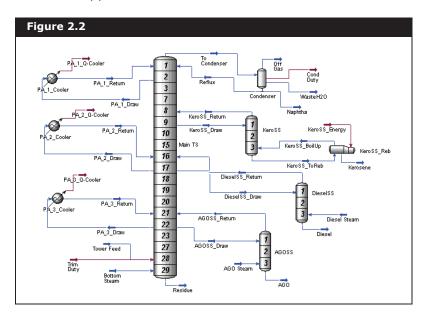
flowsheet appears below.



The following pages guide you through building a UniSim Design case for modeling this process. This tutorial illustrates the complete construction of the simulation, from selecting a property package and components, characterizing the crude oil, to installing streams and unit operations, through to examining the final results. The tools available in UniSim Design are utilized to illustrate the flexibility available to you.

Before proceeding, you should have read Chapter A - UniSim Design Tutorials which precedes the Tutorials in this guide.

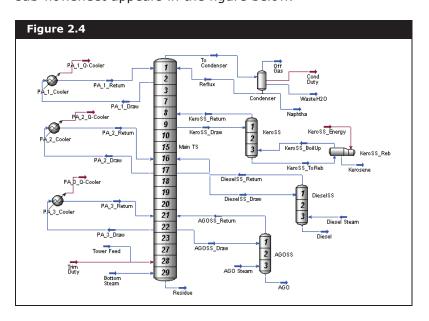
# 2.2 Steady State Simulation2.2.1 Process Description

This example models a crude oil processing facility consisting of a prefractionation train used to heat the crude liquids, and an atmospheric crude column to fractionate the crude into its straight run products. The Figure 2.3 Naphtha OffGas TrimDuty WasteH2O TowerFeed PreFlashVap −**→** Kerosene PreFlast BottomSteam Preheat Crude DieselSteam Mixer Diesel Furnace ¢ AGOSteam HotCrude PreFlashLig Residue Atmos Tower CrudeDut AGO CondDute

Preheated crude (from a preheat train) is fed to the pre-flash drum, modeled as a Separator, where vapours are separated from the crude liquids. The liquids are then heated to 650°F in the crude furnace, modeled as a Heater. The pre-flash vapours bypass the furnace and are re-combined, using a Mixer, with the hot crude stream. The combined

The crude column is modeled as a Refluxed Absorber, equipped with three pump-around and three side stripper operations. The Column sub-flowsheet appears in the figure below.

stream is then fed to the atmospheric crude column for separation.



The main column consists of 29 trays plus a partial condenser. The TowerFeed enters on stage 28, while superheated steam is fed to the bottom stage. In addition, the trim duty is represented by an energy

Main Flowsheet for this process appears in the following figure.

stream feeding onto stage 28. The Naphtha product, as well as the water stream WasteH2O, are produced from the three-phase condenser. Crude atmospheric Residue is yielded from the bottom of the tower.

Each of the three-stage side strippers yields a straight run product. Kerosene is produced from the reboiled KeroSS side stripper, while Diesel and AGO (atmospheric gas oil) are produced from the steamstripped DieselSS and AGOSS side strippers, respectively.

The two primary building tools, Workbook and PFD, are used to install the streams and operations and to examine the results while progressing through the simulation. Both of these tools provide you with a large amount of flexibility in building your simulation, and in quickly accessing the information you need.

The Workbook is used to build the first part of the flowsheet, from specifying the feed conditions through to installing the pre-flash separator. The PFD is then used to install the remaining operations, from the crude furnace through to the column.

# 2.2.2 Setting Your Session Preferences

1. Start UniSim Design and create a new case. The Simulation Basis

The Workbook displays information about streams and unit operations in a tabular format, while the PFD is a graphical representation of the flowsheet. Manager view appears.

NoName.usc							
e Edit Basis T	iools Windo	18.11.18.117. <b>8</b> 10.				Environment Mode	: Basis : Steady State
Simulation B	asis Mana	ger					
-Component Lists							
Master Compon	ent List	<u>V</u> iew					
		Add	1				
		Delete	1				
		Сору	1				
			1				
		Import					
		Export					
		<u>B</u> efresh					
Components	Fluid Pkgs	Hypotheticals	0 il Manager	Reactions	Component Maps	UserProperty	)
_						Enter Simu	llation Environment
				Initi	alization of P	VT Harness F	ailed.

Your first task is to set your Session Preferences.

- 2. From the Tools menu, select Preferences.
  - The Session Preferences view appears.

The most important preference you will set is the unit set. UniSim Design does not allow you to change any of the default unit sets listed, however, you can create a new unit set by cloning an existing one. In this tutorial you will create a new unit set based on the UniSim Design Field set and customize it.

3. Click the **Variables** tab, then select the **Units** page.

The default Preference file is named **UniSim Design.prf**. When you modify any of the preferences, you can save the changes in a new Preference file by clicking the Save Preference Set button. UniSim Design prompts you to provide a name for the new Preference file, which you can later use in any simulation case by clicking the Load Preference Set button.

4. In the Available Unit Sets group, select Field.

Variables	Available Unit Sets			
Units	Draft_CRTC5		~	Clone
Formats	EuroSI Field			Billion
onnats	Field13			Delete
	Unit Set Name Field			
	Display Units			
		Unit	<b>•</b>	<u>V</u> iew
	Acidity	mg KOH/g		100
	Act. Gas Flow	ACFM	_	A <u>d</u> d
	Act. Vol. Flow Actual Liquid Flow	barrel/day USGPM		Delete
	Actual Mass Density	kg/m3		
	Angle	deg	-	Az+

5. Click the **Clone** button.

A new unit set named NewUser appears and is automatically selected as the current unit set.

6. In the **Unit Set Name** field, rename the new unit set to **Field-density**.

You can now change the units for any variable associated with this new unit set.

7. In the Display Units group, use the vertical scroll bar to find the **Standard Density** cell.

The current default unit for **Standard Density** is lb/ft3. A more appropriate unit for this example is API\_60.

- 8. Click in the **Standard Density** cell on lb/ft3.
- 9. Press the **SPACEBAR** or the **DOWN** arrow to open the drop-down list of available units.

10. In the unit list, select **API\_60**.

Variables	Available Unit Sets		
Units	EuroSI Field		Clone
Formats	Field-density SI		Delete
		ld-density	View Users
	Display Units	lu s	
	Specific Volume	Unit ft3/lb >	<u>⊻</u> iew
	Standard Density	lb/ft3	✓ Add
	Std Gas Den	g/cm3	▲ Delete
	Std. Gas Flow Std. Vol. Flow	g/mL Ib/ft3	
	Steam Ratio	API 60	M A₂↓
		SG_60/60api SG_60/60nbs SG_15/4	-

11. Repeat steps #8-#10 to change the **Mass Density** units to **API**.

igure 2.8	
Variables	Available Unit Sets
Units	EuroSI Clone
Formats	Field-density Delete
	Unit Set Name Field-density View Users
	Display Units Unit ↓ Mass Concentration wt %
	Mass Concentration ppm ppmwt Add
	Mass Cp Btu/lb-F Delete
	Mass Enthalpy g/cm3 Mass Enthalpy Flow g/cm2
<u>,</u>	b/t3 API SG_60/60api SG_H2060nbs SG_H204 ▼
	riables Reports Files Resources Extensions Oil Input Tray Sizing

12. Your new unit set is now defined. Close the Session Preference view to return to the Simulation Basis Manager view.

# 2.2.3 Building the Simulation Selecting Components

Before defining a fluid package in UniSim Design, you will create a component list for the fluid package. In this example, the component list contains non-oil components, Light Ends, and hypocomponents. You must first add the non-oil components and Light Ends from UniSim

All commands accessed via the toolbar are also available as Menu items.

2-9

Design pure component library into the component list.

1. Click the **Components** tab, then click the **Add** button. The Component List View view appears.

Add Component	Selected Components	-		le in the Component Library	
- Components Traditional			<u>M</u> atch		View Filters
····· Electrolyte ···· Hypothetical			Sim Name	🔿 Full Name / Synonym	C Formula
Other		<add pure<="" td=""><td>Ethane</td><td>01 C2</td><td>CH4 C2H6</td></add>	Ethane	01 C2	CH4 C2H6
		<-Substitute->	i-Butane i	C3 i-C4	C3H8 C4H10
		<-Substitute->		n-C4 i-C5	C4H10 C5H12
		Remove>	n-Pentane i	n-C5 C6	C5H12 C6H14
			n-Heptane I	C7 C8	C7H16 C8H18
		Sort List	n-Nonane I	C9 C10	C9H20 C10H22
		⊻iew Component	n-C11 (	C11 C12	C11H24 C12H26
			Show Synonyms		

There are a number of ways to select components for your simulation. One method is to use the matching feature.

Notice that each component is listed in three ways on the Selected tab:

Matching Method	Description
SimName	The name appearing within the simulation.
FullName/ Synonym	IUPAC name (or similar), and synonyms for many components.
Formula	The chemical formula of the component. This is useful when you are unsure of the library name of a component, but know its formula.

At the top of each of these three columns is a corresponding radio button. Based on the selected radio button, UniSim Design will locate the component(s) that best matches the input you type in the **Match** cell.

2. Optional: To rename the component list, click in the **Name** field at the bottom of the view and type a new name.

For this tutorial example, you will add the following non-oil components: H2O, C3, i-C4, n-C4, i-C5 and n-C5.

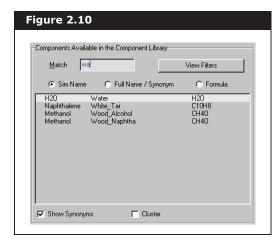
First, you will add **H2O** using the match feature.

3. Ensure the **Sim Name** radio button is selected, and the **Show Synonyms** checkbox is checked.

The Component List View view contains two tabs. In this example, the Selected tab is the only tab used, because it contains all the functions you need to add components to the list. You can also move to the Match field by pressing **ALT M**.

2-10

- 4. Click in the **Match** field.
- 5. Begin typing `water'. UniSim Design filters through its library as you type, displaying only those components that match your input.



- 6. With Water selected, add it to the Current Component List by doing **one** of the following:
  - Press the ENTER key.
  - Click the **Add Pure** button.
  - Double-click on Water.

You can also use the Family Filter to display only those components belonging to certain families. Next, you will add Propane to the component list using a Family Filter:

- 7. Ensure the **Match** field is empty, and click the **View Filter** button. The Filters view appears as shown on the left.
- 8. On the Filters view, check the **Use Filter** checkbox to activate the Family Filter.





9. Check the **Hydrocarbons** checkbox. The remaining components are known to be hydrocarbons.

Figure 2.11
On the Component Available in the Component Library Match View Filters Propene Propane appears near the top of the filtered list.

The Match feature remains active when you are using a family filter, so you could have also typed **C3** in the Match field and then added it to the component list.

To select consecutive components, use the **SHIFT** key.

To select nonconsecutive components, use the **CTRL** key. 10. Double-click Propane to add it to the component list.

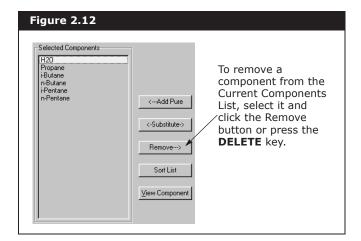
Next you will add the remaining Light Ends components i-C4 through n-C5. The following procedure shows you quick way to add components that appear consecutively in the library list.

11. Click on the first component to be added (in this case, i-C4).

12. Do **one** of the following:

- Hold down the SHIFT key and click on the last component, in this case n-C5. All components i-C4 through n-C5 are now selected. Release the SHIFT key.
- Click and drag from i-C4 to n-C5. Components i-C4 through to n-C5 are selected.

13. Click the **Add Pure** button. The selected components are transferred to the Selected Component group.



The complete list of non-oil components appears in the figure above.

14. Close the Component List View and Filters views to return to the Simulation Basis Manager view.

On the Components tab, the Component Lists group now contains the name of the new component list that you created.

# Defining a Fluid Package

In the Simulation Basis Manager view, your next task is to define a fluid package.

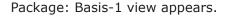
UniSim Design displays the current Environment and Mode in the upper right corner of the view. Whenever you begin a new case, you are automatically placed in the Basis environment, where you can choose the property package and non-oil components.

A fluid package contains the components and property methods that UniSim Design will use in its calculations for a particular flowsheet. Depending on what is required, a fluid package can also contain other information, such as a petroleum fluid characterization.

The fluid package for this example will contain the property package (Peng Robinson), the pure components H2O, C3, i-C4, n-C4, i-C5, n-C5, and the hypothetical components which are generated in the Oil characterization.

1. Click the Fluid Pkgs tab, then click the Add button. The Fluid

The Simulation Basis Manager allows you to create, modify, and otherwise manipulate Uwids packages in sour the left of the package With the case with table basis ple, would have be prised in the package finis/ Auropatic age by typillation hew name in the **Name** field at the bottom of the view.

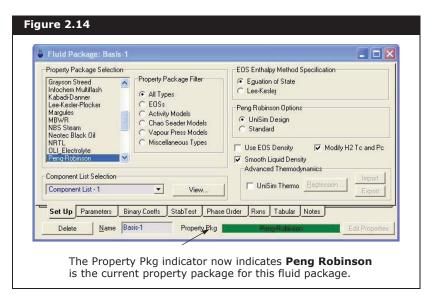


Property Package Selection Amine Pkg Antoine ASME Steam Braun K10 BW/RS Chao Seader Chao Seader Chao Seader Char Null Clean Fuels Pkg USD DBR Amine Package Esso Tabular Component List Selection Basis-1 Component List	Property Package Filter  All Types  EDSs  Activity Models  Chao Seader Models  Vapour Press Models  Electrolyte Models  Miscellaneous Types  View	Advanced Thermodynamics	Import Export
--	---	-------------------------	------------------

This view is divided into a number of tabs that allow you to supply all the information necessary to completely define the fluid package. For this tutorial, however, only the Set Up tab is used.

On the Set Up tab, the currently selected Property Package is <none>. Before you begin characterizing your petroleum fluid, you must choose a property package that can handle hypothetical components.

- 2. Select the Peng Robinson property package by doing **one** of the following:
  - Type **Peng Robinson**. UniSim Design finds the match to your input.
  - Use the up and down arrow keys to scroll through the list of available property packages until Peng Robinson is selected.
  - Use the vertical scroll bar to scroll through the list until Peng Robinson becomes visible, then click on it.



The Fluid Package: Basis - 1 view appears as shown below.

Alternatively, you could have selected the EOSs radio button in the Property Pkg Filter group. The list would then display only those property packages that are Equations of State. Peng Robinson would appear in this filtered list.

In the Component List Selection group, you could use the dropdown list to find the name of any component lists you had created (currently empty).

The View button opens the Component List View view of the selected component list.

If the selected component list contains components not appropriate for the selected property package, UniSim Design opens the Components Incompatible with Property Package view. On this view, you have the options of UniSim Design removing the incompatible components from the component list or changing to a different property package using the drop-down list or the Cancel button.

If you have multiple fluid packages and components lists in a case, you can use the drop-down list in the Component List Selection group to attache a component list to a particular property package.

2-14

3. Close the Fluid Package: Basis - 1 view to return to the Simulation Basis Manager view.

rrent Fluid Packages		Flowsheet - Fluid Pkg Ass	ociations	
asis-1 NC: 0 PP: Peng-Robinson	⊻iew	Flowsheet	Fluid Pkg To Use	
	Add	Case (Main)	Basis-1	
	Delete			
	Сору			
		<u>D</u> efault Fluid Pkg	Basis-1	
	Import			
	Export	Fluid Pkg for New Sub-Flo Use Default Fluid Pl		

The list in the Current Fluid Packages group displays the new fluid package, Basis-1, showing the number of components (NC) and property package (PP). The new fluid package is assigned by default to the main flowsheet, as shown in the Flowsheet-Fluid Pkg Associations group.

# Creating Hypocomponents

Your next task is to create and add the hypocomponents to the component list. In this example, you will characterize the oil (Petroleum Fluid) using the given Assay data to create the hypocomponents.

#### Characterizing the Oil

In this section, you will use the following laboratory Assay data:

Bulk Crude Properties				
MW	300.00			
API Gravity	48.75			

Light Ends Liquid V	olume Percent
i-Butane	0.19
n-Butane	0.11
i-Pentane	0.37
n-Pentane	0.46

TBP Distillation Assay					
Liquid Volume Percent Distilled	Temperature (°F)	Molecular Weight			
0.0	80.0	68.0			
10.0	255.0	119.0			
20.0	349.0	150.0			
30.0	430.0	182.0			
40.0	527.0	225.0			
50.0	635.0	282.0			
60.0	751.0	350.0			
70.0	915.0	456.0			
80.0	1095.0	585.0			
90.0	1277.0	713.0			
98.0	1410.0	838.0			

API Gravity Assay	
Liq Vol% Distilled	API Gravity
13.0	63.28
33.0	54.86
57.0	45.91
74.0	38.21
91.0	26.01

Viscosity Assay		
Liquid Volume Percent Distilled	Viscosity (cP) 100°F	Viscosity (cP) 210°F
10.0	0.20	0.10
30.0	0.75	0.30
50.0	4.20	0.80
70.0	39.00	7.50
90.0	600.00	122.30

# Accessing the Oil Environment

The UniSim Design Oil Characterization procedure is used to convert the laboratory data into petroleum hypocomponents.

The Associated Fluid Package drop-down list indicates which fluid package is used for the oil characterization. Since there is only one fluid package, UniSim Design has made **Basis-1** the Associated Fluid Package. 1. On the Simulation Basis Manager view, click the **Oil Manager** tab.

igure 2.16	5		
Simulation B Associated Flu Associated Flu	, ,	asis-1	
Associate	FlowSheet   Case (Main)	Fluid Pkg In Use Basis-1	To Enter the Oil environment, There must be a Fluid Package and the associated Property Package must be able to handle Hypo Components Enter Oil Environment
Components	s Fluid Pkgs Hyp	otheticals Oil Manager	Reactions Component Maps UserProperty

The text on the right side of the view indicates that before entering the Oil Environment, two criteria must be met:

- at least one fluid package must be present. In this case, only one fluid package, Basis-1, is selected.
- the property package must be able to handle Hypothetical Components. In our case, the property package is Peng Robinson, which is capable of handling Hypothetical components.

Since both criteria are satisfied, the oil is characterized in the Oil Environment.

- 2. To enter the Oil Characterization environment, do **one** of the following:
  - click the Enter Oil Environment button on the Oil Manager tab.
  - click the **Oil Environment** icon on the toolbar.



Oil Environment icon

The Oil Characterization view allows you to create, modify, and otherwise manipulate the Assays and Blends in your simulation case. For this example, the oil is characterized using a single Assay.

Oil Characterization		
Available Assa <u>y</u> s		Assay Information
	⊻iew	
	<u>A</u> dd	Description
	Delete	1
	Clone	
	Import	
	Export	] [
		Dij Input Preferences
Assay Cut/Blend L	Iser Property Correlati	ion Install Oil

The Oil Characterization view appears.

In general, three steps must be completed when you are characterizing a petroleum fluid:

- 1. Supply data to define the Assay.
- 2. Cut the Assay into hypothetical components by creating a Blend.
- 3. Install the hypothetical components into the fluid package.

#### Defining the Assay

1. On the **Assay** tab, click the **Add** button to create and view a new Assay. The Assay view appears.

When the property view for a new Assay is opened for the first time, the view contains minimal information. Depending on the Assay Data Type you choose, the view is modified appropriately. For this example, the Assay is defined based on TBP data.

UniSim Design has given the new Assay the default name of Assay-1. You can change this by typing a new name in the **Name** field. 2. From the Assay Data Type drop-down list, select TBP. The view is customized for TBP data.

Assay:Assay-1								_ 0
Assay Definition		Ine	ut <u>D</u> ata					
Bulk Properties	Not Used 💌				Assay B	asis M	ole	•
Assay Data Type	TBP				Assau	Percent	Temperati	ure
Light Ends	Ignore 💌						[C]	_
- Molecular Wt. Curve	Not Used 💌							
Density Curve	Not Used 💌							
Viscosity Curves	Not Used 💌							
TBP Distillation Cond	, _							
<ul> <li>Atmospheric</li> </ul>	C Vacuum							
Light Ends Handlin	g & Bulk Fitting Options		<u>E</u> dit Assay		A	t least 5 poir	nts are requi	red
	culation Defaults Wo	rking Cu	irves Plo	s Correl		lser Curves	Notes	

The next task is to enter the composition of the Light Ends in the Assay.

