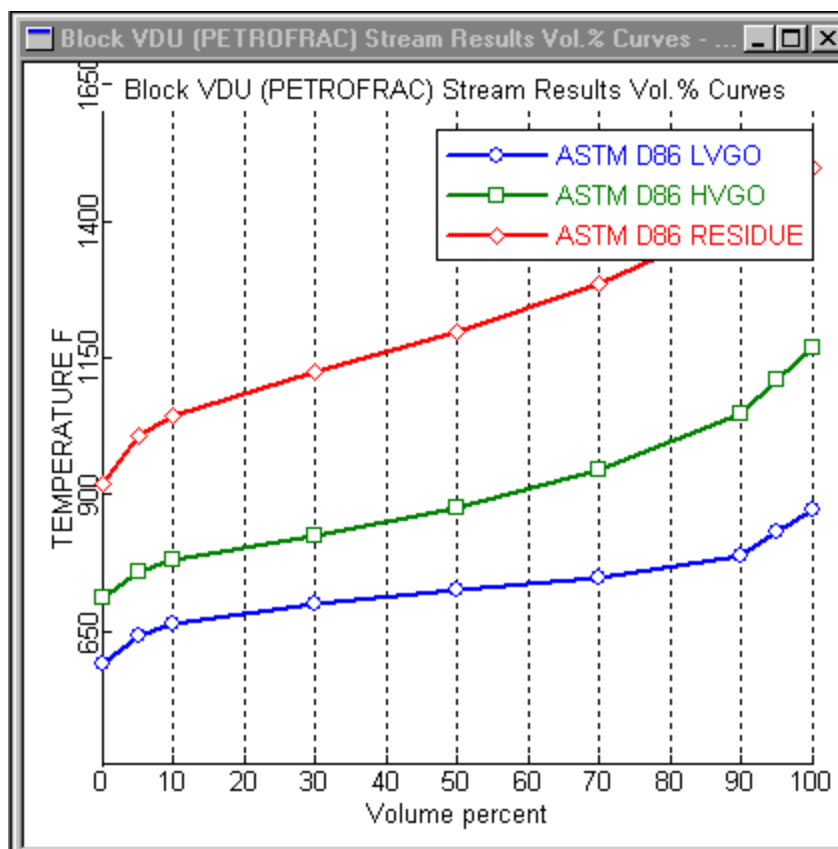
6 Close the window.

## Plotting Distillation Curves

Next, plot the TBP curve for streams LVGO, HVGO, and RESIDUE. The True Boiling Point curves appear on the **Stream Results | Vol.% Curves** sheet.

### To Plot TBP Curves

- 1 From the Data Browser, select the **VDU | Stream Results** form.  
The **VDU | Stream Results | Material** sheet appears.
- 2 Select the **Vol.% Curves** sheet.
- 3 Plot the ASTM curves for streams LVGO, HVGO, and RESIDUE following the same steps as in *Chapter 4, Plotting Distillation Curves*.



The simulation shows a good separation between the gas oils and residue. However, the separation between the light and heavy gas oils is rather sloppy.

- 4 When finished viewing the plot, close the plot window.

## Examining Stream Results

- 1 From the Data Browser, select the **VDU | Stream Results | Material** sheet.
- 2 Use the horizontal scrollbar to locate the RESIDUE stream and review the information accordingly.
- 3 Use the vertical scrollbar to locate **API Gravity**.
- 4 Use the horizontal scrollbar to locate the LVGO stream and review the API Gravity information.

The calculated value for LVGO product rate is different from the specified value of 8000 bbl/day. Since you specified a total draw, Aspen Plus treated the specified value as an initial guess. The final value is the one corresponding to the total draw from the stage.

## Exiting Aspen Plus

- 1 From the **Aspen Plus** menu bar, select **File | Exit**.  
The **Aspen Plus** dialog box appears.
- 2 Click **Yes** to save the simulation.  
The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.
- 3 Click **No**.

The Chapter 6 simulation uses this run as the starting point.

---

# Displaying Petroleum Stream Properties

It is important to monitor certain petroleum properties such as sulfur content, PNA composition, octane number, flash point, metals content, and pour point when modeling refinery operations. Frequently, operating parameters are adjusted in order to achieve a desired product quality, expressed in terms of these petroleum properties.

These properties are often available as part of the assay data for a crude feed. Aspen Plus enables you to enter these data as property curves. Once you enter these curves, Aspen Plus can calculate these properties for any flowsheet stream.

Use the calculated stream properties as column specifications, access them in Fortran blocks, use them in optimization constraints, and so on.