- 3. From the Light Ends drop-down list, select Input Composition.
- 4. In the Input Data group, select the **Light Ends** radio button.
- 5. Ensure that Liquid Volume% is selected in the Light Ends Basis drop-down list.
- 6. Click in the **Composition** cell for i-Butane.
- 7. Type 0.19, then press the **ENTER** key. You are automatically advanced down one cell to n-Butane.

8. Type the remaining compositions as shown. The total Percent of Light Ends in Assay is calculated and displayed at the bottom of the table.

Assay:Assay-1									
ssay Definition			_Input Dat	а —					
Bulk Properties	Not Used	•	<ul> <li>Light E</li> <li>Distilla</li> </ul>		Li	ight Ends B	asis	Liquid V	olume % 📃 💌
Assay Data Type	TBP		Distilia	uon		Light Ends	Compo	sition	NBP [F]
.ight Ends	Input Composition	-				H20	0.	.0000	212.0
dolecular Wt. Curve	Not Used	-				Propane	-	.0000	-43.78
		_				i-Butane		.1900	10.89
Density Curve	Not Used	•				i-Butane		1100	31.10
/iscosity Curves	Not Used	-				Pentane	-	.3700	82.18
,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,		-			<u>  · ·</u>	Pentane	U U	4600	96.91
TBP Distillation Condi	tions								
Atmospheric	C Vacuum								
		_							
Light Ends Handling	& Bulk Fitting Optic	ins	Perc	ent of Li	ght End	s in Assay	1	1300	
Input Data Calc	ulation Defaults	1.1.1.	ing Curves	Plots	Correl		Iser Curvi	es Not	

Before entering any of the assay data, you must activate the molecular weight, density, and viscosity curves by choosing appropriate curve types in the Assay Definition group. Currently, these three curves are not used.

- 9. From the Bulk Properties drop-down list, select Used. A new radio button labeled **Bulk Props** appears in the Input Data group.
- 10. From Molecular Wt. Curve drop-down list, select Dependent. A new radio button labeled **Molecular Wt** appears in the Input Data group.
- 11. From the Density Curve and Viscosity Curves drop-down lists, select Independent as the curve type. For Viscosity, two radio buttons appear as UniSim Design allows you to input viscosity assay data at two temperatures.

Your view now contains a total of seven radio buttons in the Input Data group. The laboratory data are input in the same order as the radio buttons appear.

In the next few sections, you will enter the following laboratory assay data:

- bulk molecular weight and density
- TBP Distillation assay data
- dependent molecular weight assay data
- independent density assay data
- independent viscosity assay data (at two temperatures)

Input Data C Bulk Props C Light Ends C Distillation C Molecular Wt C Density C Viscosity1 C Viscosity2

#### Entering Bulk Property Data

- 1. Select the **Bulk Props** radio button, and the bulk property table appears to the right of the radio buttons.
- Click in the Molecular Weight cell in the table. Type 300 and press ENTER. You are automatically advanced down one cell to the Standard Density cell.
- 3. In the **Standard Density** cell, enter 48.75 and press **SPACE BAR**. To the right of the cell, a field containing the current default unit associated with the cell appears. When you defined the new unit set, you specified the default unit for standard density as API\_60, which appears in the field.

Assay Data Type Light Ends Molecular Wt. Cu Density Curve Viscosity Curves TBP Distillation ( C Atmospheri		C Distantion C Molecular Wt C Density C Viscosity1 C Viscosity2	Watson UOPK Viscosity 1 Temp Viscosity 1 Temp Viscosity 1 Viscosity 2 Temp Viscosity 2	Cemptys Dynamic - 100.0 F (emptys 210.0 F (emptys)
Light Ends Har	dling & Bulk Fitting Options			

4. Since this is the correct unit, press **ENTER**, and UniSim Design accepts the density value.

No bulk Watson UOPK or Viscosity data is available for this assay. UniSim Design provides two default temperatures (100°F and 210°F) for entering bulk viscosity, but these temperature values are ignored unless corresponding viscosities are provided. Since the value for bulk viscosity is not supplied, there is no need to delete or change the temperature values.

#### Entering Boiling Temperature (TBP) Data

The next task is to enter the TBP distillation data.

- 1. Click the Calculation Defaults tab.
- 2. In the Extrapolation Methods group, select Lagrange for each method using the drop-down lists.
- 3. Return to the Input Data tab.

- 4. Select the **Distillation** radio button. The corresponding TBP data matrix appears. UniSim Design displays a message under the matrix, stating that 'At least 5 points are required' before the assay can be calculated.
- 5. From the Assay Basis drop-down list, select Liquid Volume.
- 6. Click the **Edit Assay** button. The Assay Input Table view appears.
- 7. Click in the top cell of the Assay Percent column.
- 8. Type **0** then press **ENTER**. You are automatically advanced to the corresponding empty Temperature cell.
- 9. Type **80** then press **ENTER**. You are automatically advanced down to the next empty Assay Percent cell.
- 10. Repeat steps #8 and #9 to enter the remaining Assay Percent and Temperature values as shown.

say Input Table ay Input Data	
Assay Percent [%]	Temperature [F]
0.0000	80.00
10.00	255.0
20.00	349.0
30.00	430.0
40.00	527.0
50.00	635.0
60.00	751.0
70.00	915.0
80.00	1095
90.00	1277
98.00	1410
<empty></empty>	<empty></empty>

11. Click the **OK** button to return to the Assay property view.

Entering Molecular Weight Data

- 1. Select the **Molecular Wt** radio button. The corresponding assay matrix appears. Since the Molecular Weight assay is Dependent, the Assay Percent column displays the same values as those you entered for the Boiling Temperature assay. Therefore, you need only enter the Molecular Weight value for each assay percent.
- 2. Click the **Edit Assay** button and the Assay Input Table view appears.
- 3. Click on the first empty cell in the Mole Wt column.
- 4. Type 68, then press the down arrow key.

5. Type the remaining Molecular Weight values as shown.

Assay Percent         Mole Wt.           [%]         Mole Wt.           [%]         0.0000         68.00           10.00         119.0         119.0           20.00         150.0         30.00         182.0           40.00         225.0         50.00         282.0           60.00         350.00         282.0         50.00         280.0           70.00         456.0         350.00         70.00         456.0         390.00         713.0         38.00         838.0	ssay Input Table	
Mole Wt.           0.0000         68.00           10.00         119.00           20.00         150.0           30.00         182.0           40.00         225.0           50.00         282.0           60.00         350.0           70.00         456.0           80.00         595.0           90.00         713.0	ay Input Data	
10.00         119.0           20.00         150.0           30.00         182.0           40.00         225.0           50.00         282.0           60.00         350.0           70.00         456.0           80.00         585.0           90.00         713.0		Mole Wt.
20.00         150.0           30.00         182.0           40.00         225.0           50.00         282.0           60.00         350.0           70.00         456.0           80.00         595.0           90.00         713.0	0.0000	68.00
30.00         182.0           40.00         225.0           50.00         282.0           60.00         350.0           70.00         456.0           80.00         595.0           90.00         713.0	10.00	119.0
40.00         225.0           50.00         282.0           60.00         360.0           70.00         456.0           80.00         585.0           90.00         713.0	20.00	150.0
50.00         282.0           60.00         350.0           70.00         456.0           80.00         585.0           90.00         713.0		
60.00         350.0           70.00         456.0           80.00         585.0           90.00         713.0		
70.00         456.0           80.00         585.0           90.00         713.0		
80.00 585.0 90.00 713.0		
90.00 713.0		
98.00 838.0		
	98.00	838.0

6. Click the **OK** button when you are finished.

Entering Density Data

- 1. Select the **Density** radio button. The corresponding assay matrix appears. Since the Density assay is Independent, you must input values in both the **Assay Percent** and **Density** cells.
- 2. Using the same method as for the previous assays, enter the API gravity curve data as shown here.

say Input Data	
Assay Percent [%]	Mass Density [API]
13.00	63.28
33.00	54.86
57.00	45.91
74.00	38.21
91.00	26.01
	distillation are on midpo

Entering Viscosity Data

1. Select the Viscosity 1 radio button. The corresponding assay

matrix appears.

- 2. In the Viscosity Type drop-down list above the assay matrix, ensure Dynamic is selected.
- 3. In the Viscosity Curves group, select the **Use Both** radio button. The Temperature field is for each of the two viscosity curves.
- 4. Input the Viscosity 1 assay data as shown here. This viscosity curve corresponds to Temperature 1, 100°F.

Input <u>D</u> ata			
<ul> <li>Bulk Props</li> <li>Light Ends</li> <li>Distillation</li> </ul>	Viscosity Type	ynamic _	•
C Molecular Wt C Density	Temperature 10	00.0 F	
<ul> <li>Viscosity1</li> <li>Viscosity2</li> </ul>	Assay Percent	Viscosity-1 [cP]	
	10.00	0.2000	_
Viscosity Curves	30.00	0.7500	
C Use Curve 1	50.00	4.200	
C Use Curve 2	70.00	39.00	
Use Both	90.00	600.0	_
			-

- 5. Select the **Viscosity 2** radio button.
- 6. Enter the assay data corresponding to Temperature 2, 210°F, as shown.

Fig	ure 2.26		
-4	Assay Input Data		
	Assay Percent [%]	Viscosity [cP]	
	10.00	0.1000	
	30.00	0.3000	
	50.00	0.8000	
	70.00	7.500	
	90.00	122.3	
	<empty></empty>	<empty></empty>	
			_

The Assay is now completely defined based on our available data.

 Click the **Calculate** button at the bottom of the Assay view. UniSim Design calculates the Assay, and the status message at the bottom of the view changes to Assay Was Calculated.

Click the **Edit Assay** button to access the Assay Input Table.

2-24

8. Click the Working Curves tab of the Assay property view to view

the calculated results. Figure 2.27 Assay:Assay-1 \_ 🗆 🗙 Assay Working Curves Viscosity 1 Mass Density Viscosity 2 Point # Moles Cum. Moles Mole Wt [F] [API] 92.93 [cP] 0.129 [cP] 77.18 0.061 0.00000 0.00000 96.91 0 0.01000 0.01000 92.12 105.3 89.96 85.58 0.135 1177 0.064 143.4 0.068

116.7

126.6 135.1

138.6

142.1 145.6

1491

152.7 166.4

170.3 178.8 187.6

0.01000

0.01000

0.01000

0.01000

0.01000

0.01000

0.02500 0.02500 0.02500 0.02500

Delete <u>N</u>ame: Assay-1

6

0.03000

0.04000

0.06000

0.07000

0.09000

0.10000 0.12500 0.15000 0.17500 0.20000 164.7

181.5 198.0 213.4 227.7 241.3

252.9

261.4 283.9

308.4 329.5 349.1

Input Data Calculation Defaults Working Curves Plots Correlations User Curves Notes

82.61 79.78 77.20

74.85

72.72

69.01 67.37 64.84

63.69 62.64 61.58 0.072 0.077 0.081

0.086

0.091

0.102

0.102 0.108 0.125 0.143

0.164

Calc<u>u</u>late

0.149

0.157

0.174

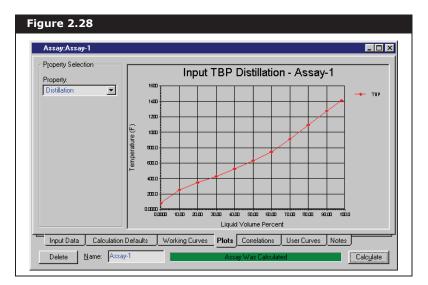
0.183

0.205

0.205 0.216 0.250 0.291 0.340 0.400

UniSim Design has calculated 50 points for each of the Assay Working Curves.

9. To view the Assay data you input in a graphical format, click the **Plots** tab. The input curve that appears is dependent on the current variable in the Property drop-down list. By default, UniSim Design plots the Distillation (TBP) data. This plot appears below.



The independent (x-axis) variable is the Assay percent, while the dependent variable is the TBP in °F. You can view any of the other input curves by selecting the appropriate variable in the Property drop-down list.

The plot view can be resized to make the plot more readable. To re-size the view, do **one** of the following:

- Click and drag the outside border to the new size.
- Click the **Maximize** icon.



Maximize icon

The remaining tabs in the Assay property view provide access to information which is not required for this tutorial.

10. Close the Assay view to return to the Oil Characterization view.

#### Cutting the Assay (Creating the Blend)

Now that the assay has been calculated, the next task is to cut the assay into individual petroleum hypocomponents.

- 1. Click the **Cut/Blend** tab of the Oil Characterization view.
- 2. Click the **Add** button. UniSim Design creates a new Blend and displays its property view.

Assess Calas Kan and C	11 La Carros - Maria		Cut Daward	
-Assay Selection and C Available Assays	Dil Flow Information	n	Cut Ranges	Auto Cut
Assay-1	Oil Flow Unit		Cut Option Selection	Auto Cut 💌
Moody-1		s now nate		
Add>	<bemo< th=""><th>ove</th><th></th><th></th></bemo<>	ove		
		Bulk <u>D</u> ata		

- 3. In the list of Available Assays, select Assay-1.
- 4. Click the **Add** button. There are two results:
  - The Assay is transferred to the Oil Flow Information table. (When you have only one Assay, there is no need to enter a Flow Rate in this table.)
  - A Blend (Cut) is automatically calculated based on the current Cut Option.

In this case, the Blend was calculated based on **Auto Cut**, the default **Cut Option**. UniSim Design calculated the Blend based on the following default values for the boiling point ranges and number of cuts per range:

- IBP to 800°F: 25°F per cut, generating [(800-IBP)/25] hypocomponents
- 800 to 1200°F: 50°F per cut, generating 8 hypocomponents

• 1200 to 1400°F: 100°F per cut, generating 2 hypocomponents The IBP, or initial boiling point, is the starting point for the first temperature range. The IBP is the normal boiling point (NBP) of the heaviest component in the Light Ends, in this case n-Pentane at 96.9°F. The first range results in the generation of (800-96.9)/25 = 28 hypocomponents. All the cut ranges together result in a total of 28+8+2 = 38 hypocomponents.

5. Click the **Tables** tab to view the calculated properties of these hypocomponents.

Blend: Blend-1						_	
Ta <u>b</u> le Type	-Component Physi	ical Propertie	\$				
Component Properties	Comp Name	NBP [F]	Mole Wt.	Density [API]	Viscosity1 [cP]	Viscosity2 [cP]	
Table Control	NBP_111	111.2	82.07	92.55	0.12785	5.9949e-00;	i—
Main Properties	NBP_136	136.0	89.45	5 91.24	0.13306	6.3050e-00;	1
C Other Properties	NBP_161	160.8	105.9	85.75	0.14056	6.7445e-00;	
Ojit:	NBP_185	184.9	122.4	81.14	0.15061	7.3233e-00;	
	NBP_210	210.4	137.4	76.89	0.16332	8.0375e-00;	
Blend-1	NBP_236	235.7	144.8	3 72.82	0.17922	8.9059e-00;	
	NBP_259	259.3	152.7	68.88	0.20002	0.10003	
	NBP_287	287.2	160.9	66.19	0.22056	0.11042	
	NBP_311	311.0	170.4	64.01	0.25354	0.12631	
	NBP_336	335.8	178.4	62.56	0.30509	0.14955	
	NBP_361	360.8	189.5	61.27	0.37361	0.17794	
	NBP_386	386.0	201.4		0.46242	0.21164	
	NRP 411	410.9	214 3	57 73	0 57674	0.25034	
Data Correlations Tai	bles Property Plot	Distribut	ion Plot	Composite Plot	Plot Summa	ry Notes	_

These components could be used in the simulation. Suppose, however, that you do not want to use the IBP as the starting point for the first temperature range. You could specify another starting point by changing the Cut Option to **User Ranges.** For illustration purposes, 100°F is used as the initial cut point.

- 6. Return to the **Data** tab.
- 7. From the Cut Option Selection drop-down list, select User Ranges. The Ranges Selection group appears.
- 8. In the **Starting Cut Point** field, enter 100°F. This is the starting point for the first range. The same values as the UniSim Design defaults are used for the other temperature ranges.
- In the Cut End point T column in the table, click on the top cell labeled <empty>. The value you will enter in this cell is the upper cut point temperature for the first range (and the lower cut point for the second range).
- 10. Type **800** then press **ENTER**.

Since the NBP of the heaviest Light Ends component is the starting point for the cut ranges, these hypocomponents were generated on a "lightends-free" basis. That is, the Light Ends are calculated separately and are not included in these hypocomponents. 11. Enter the remaining cut point temperatures and the number of cuts values as shown in the figure below.

Figure 2.31	
Cut Ranges Cut Option Selection User Ranges 💌 Ranges Selection	
Lower Temp Limit 96,906 F Upper Temp Limit 1446,828 F Starting Cut Point 100.000 F	
Cut End point T         Num. of Cuts           800.000         28           1200.000         8	
1400.000 2 <empty></empty>	

- 12. Once you have entered the data, click the **Submit** button to calculate the Blend based on the current initial cut point and range values. The message Blend Was Calculated appears in the status bar.
- 13. Click the **Tables** tab to view the properties of the petroleum hypocomponents.

Blend: Blend-1							
Ta <u>b</u> le Type	Component Phys		s				_
Component Properties 💌	Comp Name	NBP	Mole Wt.	Density	Viscosity1	Viscosity2	
		[F]		[API]	[cP]	[cP]	
Table Control	NBP_113	113.2	83.20	91.74	0.13132	6.2019e-00;	
Main Properties	NBP_139	138.6	96.04	88.73	0.13704	6.5387e-00;	
C Other Properties	NBP_164	163.6	110.8	84.10	0.14511	7.0080e-00;	
Oji:	NBP_188	187.6	125.6	80.08	0.15577	7.6155e-00:	
-	NBP_213	212.8	137.4	76.01	0.16913	8.3580e-00;	
Blend-1	NBP_238	238.2	143.4	72.13	0.18602	9.2685e-00;	
	NBP_261	260.8	150.9	68.23	0.20947	0.10485	
	NBP_289	289.1	162.6	65.36	0.23898	0.11942	1
	NBP_313	313.0	169.7	63.90	0.27790	0.13755	
	NBP_338	337.7	178.4	62.67	0.33524	0.16232	
	NBP_362	362.5	189.6	61.35	0.41217	0.19298	
	NBP_388	387.6	201.5	59.74	0.51187	0.22897	1
	NRP 412	4123	214.7	57 32	0.64025	0.26975	
Data Correlations Tab	es Property Plot	Distribut	ion Plot Ca	mposite Plot	Plot Summa	ry Notes	_
	ICS _ TOPERV FIOU			mposite Flot	<u></u>	in in in item	

Use the vertical scroll bar to view the components which are not currently visible in the **Component Physical Properties** table.

#### Viewing the Oil Distributions

1. To view the distribution data, select Oil Distributions from the Table

UniSim Design has provided the Initial Boiling Point (IBP) and Final Boiling Point (FBP). The IBP is the normal boiling point (NBP) of the heaviest component in the Light Ends (in this case, n-Pentane). The FBP is calculated by extrapolating the TBP Assay data to 100% distilled.

Blend: Blend-1							
Ta <u>b</u> le Type	-Cut Input Informat	ion		Cut Distributions			
Oil Distributions	Name	End T	1	Name	Begin T [F]	End T [F]	Fraction
Table Control	Off Gas	50.00		Lt St Run	100.3	158.0	0.033
asis: Liquid Volume 💌	Lt St Run	158.0		Naphtha	158.0	356.0	0.172
	Naphtha	356.0		Kerosene	356.0	464.0	0.129
Oil:	Kerosene	464.0		Light Diesel	464.0	554.0	0.089
Blend-1	Light Diesel	554.0		Heavy Diesel	554.0	644.0	0.083
biend-1	Heavy Diesel	644.0		Atm Gas Oil	644.0	698.0	0.049
	Atm Gas Oil	698.0		Residue	698.0	1441	0.444
	Residue	2192 💌					
	Straight Buni						
	C Cycle Oil						
	C Vacuum Oil						
	C User Custom						
Data Correlations Tabl	es Property Plot	Distributio	on Plot	Composite F	Plot Plot	Summary	Notes

Type drop-down list. The **Tables** tab is modified as shown below.

At the bottom of the Cut Input Information group, the Straight Run radio button is selected, and UniSim Design provides default TBP cut point temperatures for each Straight Run product. The Cut Distributions table shows the Fraction of each product in the Blend. Since Liquid Vol is the current Basis in the Table Control group, the products are listed according to liquid volume fraction.

These fractions can be used to estimate the product flow rates for the fractionation column. For example, the **Kerosene** liquid volume fraction is 0.129. With 100,000 bbl/day of crude feeding the tower, the Kerosene production is expected at **100,000 \* 0.129=12,900** or roughly 13,000 bbl/day.

If you want, you can investigate other reporting and plotting options by selecting another **Table Type** or by viewing information on the other tabs in the **Blend** property view.

 When you are finished, close the **Blend** view to return to the **Oil** Characterization view. Now that the Blend has been calculated, the next task is to install the oil.

#### Installing the Oil

The last step in the oil characterization procedure is to install the oil, which accomplishes the following:

- The petroleum hypocomponents are added to the fluid package.
- The calculated Light Ends and Oil composition are transferred to a material stream for use in the simulation.
- 1. On the **Oil Characterization** view, click the **Install Oil** tab.
- 2. In the Stream Name column, click in the top blank cell.

3. Type the name Preheat Crude, then press the **ENTER** key. UniSim Design creates a new stream named Preheat Crude in the flowsheet associated with the fluid package associated with this oil.

Oil Cha	racterization				
-Ojl Insta	I Information				
	Oil Name	Ready	Install	Stream Name	Flowsheet
	Blend-1			Preheat Crude	Case (Main) 🝸
Assay	Cut/Blend Us	er Property	Correlati	on Install Oil	

In this case, there is only one fluid package (Basis-1) and one flowsheet (the main flowsheet), so the stream is created in the main flowsheet. UniSim Design assigns the composition of the calculated oil and light ends to stream Preheat Crude. The properties of the new stream can be viewed from the Simulation environment.

The characterization procedure is now complete.

- 4. Return to the Basis environment by clicking the **Return to Basis Environment** icon.
- 5. Click the **Components** tab of the Simulation Basis Manager view.
- 6. Select Component List 1 from the list in the Component Lists group. Click the **View** button to open the component list property view.



7. The hypocomponents generated during the oil characterization procedure now appear in the Selected Components group.

Figure 2.35 Selected Components Hypothetical H20 components H20 Propane i-Butane i-Pentane n-Pentane NBP[0]113\* NBP[0]164\* NBP[0]213\* NBP[0]213\* NBP[0]261\* are indicated by a \* after the component <---Add Pure name. <-Substitute-> NBPI01289 NBP[0]313 Sort List NBPİDİ338 NBP NBP[0]388 NBP 0]412° 0]437° NRP  $\mathbf{T}$ 

#### Viewing Component Properties

To view the properties of one or more components, select the component(s) and click the **View Component** button. UniSim Design opens the property view(s) for the component(s) you selected.

- 1. In the Selected Components list, select H2O and NBP[0]113\*.
- 2. Click the **View Component** button. The property views for these two components appear.

Figure 2.36	_ [] X
NBP[0]113*	
-Component Identificati	on
Component Name	NBP[0]113*
Family / Class	Hydrocarbon
Chem Formula ID Number	10000
Group Name	Blend-1 Hypos
CAS Number	biend-r riypos
VNIFAC Structure	Structure Builder
User ID Tags	
	mber Tag Text mpty> Not Spec'd
ID Critical Poi	nt_TDep_UserProp
Estimate Unknown Props	Edit Properties Edit Visc Curve

See Chapter 4 -Hypotheticals in the UniSim Design Simulation Basis guide for more information on cloning library components.

The Component property view provides you with complete access to the component information. For pure components like H2O, the information is provided for viewing only. You cannot modify any parameters for a library (pure) component, however, UniSim Design

Press and hold the **CTRL** key to select more than one component.

allows you to clone a library component into a Hypothetical component, which you can then modify as required.

The petroleum hypocomponent shown here is an example of a hypothetical component. You can modify any of the parameters listed for this component. For this example, the properties of the hypothetical components generated during the oil characterization are not changed.

- 3. Close each of these two component property views.
- 4. The fluid package is now completely defined, so close the Component List view. The Simulation Basis Manager view should again be visible; if not, click the **Basis Manager** icon to access it.
- 5. Click the **Fluid Pkgs** tab to view a summary of the new fluid package.

imulation Basis Manager		()		
urrent Fluid Packages		Flowsheet - Fluid Pkg As	sociations	
Basis-1 NC: 44 PP: Peng-Robinson	View	Flowsheet Fluid Pkg To Use		
	Add	Case (Main)	Basis-1	
	Delete			
	Сору			
		<u>D</u> efault Fluid Pkg	Basis-1	•
	Import		01 .	
	Export	Fluid Pkg for New Sub-Fl Use Default Fluid F Use Parent's Fluid I	'kg	

The list of Current Fluid Packages displays the new fluid package, Basis-1, showing the number of components (NC) and property package (PP). The fluid package contains a total of 44 components:

- 6 library (pure) components (H2O plus five Light Ends components)
- 38 petroleum hypocomponents

The new fluid package is assigned by default to the Main Flowsheet, as shown in the **Flowsheet-Fluid Pkg Associations** group. Next you will install streams and operations in the Main Simulation environment.

# 2.2.4 Entering the Simulation Environment

Enter Simulation

Environment icon

1. To leave the Basis environment and enter the Simulation



Basis Manager icon

environment, do **one** of the following:

- Click the **Enter Simulation Environment** button on the Simulation Basis Manager view.
- Click the Enter Simulation Environment icon.

When you enter the Simulation Environment, the initial view that appears depends on your current preference setting for the Initial Build Home View.

Three initial views are available: **PFD**, **Workbook**, and **Summary**. Any or all of these can be displayed at any time, however, when you first enter the Simulation Environment, only one appears. For this example, open the **Workbook under the Tools menu or by pressing CTRL W**.

	eet Workbook Tools Win					
D 🛎 🖬 🖨 🛛 🕻 🖷	) A 😫 🚍 🗢 🛛	× 👁 👁	4	Enviro	Mode	->
• Workbook - Case (Mai	n)					20
Name	Preheat Crude **	New **			6	¥∰=
Vapour Fraction	<empty></empty>					
Temperature [C]	<empty></empty>					
Pressure [kPa]	<empty></empty>					FRE
Molar Flow [kgmole/h]	<empty></empty>				24	
Mass Flow [kg/h]	<empty></empty>	a la companya da			n A	
Liquid Volume Flow [m3/h] Heat Flow [kJ/h]	<empty> <empty></empty></empty>				SG Y S	
rieat riuw (ka/rij	cemptys					84
						\$ <del>1</del> - €
	Strawersterner analysis				-n n	H 100
						Be a. or
Material Streams Com	positions Energy Streams	Unit Ops				35 19 10
				a second s		1+1
ProductBlock_Preheat Crude FeederBlock_Preheat Crude			Fluid Pk	g All	- 7	
Feederblock_Preheat Lrude				lude Sub-Flowsheets	6. A	• 🖬
				w Name Only	- <b>E</b> - E	
Horizontal Matrix				of Hidden Objects:	0 -6 6	
					-66	
					<b>6</b> <sup>22</sup> -fi	-
					<b>B</b>	
					FLOW	
					SHEET	[LULUTITE]
Optional Info : Preheat Crude						
Optional Info : Preheat Crude						
					→8 =	
Optional Info : Preheat Crude Optional Info : Preheat Crude	Unknown Flow Bate					+

There are several things to note about the Main Simulation Environment. In the upper right corner, the **Environment** has changed from **Basis** to **Case (Main)**. A number of new items are now available on the menu and toolbar, and the **Workbook** and **Object Palette** are open on the Desktop. These latter two objects are described below.

Objects	Description
Workbook	A multiple-tab view containing information regarding the objects (streams and unit operations) in the simulation case. By default, the Workbook has four tabs, namely Material Streams, Compositions, Energy Streams and Unit Ops. You can edit the Workbook by adding or deleting tabs, and changing the information displayed on any tab.
Object Palette	A floating palette of buttons which can be used to add streams and unit operations.

Also notice that the name of the stream (Preheat Crude) you created during the Oil characterization procedure appears in the Workbook, and the white Object Status window at the very bottom of the environment view shows that the stream has an unknown pressure. As you specify the conditions of Preheat Crude, the message displayed in the Object Status window is updated appropriately. Before specifying the feed conditions, you can view the stream composition, which was calculated by the Oil characterization.

# Viewing the Feed Composition

1. In the Workbook, click the **Compositions** tab to view the composition of the streams.

Name	L.	Preheat Crude	**	lew **		
Comp Mole Frac (H2O)	Ĩ	0.0000				
Comp Mole Frac (Propar	ne)	0.0000				
Comp Mole Frac (i-Butar	ie)	0.0070				
Comp Mole Frac (n-Buta	ne)	0.0042				
Comp Mole Frac (i-Penta		0.0122				
Comp Mole Frac (n-Pent		0.0154				
Comp Mole Frac (NBP[0		0.0215				
Comp Mole Frac (NBP[0		0.0262				
Comp Mole Frac (NBP[0		0.0294				
Comp Mole Frac (NBP[0		0.0307				
Comp Mole Frac (NBP[0		0.0315				
Comp Mole Frac (NBP[0	]238*)	0.0350				
Material Streams	Compos	sitions Energy	Streams	Unit O	ps	
ProductBlock Preheat	Crude		Fluid Pk	a Al		-

The Light Ends and petroleum hypocomponents are listed by Mole Fraction. To view the components which are not currently visible, use the up and down arrow keys or the vertical scroll bar to advance down the component list.

Before proceeding any further to install streams or unit operations, save your case.

You can toggle the palette open or closed by pressing **F4**, or by selecting Open/Close Object Palette from the Flowsheet menu.

#### **Refining Tutorial**

Save icon

If you enter a name that already exists in the current directory, UniSim Design ask you for confirmation before over-writing the existing file.



Workbook icon

2. Do one of the following:

- Click the **Save** icon on the toolbar.
- Select **Save** from the **File** menu.
- Press CTRL S.

If this is the first time you have saved your case, the Save Simulation Case As view appears. By default, the File Path is the cases sub-directory in your UniSim Design directory.

- 3. In the **File Name** field, type a name for the case, for example REFINING. You do not have to enter the \*.usc extension; UniSim Design adds it automatically.
- 4. Once you have entered a file name, press the ENTER key and UniSim Design saves the case under the name you gave it. The Save As view does not appear again unless you choose to give it a new name using the Save As command.

# 2.2.5 Using the Workbook

Click the Workbook icon on the toolbar to ensure the **Workbook view** is active.

# Specifying the Feed Conditions

In general, the first task in the Simulation environment is to install one or more feed streams, however, the stream Preheat Crude was already installed during the oil characterization procedure. At this point, your current location should be the **Compositions** tab of the **Workbook** view.

- 1. Click the **Material Streams** tab. The preheated crude enters the pre-fractionation train at 450°F and 75 psia.
- 2. In the Preheat Crude stream, click in the **Temperature** cell and type 450. UniSim Design displays the default units for temperature, in this case °F.

gure 2.40			
Workbook - Case (	Main)		
Name	Preheat Crude	** New **	
Vapour Fraction	<empty></empty>		
vapour Fraction			
Temperature [F]	450 F	-	

3. Since this is the correct unit, press the **ENTER** key. UniSim Design accepts the temperature. UniSim Design advances to the **Pressure** cell.

If you know the stream pressure in another unit besides the default of psia, UniSim Design will accept your input in any one of a number

When you press **ENTER** after entering a stream property, you are advanced down one cell in the Workbook only if the cell below is <empty>. Otherwise, the active cell remains in its current location. of different units and automatically convert the value to the default. For example, the pressure of Preheat Crude is 5.171 bar, but the default units are psia.

- 4. In the **Pressure** cell, type 5.171.
- 5. Press **SPACE BAR**. The field containing the active cell units becomes active.
- 6. Begin typing 'bar'. The field opens a drop-down list and scrolls to the unit(s) most closely matching your input.

igure 2.41			
Workbook - Case (Main)			_ 🗆 ×
Name	Preheat Crude	** New **	•
Vapour Fraction	0.3768		
Temperature [F]	450.0		
Pressure [psia]	5.171	bar 💌	
Molar Flow [lbmole/hr]	<empty></empty>	bar 🔺	
Mass Flow [lb/hr]	<empty></empty>	N/m2	
Liquid Volume Flow [barrel/day]	<empty></empty>	atm —	
Heat Flow [Btu/hr]	<empty></empty>	at kg/cm2	
		kg/cm2 psia	
		bf/ft2	
			-

- 7. Once 'bar' is selected, press the **ENTER** key. UniSim Design accepts the pressure and automatically converts to the default unit, psia.
- Click in the Liquid Volume Flow cell, then type 1e5. The stream flow is entered on a volumetric basis, in this case 100,000 barrel/ day.
- 9. Press the **ENTER** key.

The stream is now completely defined, so UniSim Design flashes it at the conditions given to determine the remaining properties. The properties of Preheat Crude are shown below. The values you specified are a different colour (blue) than the calculated values (black).

Workbook - Case (Main)						
Name	Prehe	at Crude	**	New **		 _
Vapour Fraction		0.1292				
Temperature [F]		450.0				
Pressure (psia)	75.00					
Molar Flow [lbmole/hr]		3814				
Mass Flow [lb/hr]	1.144e+006					Т
Liquid Volume Flow [barrel/day]	1.000e+005					
Heat Flow [Btu/hr]	-7.61	9e+008				
						 _
Material Streams Comp	ositions	Energy	Streams	Unit 0	)os 🗌	 _

The next task is to install and define the utility steam streams that will be attached to the fractionation tower later.

Alternately, you can specify the unit simply by selecting the unit in the drop-down list.

If UniSim Design does not flash the stream, ensure that the Solver Active icon in the tool bar is selected.



Solver Active icon

# Installing the Utility Steam Streams

- 1. On the **Material Streams** tab, click in the header cell labeled **\*\*New\*\***.
- 2. Type the new stream name Bottom Steam, then press **ENTER**. UniSim Design creates the new stream.
- 3. In the **Temperature** cell, enter 375°F.
- 4. In the **Pressure** cell, enter 150 psia.

Workbook - Case (Main)	Preheat Crude	Bottom Steam	** New **
Vapour Fraction	0.1292	<empty></empty>	HOW
Temperature [F]	450.0	375.0	
Pressure [psia]	75.00	150.0	
Molar Flow [lbmole/hr]	3814	<empty></empty>	
Mass Flow [lb/hr]	1.144e+006	<empty></empty>	
Liquid Volume Flow [barrel/day]	1.000e+005	<empty></empty>	
Heat Flow [Btu/hr]	-7.619e+008	<empty></empty>	
Material Streams Comp	ositions Energy	Streams Unit O	

- 5. In the Mass Flow cell, enter 7500 lb/hr.
- 6. Create a new utility stream called Diesel Steam.
- 7. Define the conditions of this stream as follows:
  - Temperature 300°F
  - Pressure 50 psia
  - Mass Flow 3000 lb/hr.

The Workbook view appears as shown below.

Name	Preheat Crude	Bottom Steam	Diesel Steam	** New **
Vapour Fraction	0.0989	<empty></empty>	<empty></empty>	
Temperature [F]	450.0	375.0	300.0	
Pressure [psia]	75.00	150.0	50.00	
Molar Flow [MMSCFD]	34.73	<empty></empty>	<empty></empty>	
Mass Flow [lb/hr]	1.144e+006	7100	3000	
Liquid Volume Flow [barrel/day]	1.000e+005	<empty></empty>	<empty></empty>	
Heat Flow [Btu/hr]	-7.653e+008	<empty></empty>	<empty></empty>	
Material Streams Compu- ProductBlock Preheat Crude FeederBlock Preheat Crude	ositions _ Energy		Ips Fluid Pkg All Include Sub-Fl Show Name O Number of Hidden	inly

### Providing Compositional Input

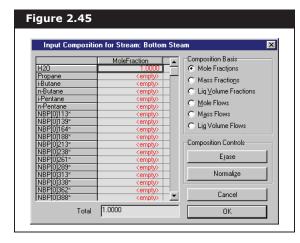
Now that the utility stream conditions have been specified, the next

UniSim Design accepts blank spaces within a stream or operation name. task is to input the compositions.

- 1. Click the **Compositions** tab in the Workbook. The components are listed by Mole Fraction by default.
- 2. In the Bottom Steam column, click in the input cell for the first component, H2O.
- 3. Since the stream is all water, type 1 for the H2O mole fraction, then press **ENTER**.

The Input Composition for Stream view appears, allowing you to complete the compositional input.

The Input Composition for Stream view is Modal, indicated by the absence of the Minimize/Maximize icons in the upper right corner.



When a Modal view is visible, you are unable to move outside the view until you are finish with it, by clicking either the Cancel or OK button.

The Input Composition for Stream view allows you to specify a stream composition quickly and easily. The following table lists and

#### describes the features available on this view:

Features	Description
Compositional Basis Radio Buttons	You can input the stream composition in some fractional basis other than Mole Fraction, or by component flows, by selecting the appropriate radio button before providing your input.
Normalizing	The Normalizing feature is useful when you know the relative ratios of components (2 parts N2, 2 parts CO2, etc.) Rather than manually converting these ratios to fractions summing to one, enter the numbers of parts for each component and click the Normalize button. UniSim Design computes the individual fractions to total 1.0.
	Normalizing is also useful when you have a stream consisting of only a few components. Instead of specifying zero fractions (or flows) for the other components, enter the fractions (or the actual flows) for the non-zero components, leaving the others <empty>. Click the Normalize button, and UniSim Design forces the other component fractions to zero.</empty>
Calculation status/colour	As you input the composition, the component fractions (or flows) initially appear in red, indicating the final composition is unknown. These values become blue when the composition has been calculated. Three scenarios result in the stream composition being calculated:
	<ul> <li>Input the fractions of all components, including any zero components, such that their total is exactly 1.0000, then click the OK button.</li> <li>Input the fractions (totalling 1.000), flows or relative number of parts of all non-zero components, then click the Normalize button then the OK button.</li> <li>Input the flows or relative number of parts of all components, including any zero components, then click the OK button.</li> </ul>

This stream is pure water, therefore, there is no need to enter fractions for any other components.

- 4. Click the **Normalize** button and all other component fractions are forced to zero.
- 5. Click the **OK** button. UniSim Design accepts the composition and you are returned to the Workbook view.

The stream is now completely defined, so UniSim Design flashes it at the conditions given to determine the remaining properties.

6. Repeat steps #2 to #5 for the other utility stream, Diesel Steam.

These are the default colours; yours can appear differently depending on your settings on the Colours page of the Session Preferences view. If you want to delete a stream, move to the Name cell for the stream, then press **DELETE**. UniSim Design ask for confirmation of your action.

7. Click the **Material Streams** tab. The calculated properties of the two utility streams appear here.

Name	Preheat Crude	Bottom Steam	Diesel Steam	** New **
/apour Fraction	0.0989	1.0000	1.0000	
emperature [F]	450.0	375.0	300.0	
Pressure (psia)	75.00	150.0	50.00	
Volar Flow [MMSCFD]	34.73	3.589	1.517	
Mass Flow [lb/hr]	1.144e+006	7100	3000	
iquid Volume Flow [barrel/day].	1.000e+005	487.1	205.8	
Heat Flow [Btu/hr]	-7.653e+008	-3.997e+007	-1.697e+007	
Material Streams Com	i positions Energy	Streams Unit C	)ps	
ProductBlock_Preheat Crude FeederBlock_Preheat Crude			Fluid Pkg All	

Next, you will learn alternative methods for creating a new stream.

- 8. To add the third utility stream, do any **one** of the following:
  - Press **F11**.
  - From the **Flowsheet** menu, select **Add Stream**.
  - Double-click the **Material Stream** icon on the Object Palette.
  - Click the **Material Stream** icon on the Object Palette, then click on the Palette's **Add Object** icon.

Each of these four methods displays the property view for the new stream, which is named according to the **Auto Naming** setting in your **Preferences**. The default setting names new material streams with numbers, starting at 1, and energy streams starting at **Q-100**.

- 9. In the stream property view, click in the **Stream Name** cell and rename the stream **AGO Steam**.
- 10. Press enter.
- 11. In the **Temperature** cell, enter **300**.



Material Stream icon



Add Object icon

Both of the temperature and pressure parameters are in the default units, so you do not need to change the unit with the values.

Do not enter a flow, it is entered through the **Composition** page.