In this simulation, as an example, include sulfur content in your model.

You will:

- Add a sulfur curve for each of the two crude oils
- Select options for the property sulfur
- Define a property set for the sulfur content
- Run the simulation
- Examine simulation results

Allow about 30 minutes to complete this simulation.

## Starting Aspen Plus

- 1 From your desktop, select **Start** and then select **Programs**.
- 2 Select **AspenTech | Aspen Engineering Suite | Aspen Plus 11.1 | Aspen Plus User Interface**.

The **Aspen Plus Startup** dialog box appears.

## Opening an Existing Run

### If You Saved the Simulation Created in Chapter 5

- 1 In the **Aspen Plus Startup** dialog box, select **Open an Existing Simulation**.
- 2 In the list, select **vacuum.bkp** and click **OK**.

### If Your Saved File vacuum.bkp Does Not Appear


- 1 In the list box, double-click **More Files**.  
The **Open** dialog box appears.
- 2 Navigate to the directory containing your saved **vacuum.bkp**.
- 3 Select **vacuum.bkp** from the list of files, and click **Open**.  
The **Aspen Plus** window appears.

---

**Note:** If you did not create the assay data analysis simulation in Chapter 5, you can open the backup file **crude.bkp** in the **Examples** folder.

---

### Opening the Backup File preflash.bkp in the Examples Folder

- 1 From the **Aspen Plus Startup** dialog box, select **Open an Existing Simulation**.
- 2 Click **OK**.  
The **Open** dialog box appears.
- 3 Click .
- 4 Double-click the **Examples** folder.
- 5 Select **vacuum.bkp** and click **Open**.

---


**Note:** If the **Connect to Engine** dialog box appears, see Chapter 7.

---

The **Aspen Plus** window appears. Aspen Plus displays the graphical flowsheet for the **VACUUM** run.

## Saving a Run Under a New Name

Before you create a new run adding sulfur content specifications, create and save a copy of VACUUM with a new Run ID, PPROP. Then make modifications under this new Run ID.

- 1 From the Aspen Plus menu bar, select **File | Save As**.  
The **Save As** dialog box appears.
- 2 Select the directory where you want to save the simulation.
- 3 In the **File name** field, enter PPROP.
- 4 In the **Save as type** field, click  and select *Aspen Plus Backup Files (\*.bkp)*.
- 5 Click **Save** to save the simulation and continue.  
The **Aspen Plus** dialog box appears, asking if you want to save the simulation as an Aspen Plus Document file.
- 6 Click **No**.  
The current Run ID, PPROP now appears in the title bar.


### To Update the Title for This Run

- 1 From the Aspen Plus menu bar, select **Data | Setup**.  
The **Setup | Specifications | Global** sheet appears.
- 2 In the **Title** field, enter *Getting Started with Petroleum -- Simulation 5*.
- 3 Close the Data Browser window.  
The graphical workspace is now active. However, you will not modify the flowsheet in this simulation.

## Adding Sulfur Content Data

In Chapter 2 you defined and blended two crude oils, OIL-1 and OIL-2. Now enter the sulfur content (in weight percent) of these oils into your simulation.

### To Enter Sulfur Content for OIL-1

- 1 From the Aspen Plus menu bar, select **Data | Components**.  
The Data Browser appears, with the **Components** folder open.
- 2 Double-click the **Components | Assay/Blend** folder.
- 3 Double-click the **OIL-1** folder.
- 4 Select **Property Curves**.  
The **Assay/Blend | OIL-1 | Property Curves | Petro Properties** sheet appears.
- 5 In the **Property type** field, click  and select SULFUR.
- 6 In the **Bulk value** field, enter 2.3.

- 7 In the **Mid percent distilled** and **Property value** fields, enter the following sulfur content percentage values:

Mid percent distilled	Property value (wt% sulfur)
2.0	0.0
5.0	0.01
10.0	0.013
20.0	0.05
30.0	1.15
40.0	1.62
45.0	1.90
50.0	2.15
60.0	2.54
70.0	3.0
80.0	3.7

The **OIL-1 | Property Curves | Petro Properties** sheet is complete:

**Petro Properties** Viscosity


Property type: ☒ **SULFUR**

Property curve data

Bulk value: 2.3

	Mid percent distilled	Property value
	45	1.9
	50	2.15
	60	2.54
	70	3
	80	3.7
*		

### To Enter Sulfur Content for OIL-2

- 1 Double-click the **Assay/Blend | OIL-2** folder.
- 2 Select **Property Curves**.  
The **Assay/Blend | OIL-2 | Property Curves | Petro Properties** sheet appears.
- 3 In the **Property type** field, click  and select **SULFUR**.
- 4 In the **Bulk value** field, enter 2.5.