2-40

12. In the **Pressure** cell, enter **50**.

AGO Steam		
Worksheet	Stream Name	AGO Stea
Conditions	Vapour / Phase Fraction	<empty< th=""></empty<>
Properties	Temperature [F] Pressure [psia]	300.0
	Molar Flow [lbmole/hr]	<empty <empty< td=""></empty<></empty 
Composition	Mass Flow [lb/hr]	<empty <empty< td=""></empty<></empty 
K Value	Std Ideal Lig Vol Flow [barrel/day]	<empty <empty< td=""></empty<></empty 
User Variables	Molar Enthalpy [Btu/Ibmole]	<empty <empty< td=""></empty<></empty 
	Molar Entropy [Btu/Ibmole-F]	<empty< td=""></empty<>
Notes	Heat Flow [Btu/hr]	<empty< td=""></empty<>
Cost Parameters	Liq Vol Flow @Std Cond [barrel/day]	<empty< th=""></empty<>
	Fluid Package	Basis-1
= Worksheet 🔺	ttachments Dynamics	
	Unknown Compositions	

13. Select the **Composition** page to begin the compositional input for the new stream.

Worksheet		Mole Fractions
	H20	<pre><empty></empty></pre>
Conditions	Propane	<empty> &lt;</empty>
Properties	i-Butane	<empty></empty>
•	n-Butane	<empty></empty>
Composition	i-Pentane	<empty></empty>
< Value	n-Pentane	<empty></empty>
< value	NBP[0]113*	<empty></empty>
Jser Variables	NBP[0]139*	<empty></empty>
Notes	NBP[0]164*	<empty></empty>
Notes	NBP[0]188*	<empty></empty>
	Total UU Edit Edit Prope	nties Basis
Worksheet	ttachments Dynamics	
	Unknown Compositions	
Delete	Define from Other Stream	

- 14. Click the **Edit** button. The Input Composition for Stream view appears.
- 15. In the Composition Basis group, select the **Mass Flows** radio button.
- 16. Click in the compositional cell for H2O.
- 17. Type 2500 for the steam mass flow, then press **ENTER**. As there are no other components in this stream, the compositional input is complete.

The current Composition Basis setting is set to the Preferences Default of Mole Fractions. The stream composition is entered on a mass basis. Since only H2O contain any significant value, UniSim Design automatically forces all other components' value to be zero. 18. Click the **OK** button to close the view and return to the stream property view.

AGO Steam		
Worksheet		Mass Flows
Conditions	H20 Propane	2500.0
D	i-Butane	0.00000
Properties	n-Butane	0.00000
Composition	i-Pentane	0.00000
K Value	n-Pentane	0.00000
	NBP[0]113* NBP[0]139*	0.00000
User Variables	NBP[0]164*	0.00000
Notes		•
Cost Parameters	Total	2500.00000 lb/hr roperties Basis
Worksheet	ttachments Dynamics	

UniSim Design performs a flash calculation to determine the unknown properties of AGO Steam, as shown by the status indicator displaying 'OK'. You can view the properties of each phase using the horizontal scroll bar in the matrix or by re-sizing the property view. In this case, the stream is superheated vapour, so no Liquid phase exists and the Vapour phase is identical to the overall phase. To view the vapour compositions for AGO Steam, scroll to the right by clicking the right scroll arrow, or by click and dragging the scroll button.

The compositions are currently displayed by Mass Flows. You can change this by clicking the Basis button and choosing another Composition Basis radio button.

19. Close the AGO Steam property view.

# 2.2.6 Installing Unit Operations

Now that the feed and utility streams are known, the next task is to install the necessary unit operations for processing the crude oil.

## Installing the Separator

The first operation is a Separator, used to split the feed stream into its liquid and vapour phases. As with most commands in UniSim Design, installing an operation can be accomplished in a number of ways. One



Workbook icon

method is through the Unit Ops tab of the Workbook.

- 1. Click the **Workbook** icon to ensure the Workbook is the active view.
- 2. Move to the **Unit Ops** tab.
- 3. Click the **Add UnitOp** button. The UnitOps view appears, listing all available unit operations.
- 4. In the Categories group, select the **Vessels** radio button. UniSim Design produces a filtered list of unit operations, showing only those in the current category.

UnitOps - Case (Main)		>
Categories Categories Vesels Heat Transfer Equipment Rotating Equipment Piping Equipment Solds Handling Reactors Prebuit Columns Short Cut Columns Short Cut Columns Categories Logicals Extensions User Ops Electrolyte Equipment Refinere Ops	Agailable Unit Operations 3 Phase Separator Cont. Stirred Tank Reactor Conversion Reactor Gibbs Reactor Separator Tank	<u>A</u> dd <u>C</u> ancel

- 5. Add the separator by doing **one** of the following:
  - Select Separator in the list of Available Unit Operations, and click the Add button or the ENTER key.
  - Double-click on Separator.

The property view for the **separator** appears in the figure below.

V-100		
Design	<u>N</u> ame V-100	
<b>Connections</b> Parameters User Variables Notes	Injets           <	Vapour Outlet
	Energy (Optional)	
	Fluid Package Basis-1	Liquid O <u>u</u> tlet
Design Read	ions Rating Worksheet Dynamics	

UniSim Design provides the default name **V-100** for the **separator**. The default naming scheme for unit operations can be changed in your Session Preferences.

Alternatively, you could have made the connection by typing the exact stream name in the cell, and pressing ENTER. A unit operation property view contains all the information defining the operation, organized into tabs and pages. The Design, Rating, and Worksheet tabs appear for most operations. Property views for more complex operations contain more tabs.

Many operations, like the separator, accept multiple feed streams. Whenever you see a matrix like the one in the **Inlets group**, the operation accepts multiple stream connections at that location. When the matrix is active, you can access a drop-down list of available streams.

- 6. Click in the **Name** field, type **PreFlash**, then press **enter**. The status indicator at the bottom of the view shows that the operation requires a feed stream.
- 7. In the Inlets matrix, click in the <**Stream>**> cell.
- 8. Click the down arrow 🖃 to open the drop-down list of available streams.
- Select Preheat Crude from the list. Preheat Crude appears in the Inlets matrix, and the <<Stream>> label is automatically moved down to a new empty cell. The status indicator now displays 'Requires a product stream'.

PreFlash	_		
Design	<u>N</u> ame PreFlash		
Connections	Injets Preheat Crude	Vapour Outlet	
Parameters User Variables	<< Stream >> <		•
Notes			
	Energy (Optional)		
	Fluid Package Basis-1	Liquid O <u>u</u> tlet	-
Design Read	tions Rating Worksheet Dynamic:		

- 10. Click in the **Vapour Outlet** field, or press **tab** to move to the field.
- 11. Type **PreFlashVap** in the field, then press **enter**. This stream does not yet exist, so UniSim Design creates this new stream.

12. Click in the Liquid Outlet field and type PreFlashLiq. UniSim Design creates another new stream.

PreFlash			
Design	<u>N</u> ame PreFlash		
<b>Connections</b> Parameters User Variables Notes	Injets Preheat Crude Control Crude	Vapour Outlet PreFlashVap	
	Energy (Optional)	Liquid O <u>u</u> tlet PreFlashLiq	
Design React	ions Rating Worksheet Dynamics		

An **Energy** stream could be attached to heat or cool the vessel contents, however, for the purposes of this example, the energy stream is not required.

Since there is no energy stream attached to the **separator**, no **Optional Heat Transfer** information is required. The status indicator displays a green **OK message**, showing that the operation and attached streams are completely calculated.

13. Select the **Parameters** page. The default **Delta P** (pressure drop) of zero is acceptable for this example. The **Liquid Level** is also acceptable at its default value.

igure 2.54		
PreFlash		
Design Connections Parameters User Variables Notes	Delta P Inlet [psi] [ 0.0000 Vapour outlet [psi] 0.0000 Volume Liquid Volume Liquid Level 50.00 %	
Design Reacti	Type Type Segarator C 3 Phase Sep C Tank Dynamics	]
Delete	OK	Ignored

14. To view the calculated outlet streams, click the **Worksheet** tab. This is a condensed Workbook displaying only those streams attached to the operation.

Worksheet	Name	Preheat Crude	PreFlashLig	PreFlashVap
Conditions	Vapour	0.1292	0.0000	1.0000
Londitions	Temperature [F]	450.0	450.0	450.0
Properties	Pressure [psia]	75.00	75.00	75.00
Composition	Molar Flow [lbmole/hr]	3814	3321	492.9
	Mass Flow [lb/hr]	1.144e+006	1.080e+006	6.454e+004
PF Specs	Std Ideal Liq Vol Flow [barrel/day]	1.000e+005	9.372e+004	6282
	Molar Enthalpy [Btu/Ibmole]	-1.998e+005	-2.175e+005	-8.049e+004
	Molar Entropy [Btu/Ibmole-F]	266.0	288.8	112.4
	Heat Flow [Btu/hr]	-7.619e+008	-7.222e+008	-3.967e+007

15. Now that the **separator** is completely known, close the PreFlash view and the UnitOps view, and return to the **Workbook view**. The new separator appears on the **Unit Ops** tab.

Name	Object Type	Inlet	Outlet	Ignored	Calc. Leve
PreFlash	Separator	Preheat Crude	PreFlashLiq PreFlashVap		500
<u>V</u> iew UnitOp	<u>A</u> dd UnitOp			<u>D</u> elete	e UnitOp

The matrix shows the operation **Name**, its **Object Type**, the attached streams (**Inlet** and **Outlet**), whether it is **Ignored**, and its **Calculation Level**.

#### Optional Methods for Accessing Property Views

When you click the View UnitOp button, the property view for the operation occupying the active row in the matrix opens. Alternatively, by double-clicking on any cell (except **Inlet** and **Outlet**) associated with the operation, you also open its property view.

You can also open the property view for a stream directly from the Unit Ops tab of the Workbook. When any of the **Name**, **Object Type**, **Ignored** or **Calc**. Level cells are active, the display field at the bottom of the view displays all streams attached to the current operation. Currently, the **Name** cell for **PreFlash** is active, and the display field displays the three streams attached to this operation. To open the property view for one of the streams attached to the **separator** (such as **Preheat Crude**), do **one** of the following:

- Double-click on Preheat Crude in the display field at the bottom of the view.
- Double-click on the **Inlet** cell for PreFlash. The property view for the first listed feed stream opens. In this case, Preheat Crude is the only feed stream, so its property view also opens.

# 2.2.7 Using Workbook Features

Before you install the remaining operations, you will examine a number of Workbook features that allow you to access information quickly and change how information appears.

#### Accessing Unit Operations from the Workbook

There are a number of ways to open the property view for an operation directly from the **Workbook** besides using the **Unit Ops** tab.

When your current location is a Workbook streams tab (Material Streams, Compositions, and Energy Streams tabs), the field at the bottom of the Workbook view displays the operations to which the current stream is attached. In this display field, you can click on any cell associated with the stream.

For example, if you click in any cell for Preheat Crude, the field displays the name of the operation, PreFlash, to which this stream is attached. The display field also displays FeederBlock\_Preheat Crude, because the Preheat Crude stream is a boundary stream. To access the property view for the PreFlash operation, double-click on PreFlash. The operation

Return to the Material Streams tab of the Workbook.

Any utilities attached to the stream with the Workbook active are also displayed in (and are accessible through) this display field. property view appears.

Stream Preheat Crude is the current Workbook location.							
	Workbook - Case (Main)						×
	Name	Prehea	at Crude-	Bottom	Steam	Diesel Steam	
	Vapour Fraction		0.1292		1.0000	1.0000	
	Temperature [F]		450.0		375.0	300.0	-
	Pressure (psia)		75.00		150.0	50.00	
	Molar Flow [lbmole/hr]		3814		416.3	166.5	
	Mass Flow [lb/hr]	1.14	4e+006		7500	3000	
	Liquid Volume Flow [barrel/day]	1.00	l0e+005		514.6	205.8	
	Heat Flow [Btu/hr]	-7.61	9e+008	-4.222	e+007	-1.697e+007	
	Name	AGO	) Steam	PreFla	shVap	PreFlashLiq	- 1
	Material Streams Compo	ositions	Energy	Streams	Unit Ops		ש
					lude Sub-Fl	/ www.heate	
	FeederBlock_Preheat Crude PreFlash				ow Name D		
				_	r of Hidden I	-	
	-			Numbe	i ul midden i	objects. 0	
		< _					
ie ope	ration to which Pre	heat	Cruc	le is a	ttache	d appear	's in t

### Adding a Tab to the Workbook

When the Workbook is active, the Workbook item appears in the UniSim Design menu bar. This item allows you to customize the Workbook.

In this section, you will create a new Workbook tab that displays only stream pressure, temperature, and flow.

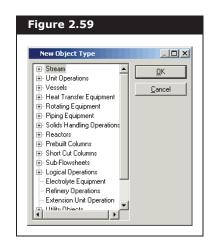
- 1. Do one of the following:
  - From the **Workbook** menu, select **Setup**.
  - Object inspect (right-click) the Material Streams tab in the Workbook, then select **Setup** from the menu that appears.

The Workbook Setup view appears.

Figure 2.58			
Setup Workbook Tabs Add Compositions Energy Streams Unit Dps Delete	Tab Contents Object Name: Material Streams Type: Material Stream Variables Variables Variable Forma Variable 4 sig1 Molar Flow 4 sig1 Mass Flow 4 sig1 Heat Flow 4 sig1 Heat Flow 4 sig1	ig <u>A</u> dd ig <u>De</u> lete ig <u>Format</u>	Currently, all variables appear with four significant figures. You can change the display format or precision of any Workbook variables by clicking the Format button.

The four existing tabs are listed in the **Workbook Tabs** area. When you add a new tab, it is inserted *before* the selected tab (currently **Material Streams**). You will insert the new tab before the **Compositions** tab.

- 2. In the Workbook Tabs group list, select **Compositions**.
- 3. Click the Add button. The New Object Type view appears.



- Click the + beside Stream, select Material Stream from the branch, then click the OK button. You return to the Setup view, and the new tab appears after the existing Material Streams tab.
- 5. In the Tab Contents Object group, click in the Name field.
- 6. Change the name of the new tab to **P,T,Flow** to better describe the tab contents.

ab Contents		
Dbject		
Name: P.T.Flow		<u>0</u> rder
Type: Material S	: Material Stream	
<u>/</u> ariables		
Variable Vapour Fraction	Format 1.4 fixed	<u>U</u> se Set
- Variable Vapour Fraction Temperature	1.4 fixed 4 sig fig	<u>U</u> se Set <u>A</u> dd
- Variable Vapour Fraction	1.4 fixed	
- Variable Vapour Fraction Temperature Pressure	1.4 fixed 4 sig fig 4 sig fig	

The next task is to customize the tab by removing the variables that are not required.

- 7. In the Variables group, click on the first variable, **Vapour Fraction**.
- 8. Press and hold the **CTRL** key.
- 9. Click on the other variables, **Molar Flow**, **Mass Flow**, **Heat Flow**, and Molar Enthalpy. These four variables are now highlighted.
- 10. Release the **ctrl** key.

If you want to remove variables from another tab, you must edit each tab individually. 11. Click the **Delete** button to remove them from this **Workbook** tab. The finished **Setup** view appears below.

Figure 2.61	
The new tab now appears in the list of Workbook Tabs in the same order as it appears in	Setup     X       Workbook Tabs     Add       P.T.Flow     Object       Delete     Tab Contents       Unit Ops     Delete       Variables     Variables
the Workbook. The new tab displays only these three Variables.	Variable Format Temperature 4 sig fig Pressure 4 sig fig Std Ideal Liq Vol Flow 4 sig fig Delete Format Order

12. Click the **Close** icon to return to the **Workbook** view and see the new tab.

Workbook - Case (Main) Name	Preheat Crude	Bottom Steam	Diesel Steam	
Temperature [F]	450.0	375.0	300.0	
Pressure [psia]	75.00	150.0	50.00	
Std Ideal Lig Vol Flow [barrel/day	1.000e+005	514.6	205.8	3
Name	AGO Steam	PreFlashVap	PreFlashLic	1
Temperature [F]	300.0	450.0	450.0	)
Pressure [psia]	50.00	75.00	75.00	)
Std Ideal Liq Vol Flow [barrel/day	171.5	6282	9.372e+004	F .
Name	** New **			-
Material Streams P, T, F	low Composition	ns EnergyStre	ams Unit Ops	F

- 13. Save your case by doing **one** of the following:
  - Click the **Save** icon on the tool bar.
  - Select **Save** from the **File** menu.
  - Press CTRL S.

# PFD icon

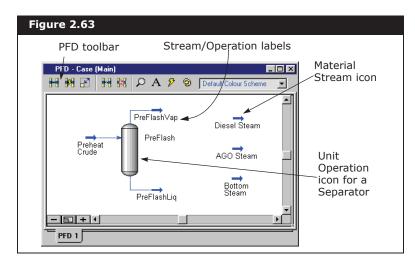
Save icon

# 2.2.8 Using the PFD

The **PFD** is the other main view used in UniSim Design. The **PFD** item appears in the UniSim Design menu bar whenever the **PFD** is active.

1. To open the **PFD**, click the **PFD** icon on the tool bar. The **PFD** view should appear similar to the one shown below, except some stream

icons may overlap each other.



As a graphical representation of your flowsheet, the **PFD** shows the connections among all streams and operations, also known as 'objects'. Each object is represented by a symbol, also known as an 'icon'. A stream icon is an arrow pointing in the direction of the flow, while an operation icon is a graphic representing the actual physical operation. The object name, also known as a 'label', appears near each icon.

The **PFD** shown above has been rearranged by moving the three utility stream icons below and to the left of the **Separator**. To move an icon, click and drag it to the new location.

You can click and drag either the icon (arrow) itself, or the label (stream name), as these two items are grouped together.

Like any other non-modal view, the **PFD** view can be re-sized by clicking and dragging anywhere on the outside border.

Other things you can do while the **PFD** is active include the following:

- Access commands and features through the PFD toolbar.
- Open the property view for an object by double-clicking on its icon.
- Move an object by click and dragging it to the new location.
- Access "pop-up" summary information for an object simply by placing the cursor over it.
- Change an icon's size by clicking the Size Mode icon, clicking on the icon, and click and dragging the sizing handles that appear around the icon.
- Display the Object Inspection menu for an object by placing the cursor over it, and right-clicking. This menu provides access to a number of commands associated with the particular object.



Size Mode icon



Zoom Out 25% icon





• Zoom in and out, or display the entire flowsheet in the PFD window by clicking the zoom buttons at the bottom left corner of the PFD view.

Some of these functions are illustrated here; for more information, see **Section 7.24 - PFD** in the **UniSim Design User Guide**.

# **Calculation Status**

Before proceeding, you will examine a feature of the PFD that allows you to trace the calculation status of the objects in your flowsheet. If you recall, the status indicator at the bottom of the property view for a stream or operation displays one of three possible states for the object:

Status	Description
Red Status	A major piece of defining information is missing from the object. For example, a feed or product stream is not attached to a separator. The status indicator is red, and an appropriate warning message appears.
Yellow Status	All major defining information is present, but the stream or operation has not been solved because one or more degrees of freedom is present, for example, a cooler where the outlet stream temperature is unknown. The status indicator is yellow, and an appropriate warning message appears.
Green Status	The stream or operation is completely defined and solved. The status indicator is green, and an OK message appears.

When you are in the **PFD**, the streams and operations are colour-coded to indicate their calculation status. The inlet separator is completely calculated, so its normal colours appear. While installing the remaining operations through the **PFD**, their colours (and status) changes appropriately as information is supplied.

A similar colour scheme is used to indicate the status of streams. For material streams, a dark blue icon indicates the stream has been flashed and is entirely known. A light blue icon indicates the stream cannot be flashed until some additional information is supplied. Similarly, a dark red icon is for an energy stream with a known duty, while a purple icon indicates an unknown duty.

# Installing the Crude Furnace

In this section, you will install a crude furnace. The furnace is modeled as a **Heater**.

1. Ensure the **Object Palette** is visible (if it is not, press F4).

You will add the furnace to the right of the PreFlash Separator, so make some empty space available by scrolling to the right using the

Keep in mind that these are the UniSim Design default colours; you can change the colours in the Session Preferences.

The icons for all streams installed to this point are dark blue, indicating they have been flashed.

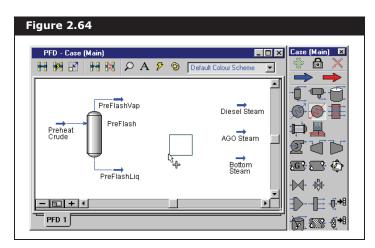


Heater icon (Red)



horizontal scroll bar.

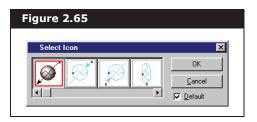
- In the Object Palette, click the **Heater** icon. The cursor changes to a special cursor, with a black frame and plus (+) symbol attached to it. The frame indicates the size and location of the operation icon.
- 3. Position the cursor over the **PFD** to the right of the **separator**.



4. Click to 'drop' the **heater** onto the **PFD**. UniSim Design creates a new **heater** with a default name, **E-100**.

Next you will change the **heater** icon from its default to one more closely resembling a furnace.

- 5. Right-click the **heater** icon. The Object Inspect menu appears.
- 6. Select **Change Icon** from the menu. The **Select Icon** view appears.



7. Click the **WireFrameHeater5** icon (scroll to the right), then click the **OK** button. The new icon appears in the **PFD**.

Notice the **heater** has

red status (colour),

indicating that it

requires feed and product streams.



Furnace icon



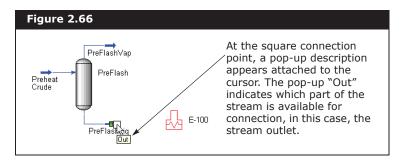
Attach Mode icon

When you are in **Attach** mode, you are not able to move objects in the **PFD**. To return to Move mode, click the **Attach** button again. You can temporarily toggle between **Attach** and **Move** mode by holding down the **ctrl** key.

2-54

#### Attaching Streams to the Furnace

- 1. Click the Attach icon on the PFD tool bar to enter Attach mode.
- 2. Position the cursor over the right end of the **PreFlashLiq** stream icon. A small box appears at the cursor tip.



- 3. With the pop-up 'Out' visible, click and hold the mouse button. The white box becomes black, indicating that you are beginning a connection.
- 4. Drag the cursor toward the left (inlet) side of the **heater**. A trailing line appears between the **PreFlashLiq** stream icon and the cursor, and a connection point appears at the **Heater** inlet.
- 5. Place the cursor near the connection point of the heater, and the trailing line snaps to that point. As well, a white box appears at the cursor tip, indicating an acceptable end point for the connection.



- 6. Release the mouse button, and the connection is made to the **heater** inlet.
- Position the cursor over the right end of the **heater** icon. The connection point and pop-up 'Product' appears.
- 8. With the pop-up visible, click and hold the mouse button. The white box again becomes black.
- 9. Move the cursor to the right of the **heater**. A stream icon appears with a trailing line attached to the **heater** outlet. The stream icon indicates that a new stream is being created.

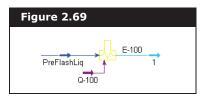
Figure 2.68	
PreFlashLiq	→ E-100



#### **Break Connection** icon If you make an incorrect connection:

- 1. Click the **Break Connection** icon on the **PFD** toolbar.
- 2. Move the cursor over the stream line connecting the two icons. A checkmark attached to the cursor appears, indicating an acceptable connection to break.
- 3. Click once to break the connection.

- 10. With the stream icon visible, release the mouse button. UniSim Design creates a new stream with the default name **1**.
- 11. Create the **Heater** energy stream, starting the connection from the bottom left connection point on the Heater icon labeled 'Energy Stream'. The new stream is automatically named **Q-100**, and the **heater** now has yellow (warning) status. This status indicates that all necessary connections have been made, but the attached streams are not entirely known.



Click the Attach icon again to return to Move mode.
 The heater outlet and energy streams are unknown at this point, so they appear light blue and purple, respectively.

#### Modifying Furnace Properties

- 1. Double-click the Heater icon to open its property view.
- Click the **Design** tab, then select the **Connections** page. The names of the Inlet, **Outlet**, and **Energy** streams appear in the appropriate fields.

E-100					
Design	<u>N</u> ame	E-100			
Connections					
Parameters	Inlet		Energy		
User Variables	PreFlashLig	<b>T</b>	Q-100	-	
Notes	Fluid <u>P</u> ackage Basis-1	<b>⊇</b> -	Outlet	<u> </u>	
Design Rating	Worksheet Performance	ce Dynami	ics		
Delete		Unknowr	n Dutv		[gnored

- 3. In the **Name** field, change the operation **name** to **Furnace**.
- 4. Select the **Parameters** page.

5. In the **Delta P field, enter 10** psi, then close the view.

Design Connections Parameters User Variables Notes	Delta P 10.00 psi
Design Rating	Worksheet Performance Dynamics

The **Furnace** has one available degree of freedom. Either the outlet stream temperature or the amount of duty in the energy stream can be specified. In this case, you will specify the outlet temperature.

- 6. Double-click the outlet stream icon (1) to open its property view.
- 7. In the Stream Name field, change the name to Hot Crude.
- 8. In the Temperature field, specify a temperature of 650°F.

Hot Crude		_ 0
Worksheet	Stream Name	Hot Crud
Conditions	Vapour / Phase Fraction Temperature [F]	0.5355
Properties	Pressure [psia]	65.00
Composition	Molar Flow [lbmole/hr]	3321.
K Value	Mass Flow [lb/hr]	1.0796e+00
	Std Ideal Liq Vol Flow [barrel/day] Molar Enthalpy [Btu/lbmole]	9371 -1.624e+00
User Variables	Molar Entropy [Btu/Ibmole-F]	-1.6248+00
Notes	Heat Flow [Btu/hr]	-5.3950e+0
Cost Parameters	Liq Vol Flow @Std Cond [barrel/day]	9345
	Fluid Package	Basis-1
= Worksheet 4	Attachments Dynamics	
	OK	

The remaining degree of freedom in the Furnace has now been used, so UniSim Design can flash **Hot Crude** and determine its remaining properties.

9. Close the view to return to the **PFD view**. The **Furnace** now has green status, and all attached streams are known.

- 10. Double-click on the energy stream icon (**Q-100**) to open its property view. The required heating duty calculated by UniSim Design appears in the **Heat Flow** cell.
- 11. In the **Stream Name** cell, rename this energy stream **Crude Duty**, then close the property view.

Figure 2.73	
Energy Stream: Crude Duty	
Properties           Stream Name         Crude Duty           Heat Flow [Btu/hr]         1.8626e+08           Ref. Temperature [F] <empty></empty>	
Stream Unit Ops Dynamics Stripchart	
Delete	

## Installing the Mixer

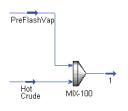
In this section, you will install a Mixer operation. The **Mixer** is used to combine the hot crude stream with the vapours bypassing the furnace. The resulting stream is the feed for the crude column.

- 1. Make some empty space available to the right of the **Furnace** using the horizontal scroll bar. Move other objects if necessary.
- 2. Click the **Mixer** icon on the **Object Palette**.
- 3. Position the cursor over the **PFD** to the right of the **Hot Crude** stream icon.
- 4. Click to 'drop' the **mixer** onto the **PFD**. UniSim Design creates a new **mixer** with the default name **MIX-100**.
- 5. Press and hold the **CTRL** key to temporarily enable the Attach mode while you make the **mixer** connections (you will not release it until step #13).
- 6. Position the cursor over the right end of the **PreFlashVap** stream icon. The connection point and pop-up 'Out' appears.
- With the pop-up visible, click and hold the mouse button, then drag the cursor toward the left (inlet) side of the **mixer**. Multiple connection points appear at the **mixer** inlet.
- 8. Place the cursor near the inlet area of the **mixer**, and when the white box appears at the cursor tip, release the mouse button to make the connection.
- 9. Repeat steps #6 to #8 to connect the **Hot Crude** stream to the **Mixer**.
- 10. Position the cursor over the right end of the **mixer** icon. The connection point and pop-up 'Product' appears.



Mixer icon

Multiple connection points appear because the **Mixer** accepts multiple feed streams.



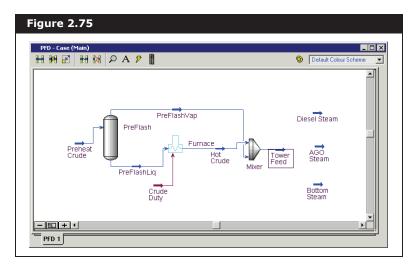
- 11. With the pop-up visible, click and drag to the right of the **mixer**. A white stream icon appears, with a trailing line attached to the **mixer** outlet.
- 12. With the white stream icon visible, release the mouse button. UniSim Design creates a new stream with the default name **1**.
- 13. Release the **ctrl** key to leave **Attach** mode.
- 14. Double-click on the outlet stream icon 1 to access its property view. When you created the **mixer** outlet stream, UniSim Design automatically combined the two inlet streams and flashed the mixture to determine the outlet conditions.
- 15. In the **Stream Name** cell, rename the stream **Tower Feed**, then close the view.

16. Double-click the **mixer** icon, **MIX-100**. Change the name to **Mixer**, then close the view.

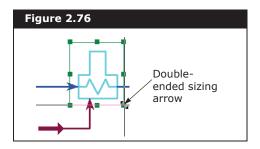
### Resizing Icons in the PFD

Resize icons in the PFD to make it easier to read.

- 1. Resize the PFD view by clicking and dragging the outside border.
- 2. Click the **Zoom All ic**on to fill the **PFD** window, including any objects that were not visible previously. A possible view of the resized **PFD** appears in the figure below.



- 3. Click the Size Mode icon on the PFD toolbar.
- 4. Click the **Furnace** icon in the **PFD**. A frame with sizing handles appears around the icon.
- 5. Place the cursor over one of the sizing handles. The cursor changes to a double-ended sizing arrow.



- 6. With the sizing arrow visible, click and drag to resize the icon.
- 7. Click the **Size Mode** icon again to return to Move mode.

# Adding an Energy Stream

In this section, you will add an energy stream. Prior to installing the column, an energy stream must be created to represent the trim duty on stage 28 of the main tower.

1. Double-click on the **Energy Stream** icon on the **Object Palette**. UniSim Design creates a new energy stream with the default name



Size Mode icon

Zoom All icon



Energy Stream icon

2-60

**Q-100** and display its property view.

- 2. In the Stream Name field, change the name to Trim Duty.
- 3. Close the view.
- 4. Save your case by doing one of the following:
  - press CTRL S.
  - from the File menu, select Save.
  - click the **Save** icon.

# Installing the Column

UniSim Design has a number of pre-built column templates that you can install and customize by changing attached stream names, number of stages and default specifications, and adding side equipment. One of these templates is going to be used for this example (a crude column with three side strippers), however, a basic **Refluxed Absorber** column with a total condenser is installed and customized in order to illustrate the installation of the necessary side equipment.

- 1. Before installing the column, select **Preferences** from the UniSim Design **Tools** menu. Click the **Simulation** tab.
- 2. On the **Options page**, ensure the **Use Input Experts** checkbox is checked, then close the view.
- 3. Double-click the **Refluxed Absorber** icon on the **Object Palette.** The first page of the **Input Expert** appears.

Figure 2.77			
Refluxed Absorber Column Input Expe	rt		×
Condenser Energy Stream		Condenser C Total C Partial C Full Rflx	Oyhd Outlets
Stream Inlet Stage # 9	2 2 2 10 10	<u>W</u> ater Draw      Optional Side Draws <u>Stream     </u>	Type Draw Stage
Stage Numbering	n-1 n		Bottoms Liguid Outlet
Top Down     C Bottom Up		Connections (page 1	of 4) Cancel

The **Input Expert** is a Modal view, indicated by the absence of the **Maximize/Minimize** icons. You cannot exit or move outside the Expert view until you supply the necessary information or click the Cancel button.

When you install a column using a pre-built template, UniSim Design

•**§**•

Beflexed Absorber icon

If you choose to use the pre-built crude column template you still have to customize the column by modifying the various draw and return stages and default specifications, Although using the template eliminates the majority of the work over the next few pages, it is recommended that you work through these pages the first time you build a crude column in UniSim Design, Once Tounstallthirfordum woongitgenpretheilsigende equipment, platesing the tendolatere-lantkustione on cassoff column icon columnteoplate prette. given in an annotation on Quethextipy deat

appears, click the Read an Existing

Column Template button. The Available Column Templates view appears, listing the template files \*.col that are provided in your UniSim Design\template directory. Both 3- and 4-side stripper crude column templates are

provided. 3. Select **3sscrude.col** and click the **OK** button. The property view for the new column appears. You can now customize the new column.

Refining Tutorial	2-61

supplies certain default information, such as the number of stages. The current active field is **# Stages** (Number of Stages), indicated by the thick border inside this field. There are some other points worth noting:

- These are theoretical stages, as the UniSim Design default stage efficiency is one.
- If present, the Condenser and Reboiler are considered separate from the other stages, and are not included in the # Stages field.

#### Entering Inlet Streams and Number of Trays

For this example, the main column has 29 theoretical stages.

- 1. Enter **29** in the **# Stages** field.
- 2. Advance to the **Optional Inlet Streams** table by clicking on the **<<Stream>>** cell, or by pressing **tab**.
- Click the down arrow 
   ■ to open the drop-down list of available feeds.

Figure 2.78 Refluxed Absorber Column Input Expert	×
Column Name T-100	Condenser C Total C Patial C Full Rfix
Optional Intel Streams     2       Stream     Intel Strage       AGD Steam     Image: Strage stream       Dissel Steam     Image: Strage stream       Tower Feed     Image: Strage stream       Time Duty     Image: Strage stream       Dissel Steam     Image: Strage stream	
Stage Numbering Top Down C Bottom Up	Bottoms Liguid Outlet
< Prev Negt >	Connections (page 1 of 4)

- Select **Tower Feed** as the feed stream to the column. UniSim Design supplies a default feed location in the middle of the Tray Section (TS), in this case stage 15 (indicated by **15\_Main TS**). However, the feed stream needs to enter stage 28.
- 5. In the Optional Inlet Streams group, click in the **Inlet Stage** cell for **TowerFeed**.
- 6. Type **28** and press **enter**, or select **28\_Main TS** from the dropdown list of stages.
- 7. Click on **<<Stream>>** in the same table, which was automatically advanced down one cell when you attached the Tower Feed stream.

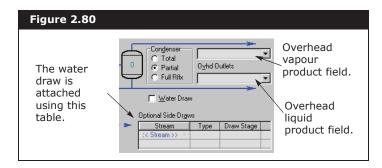
8. From the Stream drop-down list, select the **Trim Duty** stream, which is also fed to stage 28.



- 9. Advance to the **Bottom Stage Inlet** field by clicking on it or by pressing **tab**.
- 10. In the **Bottom Stage Inlet** field, click the down arrow I to open the drop-down list of available feeds.
- 11. From the list, select **Bottom Steam** as the bottom feed for the column.

#### **Entering Outlet Streams**

In the **Condenser** group of the Input Expert view, the default condenser type is **Partial**. To the right of this group, there are two **Overhead Outlets**, vapour and liquid. In this case, the overhead vapour stream has no flow, and two liquid phases (hydrocarbon and water) are present in the condenser. The hydrocarbon liquid product is attached in the liquid **Overhead Outlets** field, while the water draw is attached using the **Optional Side Draws** table.



Although the overhead vapour product has zero flow, do not change the condenser to **Total**. At this time, only the **Partial** radio button allows you to specify a three-phase condenser.

- 1. Click in the top **Ovhd Outlets fi**eld.
- 2. Enter **Off Gas** as the name of the overhead vapour product stream. UniSim Design creates and attaches a new stream with this name.
- Press tab again to move to the bottom Ovhd Outlets field, and enter the new stream name Naphtha.

The next task is to attach the water draw stream to the condenser.

- 4. In the **Optional Side Draws** table, click in the **<<Stream>>** cell.
- Enter the name of the draw stream, WasteH2O. UniSim Design automatically places a hydrocarbon liquid (indicated by the L in the Type column) draw on stage 15. You will change this to a condenser water draw.
- 6. Click on the **Type** cell (the **L**) for the **WasteH2O** stream.
- 7. Specify a water draw by typing **W** then pressing **enter**, or by selecting W from the drop-down list.
- 8. Click on the **Draw Stage** cell (**15\_Main TS**) for the **WasteH2O** stream.
- 9. Select **Condenser** from the drop-down list. The condenser is now three-phase.

Figure 2.81					
ل ب	Condenser C Total C Partial C Full Rflx	Off Gas O <u>v</u> hd O Naphth	utlets		
✓ Water Draw WasteH2D ✓					
	Optional Side Dr <u>a</u> w	/s			
	Stream	Туре	Draw Stage		
	WasteH20 😁	- W -	ondenser 👘		
	<pre>&lt;&lt; Stream &gt;&gt; **</pre>				

- 10. In the Column Name field, enter Atmos Tower.
- 11. In the **Bottoms Liquid Outlet** field, type **Residue** to create a new stream.
- 12. In the **Condenser Energy Stream** field, type **Cond Duty** to define a new stream. Press **ENTER**.

All stream attachments made on this page result in the creation of Column sub-flowsheet streams with the same names. For example, when the Main Flowsheet stream BottomSteam was attached as a feed, UniSim Design automatically created an identical stream named BottomSteam to be used in the Column sub-flowsheet.

The first page of the Input Expert should appear as shown below.

Figure 2.82		
Refluxed Absorber Column Input Expert         Condenser Energy Stream         Column Name         Atmos Towel         Optional Inlet Streams         Stream         Tower Fed         Bottom Stage Inlet         Bottom Steam         Stage Numbering         Top Down         Bottom Up	Condenser C Total Partial Full Rflx Water Draw Coptional Side Drgws Stream ValerH20 V Stream S	
< Prev Next>	Connections (page 1 of 4)	

The Next button now becomes available, indicating sufficient information has been supplied to advance to the next page of the **Input Expert**.

13. Click the **Next** button to advance to the **Pressure Profile** page.

### Entering the Initial Estimate Values

- 1. On the **Pressure Profile** page, specify the following:
  - Condenser Pressure 19.7 psia
  - Condenser Pressure Drop 9 psi
  - Bottom Stage Pressure 32.7 psia

Figure 2.83
Refluxed Absorber Column Input Expert
Condenser Pressure 19.70 psia Condenser Pressure Drop 9.000 psi Bottom Stage Pressure 12.70 psia
Yressure Profile (page 2 of 4) <u>Cancel</u>

- Click the Next button to advance to the Optional Estimates page. Although UniSim Design does not usually require estimates to produce a converged column, good estimates result in a faster solution.
- 3. Specify the following:
  - Condenser 100°F
  - Top Stage 250°F
  - Bottom Stage 700°F

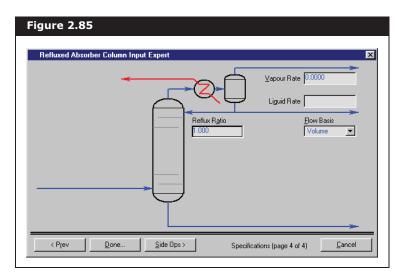
Figure 2.84
Refluxed Absorber Column Input Expert
Optional Condenser Temperature Estimate 100.0 F Optional Iop Stage Temperature Estimate 250.0 F Optional Bottom Stage Temperature Estimate 700.0 F
< <tr>         &lt; Prev         Next&gt;         Optional Estimates (page 3 of 4)         Cancel</tr>

 Click the Next button to advance to the fourth and final page of the Input Expert. This page allows you to supply values for the default column specifications that UniSim Design has created.

In general, a refluxed absorber with a partial condenser has two degrees of freedom for which UniSim Design provides two default specifications. For the two specifications given, overhead Vapour Rate is used as an active specification, and **Reflux Ratio** as an estimate only.

- 5. From the Flow Basis drop-down list, select Volume. All flow specifications are provided in barrels per day.
- 6. Specify the following:
  - Vapour Rate 0

• Reflux Ratio 1.0.



7. Click the **Done** button. The Column property view appears.

Design	Column Name Atmos Tower Sub-Flowsheet Tag COL1 Condenser
Connections Monitor Specs Specs Summary Subcooling Notes	Condenser Energy Stream Cond Duty
nues	Stream     Inlet Stage     P cond     Optional Side Draws       Tower Feed     28     Ma     T     Stages       Trim Duty     28     Ma     Image: Stage Number ing     Image: Stage Number ing       Bottom Stage Numbering     Image: Stage Number ing     Image: Stage Number ing     Image: Stage Number ing       Stage Numbering     Image: Stage Number ing     Image: Stage Number ing     Image: Stage Number ing
	Edit Trays

### Adding Specification Values

1. On the Design tab, select the **Monitor** page.

The main feature of this page is that it displays the status of your column as it is being calculated, updating information with each iteration. You can also change specification values, and activate or de-activate specifications used by the Column solver, directly from this page.

The basic column has three available degrees of freedom. Currently, two Specifications are Active, so the overall Degrees of Freedom is one. The number of available degrees of freedom increases with the addition of side equipment.