- 5 Enter the following sulfur content percentage values:

Mid percent distilled	Property value (wt% sulfur)
2.0	0.0
5.0	0.01
10.0	0.015
20.0	0.056
30.0	1.3
40.0	1.7
45.0	2.0
50.0	2.3
60.0	2.7
70.0	3.2
80.0	3.8

The **OIL-2 | Property Curves | Petro Properties** sheet is complete.

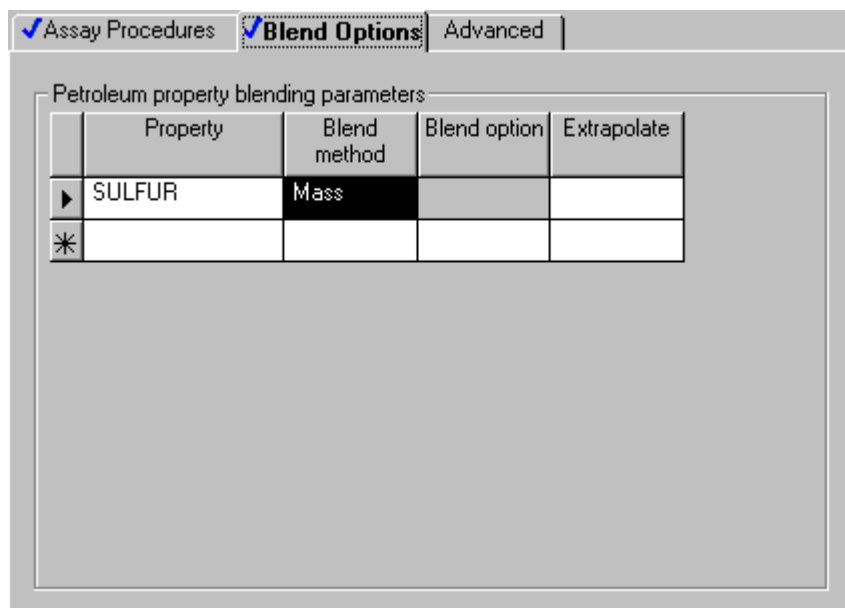
## Selecting Options for the Sulfur Property

For a petroleum property, select calculation options such as the blending and extrapolation methods.

- 1 From the Data Browser, double-click the **Components | Petro Characterization** folder.
- 2 Select the **Analysis Options** sheet.  
The **Petro Characterization | Analysis Options | Assay Procedures** sheet appears.
- 3 Select the **Blend Options** sheet.
- 4 In the first **Property** field select *SULFUR*.
- 5 In the first **Blend method** field select *Mass*.

By default, Aspen Plus extrapolates to the 0 and 100% points if these values are missing from the curve.

The **Petro Characterization | Analysis Options | Blend Options** sheet is complete:



6 Close the **Data Browser** window.

## Adding the Sulfur Property

In order to include in the report the sulfur property you just defined, create a property set.

### To Create the Property Set

- 1 From the Aspen Plus menu bar, select **Data | Properties**.
- 2 From the **Properties** folder, double-click the **Prop-Sets** folder.  
The **Prop-Sets** object manager appears.
- 3 Click **New**.
- 4 In the **Create new ID** dialog box, click **OK** to accept the default ID PS-1.
- 5 Click **Search**.
- 6 In the **Search Physical Properties** dialog box, enter *Sulfur* in the first field and click **Search**.  
A list now appears in step 2 of the dialog box.
- 7 From the list of physical property names, select *Sulfur content*.
- 8 Click **Add**.  
Sulfur content now appears under the third section.

**Search Physical Properties**

1 Type physical property you want to find and then click Search

Sulfur Search

☐ Search pure components only

2 Click one or more properties you want, and then click Add.

Physical property name	Alias
Mass fraction of sulfur atoms	MASSFRS
Mole flow of sulfur atoms	MOLEFLS
Mole fraction of sulfur atoms	MOLEFRS
Sulfur content	SULFUR
Sulfur content curve	SULFCRV

Add

3 Click OK to accept the selection.

Physical property name	Alias
Sulfur content	SULFUR

Remove

OK Cancel Help

9 Click **OK**.

SULFUR now appears as a selected physical property on the **Prop-Set Properties** sheet:

**Properties** | **Qualifiers**

Properties

	Physical properties	Units
▶	SULFUR	
*		

Substream: MIXED Search

10 Select the **Qualifiers** sheet.

11 In the **Water basis** field, click ▼ and select *Dry*.

The **Prop-Sets PS-1** sheet is now complete:


The Qualifiers dialog box shows the following qualifiers for selected properties:

Qualifier	System	Value
Phase		
Component		
2nd liquid key component		
Temperature	<input checked="" type="checkbox"/>	
Pressure	<input checked="" type="checkbox"/>	
% Distilled		
Water basis		Dry

Boiling point range:  to   
 Base boiling point range:  to

### To Add the Property Set to the Stream Property List

12 Close the **Data Browser** window.

- 1 From the Aspen Plus menu bar, select **Data | Setup**.
- 2 From the **Setup** folder, select the **Report Options** form and then select the **Stream** sheet.
- 3 Click **Property Sets**.  
The **Property Sets** dialog box appears.
- 4 From the **Available Property Sets** column, select PS-1 and click  to move it to the **Selected Property Sets** column.

The Property Sets dialog box shows the following property sets:

Available property sets	Selected property sets
HXDESIGN	PETRO
KINVIS	PS-1
LIGHT	
MIDDLE	
TBP-5	
TBP-95	

Property set ID. Use List to list existing IDs; or enter a new ID.

5 Click **Close**.

The **Setup Report Options** sheet is now complete.

6 Close the **Data Browser** window.

7 Click .

The **Required Input Complete** dialog box appears.


## Running the Simulation

1 In the **Required Input Complete** dialog box, click **OK**.

The **Control Panel** appears.

As the run proceeds, status messages appear in the Control Panel. It takes several seconds for Aspen Plus to process input specifications and perform the simulation. When the calculations finish, the message *Results Available* in the status bar at the bottom of the main window.

2 Use the vertical scrollbar to see the messages.

3 When the *Results Available* message appears in the toolbar, click  at the top right corner of the Control Panel to close it.

## Examining Results


Review the stream results for the crude tower products.

1 Click .

The **Results Summary | Run-Status** form appears and indicates that the calculations were completed normally.

2 In the Data Browser, click .

The **Results Summary | Streams | Material** sheet appears.

3 In the first column heading, click  and select HNAPHTHA.

4 In the next four columns, click  and select KEROSENE, DIESEL, AGO, and RED-CRD.

5 Use the vertical scrollbar to locate **SULFUR percent**.

The **Results Summary Streams Material** sheet displays sulfur content:

		HNAPHTHA	KEROSENE	DIESEL	AGO	RED-CRD
50 %		96.7	207.5	304.1	432.7	650.3
70 %		124.2	225.2	337.0	463.6	812.6
90 %		154.1	250.8	380.5	502.3	1075.2
95 %		167.6	259.8	398.3	516.3	1181.8
100 %		191.9	286.7	424.4	566.5	1284.8
SULFUR percent		0.303	1.484	2.225	2.927	4.057

As expected, the sulfur content of the heavier products, such as RED-CRD, is higher than the sulfur content for the lighter products, such as HNAPHTHA. By following the same procedure described in this simulation, incorporate other petroleum stream properties into your model and track their distribution throughout the flowsheet.

## Leaving Aspen Plus

- 1 From the **Aspen Plus** menu bar, select File | Exit.

The **Aspen Plus** dialog box appears.

- 2 Click **Yes** to save the simulation.

The **Aspen Plus** dialog box appears and offers you the opportunity to save the simulation as an Aspen Plus Document file.

- 3 Click **No**.


---

# Connecting to the Aspen Plus Simulation Engine

If either of the following conditions exist, you will be prompted to specify the host computer for the Aspen Plus simulation engine after you start the Aspen Plus User Interface:

- The simulation engine is not installed on your PC.
- The simulation engine is installed on your PC, but the Activator security device is not connected to your PC.

In these cases, the **Connect to Engine** dialog box appears.

- 1 In the **Server type** field, click  and select the type of host computer for the simulation engine.
- 2 If you choose *Local PC* as the server for the simulation engine, you do not need to enter any more information into the dialog box. Click **OK** to continue.

If you choose *Windows NT or 2000 server* as the server for the simulation engine, specify additional options.

- 3 In the **Node name** field, enter the name of the computer on which the Aspen Plus simulation engine will execute.
- 4 In the other fields, enter the following information:

<b>User name</b>	Your user name for the specified host/server.
<b>Password</b>	Your password for the above user name.
<b>Working directory</b>	The associated working directory.
- 5 Click **OK**.

When the network connection is established, the message *Connection Established* appears in the message box.

---

**Note:** If the *Connection Established* message does not appear, see your Aspen Plus system administrator for more information on network protocols and host computers for the Aspen Plus simulation engine.

---