The Draw Spec is entered so that the degrees of freedom is kept at zero throughout this tutorial. It is good practice to keep the degrees of freedom at zero as you modify your column so that you can solve the column after every modification. The current **Degrees of Freedom** is one, indicating that only two specifications are **active**. As noted earlier, a Refluxed Absorber with a partial condenser has two degrees of freedom and, therefore, requires two **active** specifications. In this case, however, a third degree of freedom was created when the **Trim Duty** stream was attached as a feed, for which the heat flow is unknown. UniSim Design has not made a specification for the third degree of freedom, therefore you need to add a water draw spec called **WasteH2O Rate** to be the third active specification.

- 2. Select the **Specs** page. Here you will remove two specifications and add one new specification.
- 3. In the Column Specifications group, select Reflux Rate and then click the **Delete** button.
- 4. Delete the Btms Prod Rate specification also.
- 5. Next you will add the WasteH2O Rate specification. Click the **Add** button. The Add Specs view appears.
- Select Column Draw Rate and click the Add Spec(s) button. The Draw Spec property view appears.
- 7. In the **Name** cell, type WasteH2O Rate. No further information is required as this specification is de-activated and only estimated when you run the column.

Figure 2.87	
Draw Spec: Was	teH20 Bate
Name	WasteH20 Rate
Draw Flow Basis	<< Stream >> * Volume *
Spec Value	<empty></empty>
Parameters	Summary   Spec Type
Delete	

- 8. Close the view. The new specification appears in the Column Specifications group. The Degrees of Freedom is now zero.
- 9. Select the **Connections** page. See Figure 2.86.

The Connections page is similar to the first page of the **Input Expert**. Currently, the column is a standard type, so this page shows a column schematic with the names of the attached streams. When the side equipment is added to the column, the page becomes non-standard. There are a large number of possible nonstandard columns based on the types and numbers of side operations that are added. Therefore, UniSim Design modifies the **Connections** page into a tabular format, rather than a schematic format, whenever a column becomes non-standard. In the next section you will add the side equipment and observe how the Connections page is modified.

## Installing the Side Strippers

1. Click the **Side Ops** tab of the Column property view.

Side Ops	-Side Stripper Summar	y				
Side Strippers		# Stages	Liq Draw Stage	Vap Return Stage	Outlet Flow	Reboiler Duty [Btu/hr]
Pump Arounds						
Vap Bypasses						
Side Draws						
	Elow Basis	<u> </u>	ne <u>V</u> iew	Add	Delete Si	de Ops Input Expert
	• Molar • Ma	ss (O Volur		<u></u>		de opsinipar Expert

On this tab, you can Install, View, Edit, or Delete all types of Side Equipment. The table displays summary information for a given type of side operation, depending on the page you are currently on.

- 2. Ensure that you are on the **Side Strippers** page.
- 3. Click the Add button. The Side Stripper view appears.

Side Stripper	551	_ 🗆 ×
Name SS1		
<u>R</u> eturn Stage		
	→ 1	Configuration
Draw Stage	2	Reboiled
1		C Steam Stripped
	k = 3	
	k-1	
Elow Basis	k	Boil Up Ratio
Molar		0.75
C Mass	T	$\rightarrow$
C Volume		( <u>∠</u> )——>
C Std Volume		
C Std Volume		Product Stream Information

When you install side equipment, it resides in the Column subflowsheet. You can build a complex column in the sub-flowsheet while in the Main Flowsheet, the column appears as a single operation. You can then transfer any needed stream information from the sub-flowsheet by simply attaching the stream to the Main Flowsheet.

2-68

This is a **reboiled** 3stage stripper with a 0.75 boil up ratio, so leave the **Configuration** radio button at Reboiled, and the **k** = and **Boil Up Ratio** fields at their defaults.

- 4. In the **Name** field, change the name to **KeroSS**.
- 5. In the **Return Stage** drop-down list, select stage 8 (**8\_Main TS**).
- 6. In the **Draw Stage** drop-down list, select stage 9 (**9\_Main TS**).
- 7. In the **Flow Basis** group, select the **Std Ideal Vol** radio button.
- 8. In the Product Stream field, enter Kerosene.
  - The straight run product distribution data calculated during the Oil Characterization appears in the figure below.

Figure 2.9	90				
Cut Distributions					
Name	Begin T (F)	End T (F)	Fraction		
Lt St Run	100.3	158.0	0.033		Kerosene
Naphtha	158.0	356.0	0.172		
Kerosene	356.0	464.0	0.129	<	🛏 Liquid
Light Diesel	464.0	554.0	0.089		Volume
Heavy Diesel	554.0	644.0	0.083		Fraction
Atm Gas Oil	644.0	698.0	0.049		Fraction
Residue	698.0	1441	0.444		
l					
,					I.

The Kerosene liquid volume fraction is 0.129. For 100,000 bbl/day of crude fed to the tower, Kerosene production can be expected at **100,000 \* 0.129 = 12,900** or approximately 13,000 bbl/day.

9. In the **Draw Spec** field, enter **13000**. The completed **Side Stripper** view appears below.

Figure 2.91	
Side Stripper - KeroSS	×
Name KeroSS	
Return Stage	
8_Main TS	
Draw Stage	
9_Main TS  Steam Stripped	
k = 3	
Elow Pasia	
LIOW Dasis	
C Molar	-
Vol @ Std Lond Product Stream Specs	_
Delete Install Draw Spec 1.300e+004 barrel	7
	-

10. Click the **Install** button, and a view summarizing your input appears.

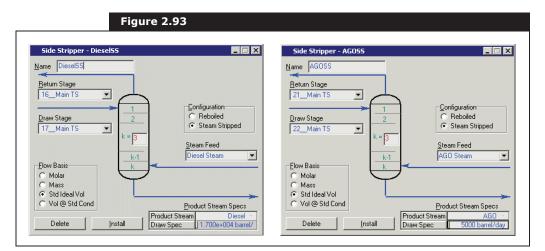


11. Click the **Close** icon to return to the Column property view. Summary information for the new side operation appears in the table on the **Side Ops** tab.

gure 2.9	92				
C.1 CI. C					
Si <u>d</u> e Stripper Su	mmary				
sige stripper Su	mmary # Stages	Liq Draw Stage	Vap Return Stage	Outlet Flow [lbmole/hr]	Reboiler Duty [Btu/hr]

12. Use the previous steps to install the two remaining side strippers DieseISS and AGOSS. These are both Steam Stripped, so choose the appropriate Configuration radio button and create the Steam Feed and Product streams as shown in the following figures. The @COL1 suffix is added automatically.

The completed **DieseISS and AGOSS side stripper** views appear in the following figure.



Although not a requirement, the names of the Steam Feed streams created for these side strippers are identical to the names of the utility steam streams that were created previously in the Main Flowsheet. The conditions of these Steam Feed streams, which reside in the Column subflowsheet, are unknown at this point. The conditions of the Main Flowsheet streams are duplicated into these subflowsheet streams when the stream attachments are performed. The completed Side Stripper Summary table appears below.

gure 2.9	4				
Side Stripper Sur					
olge okipper our	imary			Outline Flam	Dahala Daha
	# Stages	Liq Draw Stage	Vap Return Stage	Outlet Flow [Ibmole/hr]	Reboiler Duty [Btu/hr]
		9 Main TS 👻	8 Main TS 👻	<empty></empty>	<empty></empty>
KeroSS	3	3Main 15	o_main is *	<empty <="" td=""><td>Composition</td></empty>	Composition
KeroSS DieselSS	3	17Main TS	6Main TS 16Main TS	<empty></empty>	<empty></empty>

13. Click the **Design** tab and select the **Monitor** page.

The **Specifications** table on this page has a vertical scroll bar, indicating that new specifications have been created below the default ones. Resize the view to examine the entire table.

14. Click and drag the bottom border of the view down until the scroll bar disappears, making the entire matrix visible.

igure 2.95						
	Specified Value	Current Value	Wt. Error	Active	Estimate	Current
Reflux Ratio	1.000	<pre>cancing value <empty></empty></pre>	<empty></empty>			
Distillate Rate	<empty></empty>	<empty></empty>	<empty></empty>	ম	<u> </u>	<u> </u>
Vap Prod Rate	0.0000 barrel/day	<empty></empty>	<empty></empty>	Г		
WasteH20	<empty></empty>	<empty></empty>	<empty></empty>	<b>N</b>		2
KeroSS Prod Flow	1.300e+004 barrel/day	<empty></empty>	<empty></empty>	<b>N</b>		2
KeroSS BoilUp Ratio	0.7500	<empty></empty>	<empty></empty>	<b>v</b>		2
DieselSS Prod Flow	1.700e+004 barrel/day	<empty></empty>	<empty></empty>	<b>N</b>		
AGOSS Prod Flow	5000 barrel/day	<empty></empty>	<empty></empty>	V		V

The addition of the side strippers has created four more degrees of freedom above the basic column, resulting in a total of seven available degrees of freedom. Currently, however, seven Specifications are Active, so the overall Degrees of Freedom is zero.

The installation of the side strippers created four additional degrees of freedom, so UniSim Design created a **Prod Flow** (product flow) specification for each side stripper, plus a **BoilUp Ratio** specification for the Kerosene side stripper. The new specifications were automatically made **Active** to exhaust the four degrees of freedom, returning the overall **Degrees of Freedom** to **0**.

## Installing the Pump Arounds

- 1. Click the **Side Ops** tab and select the **Pump Arounds** page.
- 2. Click the Add button. The initial Pump Around view appears.
- 3. In the **Return Stage** drop-down list, select stage 1 (**1\_Main TS**).
- 4. In the **Draw Stage** drop-down list, select stage 2 (**2\_Main TS**).

5. Click the **Install** button, and a more detailed **Pump Around** view appears.

igure 2.96		
Pump Around - PA_1		
1st Active 2nd Act PA_1_Rate(Pa)	ive Spec PA_1_Dt(Pa) <empty></empty>	
Return Stage 1_Main TS ▼		
	Calculated Inform	
	Draw Temp.	<empty></empty>
Draw Stage 🖌 🗼	dT	<empty></empty>
2 Main TS 👻	Return Temp.	<empty></empty>
	Flow Rate	<empty></empty>
	Duty	<empty></empty>
	View <u>P</u> ump	View <u>V</u> alve
Delete PA_1		

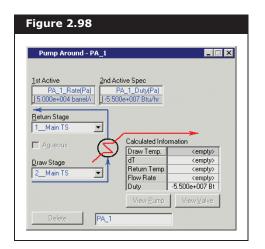
Each cooled pump around circuit has two specifications associated with it. The default **Pump Around Specifications** are circulation rate and temperature drop (**Dt**) between the liquid draw and liquid return. For this example, the **Dt** specification is changed to a Duty specification for the pump around cooler. The pump around rate is 50,000 bbl/day.

- In the empty cell under the PA\_1\_Rate(Pa) specification, enter 5e4.
- Double-click in the blank space under the PA\_1\_Dt(Pa) specification, and the Spec view appears.
- 8. In the **Spec Type** drop-down list, select **Duty**.
- 9. in the **Spec Value** cell, enter **-55e6**.

Figure 2.97	
Pump Around Spe	ec: PA_1_Duty( 💶 💌
Spec Type	
Duty	<b>_</b>
Name	PA_1_Duty(Pa)
Pump Around	PA_1 ×
Spec Value	-5.500e+007 Btu/hr
Parameters S	Summary Spec Type
Delete	

Notice the negative sign convention indicates cooling.

10. Click the Close icon to return to the Pump Around view.



The remainder of the information on the above view is calculated by the Column solver.

- 11. Click the **Close** icon on the main **Pump Around** view to return to the Column property view.
- 12. Repeat the previous steps to install the two remaining pump arounds. Enter Rate specifications of **3e4 barrel/day** and **Duty** specifications of **-3.5e7 Btu/hr** for both of these pump arounds.

The completed **Pump Around** views and **Liquid Pump Around Summary** table appear in the following figures.

Pump Around - PA 2	Pump Around - PA 3
1st Active     2nd Active Spec       PA_2_Rate(Pa)     PA_2_Duty(Pa)       3.000e+004 barrel/     -3.500e+007 Btu/hr       Return Stage     Calculated Information       16_Main TS     Calculated Information       Draw Stage     Draw Temp.       17_Main TS     Return Temp.       Kage     Calculated Information	1st Active       2nd Active Spec         PA_3_Rate(Pa)       PA_3_Duty(Pa)         ]3.000e+004 barrel/       J-3.500e+007 Btu/hr         Beturn Stage       21_Main TS         Calculated Information       Draw Temp.         Draw Stage       dT         22_Main TS       Flow Rate
Duty         -3.500e+007 Bt           View <u>P</u> ump         View <u>V</u> elve           Delete         PA_2	Duty     -3.500e+007 Bt       View <u>Pump</u> View <u>V</u> alve       Delete     PA_3

gure 2.100							
Liquid Pump	Around Summary						
	Draw Stage	Return Stage	Flow [Ibmole/hr]	Duty [Btu/hr]	Draw T [F]	Return T (F1	Export
PA_1	2Main TS 👻	1Main TS 👻	<empty></empty>	-5.500e+007	<empty></empty>	<empty></empty>	
PA_2	17Main TS 🝸	16Main TS 👘	<empty></empty>	-3.500e+007	<empty></empty>	<empty></empty>	
PA_3	22Main TS 👻	21Main TS 👻	<empty></empty>	-3.500e+007	<empty></empty>	<empty></empty>	

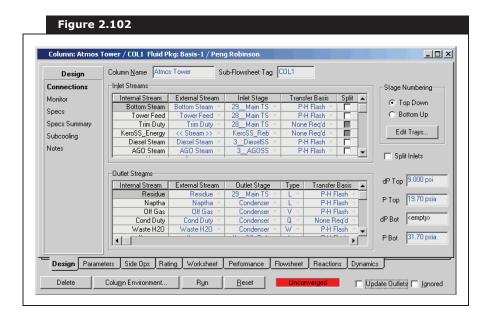
- 1. Click the **Add** button.
- 2. Specify the Return Stage and Draw Stage.
- 3. Click the **Install** button. The second view appears.
- 4. Specify the 1st Active spec.
- 5. Double-click the empty cell in the 2nd Active spec.
- 6. Select Duty from the Spec Type dropdown list.
- 7. Enter the Spec Value.
- 8. Close the view.

Specifications						
speenegaene	Specified Value	Current Value	Wt. Error	Active	Estimate	Current
Distillate Rate	<pre><empty></empty></pre>	<empty></empty>	<empty></empty>	ন	<b>v</b>	<u> </u>
Vap Prod Rate	0.0000 barrel/day	<empty></empty>	<empty></empty>	Г	•	Г
Reflux Ratio	<empty></empty>	<empty></empty>	<empty></empty>	<b>v</b>	<b>N</b>	<b>v</b>
WasteH20 Rate	<empty></empty>	<empty></empty>	<empty></empty>	<b>V</b>	•	<b>v</b>
KeroSS Prod Flow	1300 barrel/day	<empty></empty>	<empty></empty>			<b>v</b>
KeroSS BoilUp Ratio	0.7500	<empty></empty>	<empty></empty>	<b>v</b>	<b>N</b>	•
DieselSS Prod Flow	17000 barrel/day	<empty></empty>	<empty></empty>	<b>v</b>		<b>v</b>
AGOSS Prod Flow	5000 barrel/day	<empty></empty>	<empty></empty>	<b>v</b>		•
PA_1_Rate(Pa)	5.000e+004 barrel/day	<empty></empty>	<empty></empty>	<b>v</b>		•
PA_1_Duty(Pa)	-5.500e+007 Btu/hr	-5.50e+007	0.0000			•
PA_2_Rate(Pa)	3.000e+004 barrel/day	<empty></empty>	<empty></empty>	<b>v</b>		<b>v</b>
PA_2_Duty(Pa)	-3.500e+007 Btu/hr	-3.50e+007	0.0000			•
PA_3_Rate(Pa)	3.000e+004 barrel/day	<empty></empty>	<empty></empty>	<b>v</b>		
PA 3 Duty(Pa)	-3.500e+007 Btu/hr	-3.50e+007	0.0000			<b>N</b>

13. Click the **Design tab and select the Monitor** page. Re-size the property view again so the entire **Specifications** table is visible.

The addition of each pump around created two additional degrees of freedom. As with the side strippers, the specifications for the pump arounds have been added to the list and were automatically activated.

#### 14. Select the **Connections** page.



The **Connections** page of a standard refluxed absorber property view is essentially identical to the first page of the refluxed absorber **Input Expert**, with a column schematic showing the feed and product streams. Side equipment have been added to the standard refluxed absorber, however, making the column non-standard. The **Connections** page has therefore been modified to show tabular summaries of the **Column Flowsheet Topology** (i.e., all

The addition of the pump arounds has created six more degrees of freedom, resulting in a total of 13 available degrees of freedom. Currently, 13 Specifications are active, so the overall Degrees of Freedom is zero.

#### equipment), Feed Streams, and Product Streams.

The column has 40 Total Theoretical Stages:

- 29 in the main tray section
- 1 condenser for the main column
- 9 in the side strippers (3 side strippers with 3 stages each)
- 1 reboiler for the Kerosene side stripper

This topology results in 4 **Total Tray Sections**—one for the main column and one for each of the three side strippers.

#### Completing the Column Connections

When the stream attachments were made on the initial page of the Input Expert, UniSim Design automatically created Column subflowsheet streams with the same names. For example, when **Bottom Steam** was attached as a column feed stream, UniSim Design created an identical sub-flowsheet stream named **Bottom Steam**. In the **Inlet Streams** table on the **Connections** page, the Main Flowsheet stream is the **External Stream**, while the sub-flowsheet stream is the **Internal Stream**.

gure 2.10	3				
nļet Streams					
Internal Stream	External Stream	Inlet Stage	Transfer Basis	Split	
Bottom Steam	Bottom Steam 👻	29Main TS 👻	P-H Flash 👻	Ē	
Tower Feed	Tower Feed 👻	28Main TS 👻	P-H Flash 👻		
Trim Duty	Trim Duty 👻	28Main TS 👻	None Reg'd 👻	Г	1
KeroSS_Energy	<< Stream >> =	KeroSS_Reb 👻	None Reg'd 👻	Г	1
Diesel Steam	Diesel Steam 👻	3_DieselSS 🕆	P-H Flash 👻		1
AGO Steam	AGO Steam 👻	3_AGOSS -	P-H Flash 👻	Г	-
	-				- <u></u>

If you scroll down the list of **Inlet Streams**, notice that the two side stripper steam streams, **DieselSteam** and **AGOSteam**, are **Internal** and External, meaning that these streams are attached to the Main Flowsheet streams that were created earlier.

For the purposes of this tutorial, it is not required to export the pump around duty streams **PA\_1\_Q**, **PA\_2\_Q**, and **PA\_3\_Q** to the Main Flowsheet, so their **External Stream** cells remain undefined.

#### Adding Column Specifications

Select the **Monitor** page of the Column property view.

The current **Degrees of Freedom** is zero, indicating the column is ready to be solved. Before you run the column, however, you will have to replace two of the active specifications, **Waste H2O Rate** and **KeroSS BoilUp Ratio**, with the following new ones:

- Overflash specification for the feed stage (Tray Net Liquid Flow specification)
- Kerosene side stripper reboiler duty specification

Adding the Overflash Specification

1. On the **Design** tab, move to the **Specs** page.

Design	Column Specifications	Specification Details
Connections	Distillate Rate	Spec Name WasteH20 Rate
Monitor	Reflux Ratio Add 1	Converged ? No Current
Specs	KeroSS Prod Flow	
Specs Summary	KeroSS BoilUp Ratio Delete	Spec Type
Subcooling	AGOSS Prod Flow	Fixed/Ranged Spec Fixed
Notes	PA_1_Rate(Pa) PA_1_Duty(Pa)	Primary/Alternate Spec Primary *
		Values
	Update Specs from Dynamics	Specification Value <empty></empty>
	Default Basis Volume	Current Calculated Value <empty></empty>
		Errors
	Degrees of Freedom	Weighted Tolerance 1.000e-002
		Weighted Calculated Error <empty> Absolute Tolerance 151.0 barrel/day</empty>
	Switch To Alternate Specs	Absolute Calculated Error <empty></empty>
	SWITCH TO Alternate Specs	TAbsolute calculated Elitor

- In the Column Specifications group, click the Add button. The Add Specs view appears.
- 3. Select Column Liquid Flow as the Column Specification Type.
- 4. Click the **Add Spec(s)** button, and the **Liq Flow Spec** view appears.
- 5. Change the **name** from its default to **Overflash**.
- 6. In the **Stage** cell, select **27\_Main TS** from the drop-down list of available stages.

A typical range for the Overflash rate is 3-5% of the total feed to the column. In this case, the total feed rate is 100,000 barrels/day. For the Overflash specification 3.5%, or 3,500 barrels/day is used.

Add Specs - Atmos Tow 🗙
Column Specification Types
Column Cold Properties Spec Column Component Flow Column Component Ratio Column Component Ratio Column Component Recovery Column Day Rate Column Day Rate Column DI Spec Column DI Spec Column DU Column Duty Ratio Column Feed Ratio Column Feed Ratio Column Feed Cut Point
Column Liquid Flow Column Physical Properties Spec Column Physical Properties Spec Column Recovery Column Reflux Feed Ratio Spec Column Reflux Fraction Spec Column Reflux Fraction Spec
Eas opeo(o)

7. In the **Spec Value** cell, enter **3500**.

Figure 2.105	
Liq Flow Spec: Ove	erFlash 📃 🔲 🕨
Name	OverFlash
Stage	27Main TS 👻
Flow Basis	Std Ideal Vol 👻
Spec Value	3500.00 barrel/day
Parameters S	ummary Spec Type
Delete	

8. Close the view to return to the Column property view. The new specification appears in the list of **Column Specifications** group on the **Specs** page.

#### Adding the Duty Specification

- 1. Click the Add button again to add the second new specification.
- Select Column Duty as the Column Specification Type, then click the Add Spec(s) button. The Duty Spec view appears.
- 3. In the Name cell, change the name to Kero Reb Duty.
- 4. In the **Energy Stream** cell, select **KeroSS\_Energy @COL1** from the drop-down list.
- 5. In the Spec Value cell, enter 7.5e6 (Btu/hr).

Figure 2.106	5
Duty Spec: Kero	Reb Duty
Name	Kero Reb Duty
Energy Stream Spec Value	KeroSS_Energy @COL1  7.5e+006 Btu/hr
Delete	Summary Spec Type

 Close the view to return to the Specs page of the Column property view. The completed list of Column Specifications is shown in the figure below

Figure 2.107	
Column Specifications       Beflux Ratio       Distillate Rate       Vap Prod Rate       WastH2D0 Rate       KeroSS Prod Flow       AGD0S Prod Flow       PA_1 Date(Pa)       PA_2 Daty(Pa)       PA_2 Daty(Pa)       PA_3 Duty(Pa)       PA_3 Duty(Pa)	
Update Specs from Dynamics	
Default Basis Volume	
Degrees of Freedom	

#### Running the Column

1. Select the **Monitor** page to view the **Specifications** matrix.

The **Degrees of Freedom** is again zero, so the column is ready to be calculated, however, a value for the distillate (Naphtha) rate specification must be supplied initially. In addition, there are some specifications which are currently **Active that you want to use** as **Estimates** only, and vice versa.

Make the following final changes to the specifications:

- In the Specified Value cell for the Distillate Rate specification, enter 2e4 (barrel/day).
- 3. Activate the **Overflash** specification by clicking its **Active** checkbox.
- 4. Activate the Kero Reb Duty specification.
- 5. Activate the Vap Prod Rate specification.
- 6. Deactivate the **Reflux Ratio** specification.
- 7. Deactivate the Waste H2O Rate specification.
- 8. Deactivate the KeroSS BoilUp Ratio specification.

UniSim Design begins calculations and the information displayed on the page is updated with each iteration. The column converges as

If the column begins to run on its own before you click the **Run** button, click the **Stop** button and continue activating or deactivating specifications. shown in the figure below.

This matrix dis Iteration numb Equilibrium err Spec error.	ber, Step size,	here	column te e. You can files by pic con.	view the	e pres	sure	or flow
Column: Atmos Tow Design Connections Monitor Specs Specs Summary Subcooling Notes	Optional Checks           Input Summary           Iter         Step           4         1.0000           5         1.0000           6         1.0000           7         1.0000           8         1.0000	Basis-1 / Peng-Robinso           View Initial Estimates           Ilibrium         Heat / Spec           1003138         0.002816           1000614         0.001218           1000013         0.000237           1000005         0.000107	Profile     Temp     C Press     C Flows		ature us. Tray	Position from	
	Specific <u>a</u> tions	Specified Value	Current Value	Wt. Error	Activo	Estimate	Current 🔺
	Reflux Ratio	1 000		-0.6308	Active		
	Distillate Rate	2.000e+004 barrel/day	2.00e+004	0.0000		<u> </u>	
	Vap Prod Rate	0.0000 barrel/day	3.16e-003	0.0000	, V	ম	<u> </u>
	WasteH20	<empty></empty>	<empty></empty>	<empty></empty>		<b>N</b>	
	KeroSS Prod Flow	1.300e+004 barrel/day	1.30e+004	0.0000	V	<b>V</b>	N
	KeroSS BoilUp Ratio	0.7500	0.545	0.2054		N	
	DieselSS Prod Flow	1.700e+004 barrel/day	1.70e+004	0.0001	V	V	N
	AGOSS Prod Flow	5000 barrel/day	5.00e+003	0.0002		<b>v</b>	V -
Design Parameter		Add Spec Group		date Inactive Reactions	Degre	es of Free	dom 0
						_	
		1	Conve				

The converged temperature profile is currently displayed in the upper right corner of the view. To view the pressure or flow profiles, select the appropriate radio button.

9. Click on the **Performance** tab, then select the **Column Profiles** or **Feed/Products** page to see a more detailed stage summary.

Column Profiles Feeds/Products	eboil Ratio	0.6325 Temperature [F] 165.5	<ul> <li>Flows</li> <li>Pressure</li> <li>[psia]</li> </ul>	C Energy Net Liquid	C Molar	C M <u>a</u> ss (	• Liq⊻ol
Feeds/Products r( Plots 1. 2. 3. 4. 5.	_Main TS	[F] 165.5		Net Liquid	NI 111		
Plots 1. 2. 3. 4. 5.	_Main TS	165.5	[psia]		Net Vapour	Net Feed	Net Draws 🔺
Plots 1. 2. 3. 4. 5.	_Main TS			[barrel/day]	[barrel/day]	[barrel/day]	[barrel/day]
2 3 4 5			19.70	7221.94	1		20862
3 4 5	Main TS	297.4	28.70	92972.1	28083.7	50002	
4		334.2	28.84	55537.7	63832.1		50002
5	Main TS	355.9	28.99	58098.9	76399.4		
	Main TS	367.0	29.13	58529.6	78960.6		
	Main TS	375.0	29.27	58165.3	79391.3		
[6]	Main TS	382.2	29.41	57277.4	79027.1		
7.	Main TS	389.7	29.56	55848.2	78139.1		
8	Main TS	398.7	29.70	53812.5	76709.9	3307.6	
9	Main TS	409.3	29.84	34904.6	71366.6		16308
1	DMain TS	424.0	29.99	33248.1	68766.9		
1	1Main TS	435.9	30.13	32135.2	67110.4		
1:	2Main TS	445.0	30.27	31142.1	65997.5		
1:	3Main TS	452.2	30.41	29968.3	65004.4		
1.	4Main TS	458.7	30.56	28236.7	63830.6		
1!	5Main TS	466.0	30.70	25125.2	62099.0		
11	6Main TS	476.8	30.84	71275.0	58987.5	34743	
1	7Main TS	507.4	30.99	16986.5	70394.3		51539
1	BMain TS	547.8	31.13	13612.4	67645.0		
1:	9Main TS	565.5	31.27	11840.8	64270.9		
2	DMain TS	573.0	31.41	10480.2	62499.4		
2	1Main TS	577.2	31.56	53248.5	61138.7	32188	
2	2Main TS	607.5	31.70	17762.4	71719.1		37018
2	3Main TS	630.6	31.84	15652.2	73251.0		
2.	4Main TS	640.2	31.99	13530.2	71140.7		
2	5 Main TS	645.7	32.13	11548.7	69018.7		
2	6_Main TS	650.0	32.27	9180.89	67037.3		
10	7 1.1 . 10	0545	22.44	000001	C 4000 4		

The **Column Profiles** page appears below.

In the Basis group near the top of the view, select the **Liq Vol** radio button to examine the tray vapour and liquid flows on a volumetric basis.

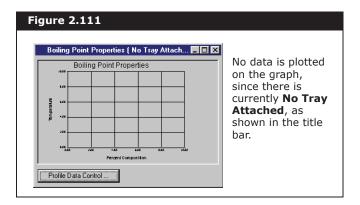
# Viewing Boiling Point Profiles for the Product Stream

You can view boiling point curves for all the product streams on a single graph:

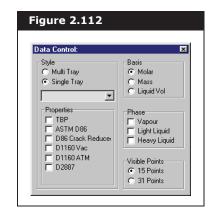
1. On the **Performance** tab, click on the **Plots** page.

Performance Summary Column Profiles Feeds/Products Plots	Tray by Tray Properties       View Graph       Column Tray Ranges         Pressure Flow Transport Properties       View Table       Column Tray Ranges         Composition       View Table       All         K Values       Live Updates       From/To         Boling Point Assay       View Graph       From/To         Boling Point Assay       View Graph       View Table         User Properties       View Table       View Table

- 2. In the Assay Curves group, select Boiling Point Assay.
- 3. Click the **View Graph** button, and the **Boiling Point Properties** view appears.



4. Click the **Profile Data Control** button, and the **Data Control** view appears as shown below.



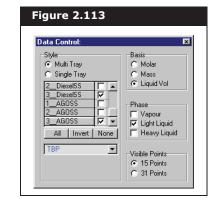
You can view boiling point properties of a single tray or multiple trays. The boiling point properties of all stages, from which products are drawn, are important for this Tutorial.

- 5. Select the **Multi Tray** radio button in the **Style** group. The **Data Control** view is modified, showing a matrix of column stages with a checkbox for each stage.
- 6. Activate the following stages by clicking on the corresponding checkboxes:
  - Condenser (Naphtha product stage)
  - 29\_Main TS (Residue)
  - KeroSS\_Reb (Kerosene)
  - 3\_DieselSS (Diesel)
  - 3\_AGOSS (AGO)

The TBP profile for the light liquid phase on each stage can be viewed, on a liquid volume basis.

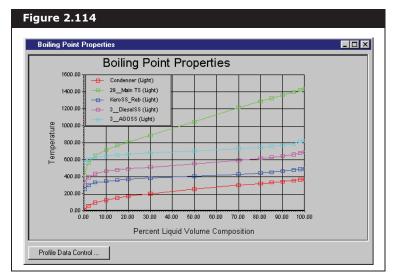
- 7. Select **TBP** in the drop-down list under the tray matrix in the **Style** group.
- 8. In the **Basis** group, select the **Liquid Vol** radio button.
- 9. Activate the **Light Liquid** checkbox in the Phase group to activate it.
- Leave the Visible Points at its default setting of 15 Points. You can display more data points for the curves by selecting the 31 Points radio button.

The independent (xaxis) variable is the Assay Volume Percent, while the dependent (yaxis) variable is the TBP in °C. The completed **Data Control** view is shown below.



- 11. Click on the **Close** icon **I** to close the Data Control view. You return to the Boiling Point Properties view, which now displays the TBP curves.
- 12. Make the **Boiling Point Properties** view more readable by clicking the **Maximize** icon in the upper right corner of the view, or by clicking and dragging its border to a new view size.

The Boiling Point Properties view is shown below.



13. When you are finished viewing the profiles, click the **Close** icon.

#### Moving to the Column Sub-Flowsheet

When considering the column, you might want to focus only on the column sub-flowsheet. You can do this by entering the column environment.

Move the graph legend by double-clicking inside the plot area, then click and drag the legend to its new location.

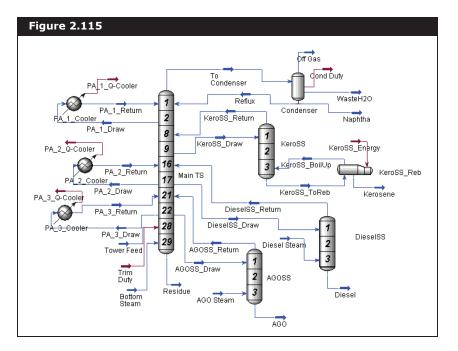






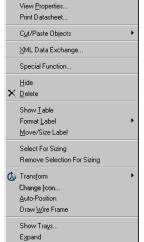
- 1. Click the **Column Environment** button at the bottom of the column property view.
- 2. While inside the column environment, you might want to:
  - view the Column sub-flowsheet PFD by clicking the **PFD** icon.
  - view a Workbook of the Column sub-flowsheet objects by clicking the **Workbook** icon.
  - access the "inside" column property view by clicking the Column Runner icon. This property view is essentially the same as the "outside", or Main Flowsheet, property view of the column.

The Column sub-flowsheet PFD is shown below.





Zoom All icon



Object Inspect menu

### Customizing the Column PFD

You can customize the PFD shown above by re-sizing the column and "hiding" some of the column trays to improve the overall readability of the PFD. To hide some of the trays in the main column:

- 1. Click the **PFD** icon to ensure the column **PFD** is active.
- 2. Click the **Maximize** icon in the upper right corner of the **PFD** view to make it full-screen.
- 3. Click the **Zoom All** icon at the bottom left of the **PFD** view to fill the re-sized **PFD** view.
- 4. Object inspect (right-click) the main column tray section and the object inspection menu appears.
- 5. Select **Show Trays** from the object inspection menu. The Stage Visibility view appears.

- 6. Select the **Selected Expansion** radio button.
- 7. Click the Check All button.
- 8. Hide stages 4, 5, 6, 11, 12, 13, 14, 24, 25, and 26 by deactivating their **Shown** checkboxes.

Stage Visibility		2
Tray Section Representatio	n	
C Eull Expansion	Selected E	xpansion
Selected Control		
Stage	Shown 🔺	Check <u>A</u> ll
1Main TS		
2Main TS		Uncheck All
3Main TS		
4Main TS		
5Main TS		
6Main TS		
7Main TS		
8Main TS		
9Main TS		
10 Main TS		

- Click the **Close** icon on the Stage Visibility view to return to the PFD. The routing of some streams in the **PFD** can be undesirable. You can improve the stream routing by completing the next step.
- 10. From the **PFD** menu item, select **Auto Position All**, and UniSim Design rearranges the **PFD** in a logical manner.

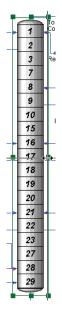
#### Enlarge Icon

The next task in customizing the **PFD** is to enlarge the icon for the main column:

- 1. Click on the icon for the main tray section (Main TS).
- 2. Click the **Size** icon on the PFD button bar, and a frame with eight sizing handles appears around the tray section icon.
- 3. Place the cursor over the handle at the middle right of the icon, and the cursor changes to a double-ended sizing arrow.
- 4. With the sizing cursor visible, click and drag to the right. An outline appears, showing what the new icon size is when you complete the next step.
- 5. When the outline indicates a new icon size of about 1.5 to 2 times the width of the original size, release the button. The tray section icon is now re-sized.
- 6. Click the **Size** icon again to return to **Move** mode.

The final task is to customize the **PFD** by moving some of the streams and operation labels (names) so they do not overlap. To move a label:





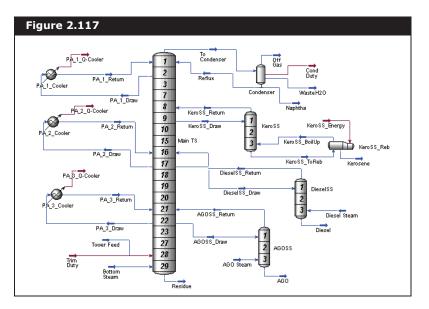
- 7. Click on the label you want to move.
- 8. Right-click and select **Move/Size Label**.
- 9. Move the label to its new position by clicking and dragging it, or by pressing the arrow keys.

You can also move the icon on its own simply by clicking and dragging it to the new location.

- 10. When you are finished working with the maximized Column **PFD**, click the **Restore** icon **I** for the **PFD** (not for the UniSim Design Application view) in the upper right corner of the view of the PFD. The **PFD** returns to its previous size.
- 11. You can manually resize the view, and expand the **PFD** to fill the new size by again clicking the **Zoom All** icon in the lower left corner of the **PFD** view.

For more information on customizing the PFD, refer to **Section 7.24 - PFD** in the **UniSim Design User Guide**.

The customized **PFD** appears below.

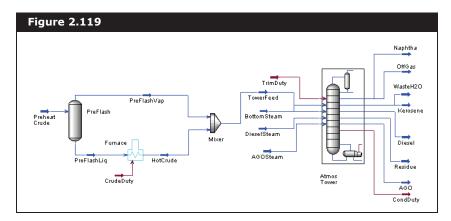


2-86

Name	Reflux	To Condenser	Residue	Naphtha	OffGas	BottomSteam
Vapour Fraction	0.0000	1.0000	0.0000	0.0000	1.0000	1.0000
Temperature [F]	165.5	297.4	657.4	165.5	165.5	375.0
Pressure [psia]	19.70	28.70	32.70	19.70	19.70	150.0
Molar Flow [MMSCFD]	4.884	24.77	8.038	13.54	3.934e-006	3.791
Mass Flow [lb/hr]	7.288e+004	2.875e+005	5.524e+005	2.021e+005	2.940e-002	7500
Liquid Volume Flow [barrel/day]	7212	2.807e+004	4.503e+004	2.000e+004	3.089e-003	514.6
Heat Flow [Btu/hr]	-6.323e+007	-2.618e+008	-2.828e+008	-1.754e+008	-34.08	-4.222e+007
Name	TowerFeed	WasteH20	Kerosene	KeroSS_Draw	KeroSS_Return	KeroSS_BoilUp
Vapour Fraction	0.6035	0.0000	0.0000	0.0000	1.0000	1.0000
Temperature [F]	641.8	165.5	451.6	409.3	427.1	451.6
Pressure [psia]	65.00	19.70	29.84	29.84	29.84	29.84
Molar Flow [MMSCFD]	34.66	6.348	6.053	7.803	1.749	3.302
Mass Flow [lb/hr]	1.142e+006	1.256e+004	1.408e+005	1.760e+005	3.516e+004	7.267e+004
Liquid Volume Flow [barrel/day]	1.000e+005	861.6	1.300e+004	1.631e+004	3307	6754
Heat Flow [Btu/hr]	-5.765e+008	-8.438e+007	-9.579e+007	-1.250e+008	-2.171e+007	-4.348e+007
Name	KeroSS_ToReb	DieselSteam	Diesel	DieselSS_Draw	DieselSS_Retur	AGOSteam
Vapour Fraction	0.0000	1.0000	0.0000	0.0000	1.0000	1.0000
Temperature [F]	441.3	300.0	479.7	507.4	499.0	300.0
Pressure [psia]	29.84	50.00	30.99	30.99	30.99	50.00
Molar Flow [MMSCFD]	9.355	1.517	5.926	7.869	3.460	1.264
Mass Flow [lb/hr]	2.135e+005	3000	1.897e+005	2.390e+005	5.232e+004	2500
Liquid Volume Flow [barrel/day]	1.975e+004	205.8	1.700e+004	2.153e+004	4739	171.5
Heat Flow [Btu/hr]	-1.468e+008	-1.697e+007	-1.248e+008	-1.520e+008	-4.418e+007	-1.414e+007
Name	AGO	AGOSS_Draw	AGOSS_Return	PA_1_Draw	PA_1_Return	PA_2_Draw
Vapour Fraction	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000
Temperature [F]	568.5	607.5	596.2	334.2	154.0	507.4
Pressure [psia]	31.70	31.70	31.70	28.84	28.84	30.99
Molar Flow [MMSCFD]	1.329	2.010	1.945	27.93	27.93	10.96
Mass Flow [lb/hr]	5.730e+004	7.974e+004	2.494e+004	5.262e+005	5.262e+005	3.329e+005
Liquid Volume Flow [barrel/day]	5001	7009	2180	5.000e+004	5.000e+004	3.000e+004
Material Streams Comp	ositions Energy	Streams Unit C	)ps			
Main TS					Fluid Pkg All	-

12. To view the workbook for the column, click the **Workbook** icon.

- 13. When you are finished working in the Column environment, return to the Main Flowsheet by clicking the **Enter Parent Simulation Environment** icon.
- 14. Open the **PFD** for the Main Flowsheet, then select **Auto Position All** from the **PFD** menu item. UniSim Design arranges the Main Flowsheet **PFD** in a logical manner according to the layout of the flowsheet.





Enter Parent Simulation Environment icon

The **PFD** shown in the **Figure 2.119** has been manually rearranged by moving some of the stream icons, and by enlarging the furnace icon.

## 2.2.9 Viewing and Analyzing Results

1. Open the **Workbook** to access the calculated results for the Main Flowsheet. The **Material Streams** tab of the **Workbook** appears below.

Name	Preheat Crude	BottomSteam	DieselSteam	AGOSteam	PreFlashVap	PreFlashLig	HotCrude	TowerFeed
/apour Fraction	0.0986	1.0000	1.0000	1.0000	1.0000	0.0000	0.5483	0.6035
emperature [F]	450.0	375.0	300.0	300.0	450.0	450.0	650.0	641.8
Pressure (psia)	75.00	150.0	50.00	50.00	75.00	75.00	65.00	65.00
violar Flow [MMSCFD]	34.66	3.791	1.517	1.264	3.418	31.25	31.25	34.66
Mass Flow [lb/hr]	1.142e+006	7500	3000	2500	4.886e+004	1.093e+006	1.093e+006	1.142e+006
iquid Volume Flow [barrel/day]	1.000e+005	514.6	205.8	171.5	4871	9.513e+004	9.513e+004	1.000e+005
Heat Flow [Btu/hr]	-7.630e+008	-4.222e+007	-1.697e+007	-1.414e+007	-2.991e+007	-7.331e+008	-5.466e+008	-5.765e+008
Name	OffGas	Naphtha	WasteH20	Residue	Kerosene	Diesel	AGO	** New **
/apour Fraction	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
Temperature [F]	165.5	165.5	165.5	657.4	451.6	479.7	568.5	
Pressure (psia)	19.70	19.70	19.70	32.70	29.84	30.99	31.70	
Molar Flow [MMSCFD]	3.934e-006	13.54	6.348	8.038	6.053	5.926	1.329	
Mass Flow [lb/hr]	2.940e-002	2.021e+005	1.256e+004	5.524e+005	1.408e+005	1.897e+005	5.730e+004	
_iquid Volume Flow [barrel/day]	3.089e-003	2.000e+004	861.6	4.503e+004	1.300e+004	1.700e+004	5001	
Heat Flow [Btu/hr]	-34.08	-1.754e+008	-8.438e+007	-2.828e+008	-9.579e+007	-1.248e+008	-3.362e+007	
=	w Compositions	Energy Stream	s Unit Ops				· ·	
Material Streams P.T.Flo								

#### Using the Object Navigator

Now that results have been obtained, you can view the calculated properties of a particular stream or operation. The **Object Navigator** allows you to quickly access the property view for any stream or unit operation at any time during the simulation.

1. Open the Navigator by doing **one** of the following:

- Press F3.
- From the Flowsheet menu, select Find Object.
- Double-click on any blank space on the UniSim Design Desktop.
- Click the **Object Navigator** icon.



Object Navigator icon

Object Navigator         Flowsheets         Case       [Main]         Atmos Tower       C All         Funce       C All         Streams       C LnitOps         Coupled       C Logicals         C Logicals       C Custom         Setup Custom       Setup Custom	Figure 2.121		
	Flowsheets Case (Main)	Atmos Tower Furnace Mixer	Filter C All C Streams C UnitOps C Logicals C Custom
Build Find View Cancel	Build	Fin <u>d</u> <u>V</u> iew	<u>C</u> ancel

The **Object Navigator** view appears:

The **UnitOps** radio button in the **Filter** group is currently selected, so only **Unit Operations** appear in the list of objects. To open a property view, select the operation in the list and click the View button, or double-click on the operation. You can change which objects appear by selecting a different **Filter** radio button. For example, to list all the streams and unit operations, select the **All** radio button.

You can also search for an object by clicking the Find button. When the Find Object view appears, enter the **Object Name** and click the OK button. UniSim Design opens the property view for the object whose name you entered.

#### 2.2.10 Installing a Boiling Point Curves Utility

Previously, the boiling point profiles for the product streams was viewed using the **Plots** page in the column property view. You can also view boiling point curves for a product stream using UniSim Design' BP Curves Utility. To create a Boiling Point Curves utility for the Kerosene product:

- 1. Open the Navigator using one of the methods described above.
- 2. Select the **Streams** radio button.
- 3. Scroll down the list of Streams and select **Kerosene**.
- 4. Click the **View** button, and the property view for stream **Kerosene** appears.
- 5. On the **Attachments** tab, move to the **Utilities** page of the stream property view.

You can start or end the search string with an asterisk (\*), which acts as a wildcard character. This lets you find multiple objects with one search. For example, searching for VLV\* will open the property view for all objects with VLV at the beginning of their name. 6. Click the **Create** button. The Available Utilities view appears, presenting you with a list of UniSim Design utilities.

Figure 2.122
Available Utilities
CO2 Solids Cold Properties Critical Property Dynamic Depressuring Envelope Hydrate Formation Pripe Sizing Property Table User Property
Add Utility

- 7. Find BP Curves and do **one** of the following:
  - Select BP Curves, then click the Add Utility button.
  - Double-click on BP Curves.
- 8. UniSim Design creates the utility and opens the Boiling Point Curves view.
- On the Design tab, go to the **Connections** page. Change the name of the utility from the default Boiling Point Curves-1 to **Kerosene BP Curves**.
- 10. Change the curve basis to **Liquid Volume** by selecting it from the **Basis** drop-down list.

Design	Name	Kerosene BP Curves	
Connections	Object Type	Stream	
Notes			Coloris Object
	Stream	Kerosene	Select Object
	<u>B</u> asis	Liquid Volume	

A Utility is a separate entity from the stream it is attached to; if you delete it, the stream is not affected. Likewise, if you delete the stream, the Utility remains but cannot display any information until you attach another stream using the Select Object button. 11. You can scroll through the matrix of data to see that the **TBP** ranges from 267°F to 502°F by going to the **Performance** tab and selecting the **Results** page.

Performance	Results				
Results	Cut Point	TBP	ASTM D86	D86 Crack Reduced	AST 🖌
	[%]	[F]	[F]	(F)	
Critical Props	0.00	267.7	326.8	326.8	
Cold Props	1.00	291.8	344.9	344.9	
Plots	2.00	310.2	356.6	356.6	
1 10(3	3.50	322.7	363.5	363.5	
	5.00	336.4	371.4	371.4	
	7.50	343.0	374.4	374.4	
	10.00	349.7	377.8	377.8	
	12.50	360.5	383.5	383.5	
	15.00	362.9	384.4	384.4	
	17.50	368.0	387.0	387.0	
	20.00	372.4	389.2	389.2	
	25.00	379.6	392.8	392.8	
	30.00	385.4	395.8	395.8	
	35.00	391.3	398.8	398.8	
	40.00	397.2	402.0	402.0	
					•
Design Perform	nance Dynamic	. —			

This boiling range predicted by the utility is slightly wider than the ideal range calculated during the Oil characterization procedure for Kerosene, 356°F to 464°F.

Figure 2.	125				
Cut Distributions	Begin T (F)	End T (F)	Fraction	_	
Lt St Run	100.3	158.0	0.033	_	
Naphtha	158.0	356.0	0.172		
Kerosene	356.0	464.0	0.129		
Light Diesel	464.0	554.0	0,089		Ideal boiling
Heavy Diesel	554.0	644.0	0.089	<u> </u>	range calculated
Atm Gas Oil	644.0	698.0	0.049	$\overline{}$	during Oil
Residue	698.0	1441	0.444		5
					Characterization.
				_	
				_	

- Figure 2.126 Boiling Point Curves: Kerosene BP Curves - 🗆 × Dependent Variable BP Curve • Performance Results Critical Props Cold Props Plots 9 €⊞. 350 emperatur ш 1001 Volume Percent (%) Design Performance Dynamics <u>∏</u> <u>I</u>gnored Delete
- 12. Select the **Plots** page on the **Parameters** tab of the utility property view to view the data in graphical format.

To make the envelope more readable, maximize or resize the view.

- 13. When you move to the **Plots** view, the graph legend can overlap the plotted data. To move the legend, double-click anywhere in the plot area then click and drag the legend to its new location.
- 14. When you are finished viewing the **Boiling Point Curves**, click the **Close** icon.

#### Installing a Second Boiling Point Curves Utility

Alternative to using the **Utilities** page of a stream property view, you can also install a utility using the **Available Utilities** view. Another **BP** Curves utility is installed for stream **Residue**. This utility is used for the case study in the next section. To install the utility:

- 1. Do one of the following:
  - press CTRL U.
  - from the **Tools** menu, select **Utilities**.

Notice the name of the utility created previously, Kerosene BP Curves, appears in the Available Utilities view. The Available Utilities view appears.

Available Utilities	
Kerosene BP Curves	Boiling Point Curves CO2 Freeze Dut Co2 Freeze Dut Cold Properties Data Recon Ubility Depressuring Derivative Utility Envelope Utility Hydrate Formation Utility Parametric/LP Utility Prane Utility Pripe Sizing Property Balance Utility
⊻iew Utility	Add Utility
Delete Utility	]

2. Select Boiling Point Curves and click the **Add Utility** button. The Boiling Point Curves view appears, opened to the **Design** tab.

- 3. Change the name from its default Boiling Point Curves-1 to Residue BP Curves.
- 4. Change the Basis to **Liquid Volume** by selecting it in the dropdown list. The next task is to attach the utility to a material stream.

Click the Select Object button, and the Select Process Stream view appears.

Select Process Stream					
Flowsheet	<u>O</u> bject				
Case (Main) Atmos Tower (CDL1	AGD AGD Steam Bottom Steam Cond Duty Crude Duty Diesel Steam Hot Crude Kerosene Naphtha Off Gas PreflashVap PreflashVap PreflashVap PreflashVap Tower Feed Tim Duty WasteH2D	OLJ Object Filter C All C Streams C UnitOps C Logicals C ColumnOps C Custom Custom			

 Select Residue in the Object list, then click the OK button. UniSim Design calculates the boiling point curves. The completed Performance tab appears below.

Performance	Results				
Results	Cut Point	TBP	ASTM D86	D86 Crack Reduced	AST 🔺
Critical Props	[%]	[F]	[F]	[F]	
•	0.00	417.2	458.3	458.3	
Cold Props	1.00	501.8	506.7	500.6	
Plots	2.00	565.0	549.3	539.9	_
	3.50	617.1	587.9	574.2	
	5.00	650.2	613.8	596.3	_
	7.50	689.1	645.5	622.3	
	10.00	720.1	671.5	642.6	
	12.50	746.9	694.7	659.7	
	15.00	771.4	716.0	674.9	
	17.50	793.4	735.5	688.0	
	20.00	813.3	753.4	699.5	
	25.00	851.7	788.3	720.5	_
	30.00	890.4	823.5	739.8	
	35.00	929.7	859.4	757.6	
	40.00	969.5	895.6	774.0	
					•
Design Perfor	mance Dynamic				
Design Tenon	mance Dynamic	<u></u>			

7. Click the **Close** icon on the Residue BP Curves view, and then on the Available Utilities view.

# 2.2.11 Using the Databook

The UniSim Design **Databook** provides you with a convenient way to examine your flowsheet in more detail. You can use the **Databook** to monitor key variables under a variety of process scenarios, and view

Notice that the stream name Residue now appears in the Stream cell. the results in a tabular or graphical format.

- 1. To open the **Databook**, do **one** of the following:
  - press CTRL D.
  - from the **Tools** menu, select **Databook**.

The Databook appears below.

Dbject	N			
	Variable	!	 	
			 	E <u>d</u> it
				Insert
			Ī	Delete

#### Adding Variables to **Databook**

The first step is to add the key variables to the **Databook** using the **Variables** tab. For this example, the Overflash specification is varied and examined to investigate its effect on the following variables:

- D1160 Boiling Temperature for 5% volume cut point of stream Residue
- heat flow of energy stream Trim Duty
- column reflux ratio
- 1. Click the **Insert** button and the Variable Navigator view appears.
- Select the **UnitOps** radio button in the Object Filter group. The Object list is filtered to show unit operations only.
- 3. Select Atmos Tower in the Object list, and the Variable list available for the column appears to the right of the Object list.

The Variable Navigator is used extensively in UniSim Design for locating and selecting variables. The Navigator operates in a left-toright manner—the selected Flowsheet determines the Object list, the chosen Object dictates the Variable list, and the selected Variable determines whether any Variable Specifics are available. 4. Select Reflux Ratio in the Variable list.

igure 2.132				
Variable Navigator				
Flowsheet Case (Main) Atmos Tower (CDL1) Navigator Scope © Flowsheet © Case © Basis	Object WaterH20 Atmos Tower Furnace Mixer PreFlash FeederBlock, AGD Stea FeederBlock, Botom Stte FeederBlock, Diseal ProductBlock, Verosene ProductBlock, Kerosene ProductBlock, Verosene ProductBlock, Verosene ProductBlock, Uff Gas	Variable Product Stream Comp Product Stream Comp Product Stream Comp Reboil Ratio Spec Calc Value Spec Calc Value Spec Ino Spec Is Active Stage Efficiency Stage Heat Flow Stage Lig Comp LigV Stage Lig Comp Mass Stage Lig Comp Mess	V <u>a</u> riable Specifics	<u>D</u> K <u>A</u> dd Object Fjlter G All Streams C UnitOps C Logicals C ColumnOps C Custom Custom
C Utility Variable Description:	ProductBlock_Residue ProductBlock_WaterH2r	Stage Liq UiqVolume   Stage Liq Mass Frac Stage Lin Mole Frac		<u>C</u> ancel

- 5. Click the **Add** button. The variable appears in the Databook and the Variable Navigator view remains open.
- To add the next variable, select the **Streams** radio button in the **Object Filter** group. The **Object** list is filtered to show streams only.
- Scroll down and click on Trim Duty in the Object list, and the Variable list available for energy streams appears to the right of the Object list.
- 8. Select Heat Flow in the Variable list.

The variable name is duplicated in the Variable Description field. If you want, you can edit the default description. To edit the default description:

- 9. Click inside the **Variable Description** field and delete the default name.
- 10. Type a new description, such as **Trim Duty**, and click the **Add** button. The variable now appears in the Databook.

Variable Navigator				
Flowsheet Case (Main) Atmos Tower (COL1)	Object PreFlashWap Preheat Crude Residue Tower Feed Trim Duty WaterH2D Atmos Tower	<u>V</u> ariable Heat Flow Overall UA Power Temperature Approach User Variables Utility flow rate Utility Fluid Cp	Variable Specifics	<u>O</u> K <u>A</u> dd ⊙bject Fjlter ⊙ All ⊙ Streams
Navigator Scope Flowsheet Case Basis Utility	Furnace Mixer PreFlash FeederBlock, AGD Stea FeederBlock, Bottom Str FeederBlock, Dissel Ste FeederBlock, AGD ProductBlock, AGD ProductBlock, AGD ProductBlock, Cassa	Utility Fluid Holdup Utility Inlet Temp Utility maximum flow rate Utility minimum flow rate Utility Outlet Temp		C UnitOps C Logicals C ColumnOps C Custom Custom

- 11. To add the third variable, the ASTM D1160 cut point from the **Residue BP Curves** utility, select the **Utility** radio button in the **Navigator Scope** group.
- 12. Select **Residue BP Curves** in the **Object** list.
- 13. Select ASTM D1160 Vac in the Variable list.
- 14. Select Cut PT-5.00% in the **Variable Specifics** column. This corresponds to the 5% volume cut point.
- 15. In the **Variable Description** field, change the variable name to **ASTM 1160 Vac 5% Residue**, and click the **Close** button.

Variable Navigato	r			
Case	Object	Variable	Variable Specifics	<u>0</u> K
Case	Kerosene BP Curves Residue BP Curves	Acentric Factor Aromatic Mole% ASTM D86 ASTM D2887 ASTM D1160 - Atm ASTM D1160 - Vac ASTM D37 Flash Pt ASTM D93 Flash Pt ASTM D97 Pour Pt Detane Index	Cut Pt-15.00% Cut Pt-17.50% Cut Pt-20.00% Cut Pt-25.00% Cut Pt-35.00% Cut Pt-30.00% Cut Pt-30.00% Cut Pt-40.00%	Add Object Eilter C All C C02 C Cold C Envelope
C Flowsheet C Case C Basis C Utility		Critical Pressure Critical Temperature D86 Crack Reduced Mass Density Mole Weight Napthene Mole% Paraffin Mole% Paraffin Mole%	Cut Pt-45.00% Cut Pt-5.00% Cut Pt-50.00% Cut Pt-55.00% Cut Pt-65.00% Cut Pt-75.00% Cut Pt-75.00% Cut Pt-7.50%	C Hydrate

16. The completed Variables tab of the Databook appears below.

Available Data Entries		
Object	Variable	
Atmos Tower	Reflux Ratio	E <u>d</u> it
Trim Duty Residue BP Curves	Trim Duty ASTM 1160 - Vac 5% Residue	
		Insert
		Delete

#### Create a Data Table

Now that the key variables to the **Databook** have been added, the next task is to create a data table to display those variables:

- 1. Click on the **Process Data Tables** tab.
- Click the Add button in the Available Process Data Tables group. UniSim Design creates a new table with the default name ProcData1.

DataBook	
Available Process <u>D</u> ata Tables ProcData1 View	Individual Process Data Selection Process Data Table ProcData1
<u>A</u> dd Delete €etup	Object         Variable         Show           Atmos Tower         Reflux Ratio         □           Trim Duty         Trim Duty         □           Residue BP Curv         ASTM 1160 - Vac 5% Residu         □
Variables Process Data Tables	Add All Variables Invert All Variables Remove All Variable Strip Charts Data Recorder Case Studies

3. Change the default name from **ProcData1** to **Key Variables** by editing the **Process Data Table** field.

Notice that the three variables added to the **Databook** appear in the matrix on this tab.

4. Activate each variable by clicking on the corresponding **Show** checkbox.

P <u>r</u> ocess Data Tabl	e Key Variables	
Object	Variable	Show
Atmos Tower	Reflux Ratio	<b>N</b>
Trim Duty	Trim Duty	<b>V</b>
Residue BP Curv	ASTM 1160 - Vac 5% Residu	V

5. Click the **View** button to view the new data table, which is shown below.

Key Variables Data 💶 🗖						
Object	Variable	Value	Units			
Atmos Tower	Reflux Ratio	0.7720				
Trim Duty	Trim Duty	3.179e+007	Btu/hr			
Residue BP Cur	ASTM 1160 · Vac 5% Resi	389.6	F			
d l		·	· · ·			

This table is accessed later to demonstrate how its results are updated whenever a flowsheet change is made.

 For now, click the Minimize icon in the upper right corner of the Key Variables Data view. UniSim Design reduces the view to an icon and place it at the bottom of the Desktop.

#### **Recording Data**

Suppose you now want to make changes to the flowsheet, but you would like to record the current values of the key variables before making any changes. Instead of manually recording the variables, you can use the Data Recorder to automatically record them for you.

To record the current values:

1. Click on the **Data Recorder** tab.

<u>A</u> vailable Scenarios		Data	Recorder Da	ta Selection	
	<u>R</u> ecord	Cu	rrent Sce <u>n</u> ario		
1	Add		Object	Variable	Include
-	20		Atmos Tower	Reflux Ratio	0
_	Delete	Re	Trim Duty sidue BP Cui	Trim Duty ASTM 1160 - Vac 5% Residue	
C Table	⊻iew				

When using the Data Recorder, you first create a **Scenario** containing one or more of the key variables, then record the variables in their current state.

 Click the Add button in the Available Scenarios group, and UniSim Design creates a new scenario with the default name Scenario 1. It is required to include all three key variables in this scenario. 3. Activate each variable by clicking on the corresponding Include checkbox.

D . D .					
DataBook		_			_ [
<u>Available Scenarios</u>			-Data Recorder Da	ta Selection	
Scenario 1	<u>R</u> ecord		Current Sce <u>n</u> ario	Scenario 1	
	Add	1	Object	Variable	Include
		1	Atmos Tower	Reflux Ratio	বাব
	Delete		Trim Duty	Trim Duty ASTM 1160 - Vac 5% Residue	
			Residue BP Cui	ASTM TIBU - Vac 5% Residue	
-Avajlable Display					
C Table					
🕫 Graph		1			
		<u>.</u>	,	1	1
Variables Proces	s Data Tables	Strip	p Charts Data R	ecorder Case Studies	

	New Solved St 🗙
Na	me for New State
Sta	ite 1
	<u>0</u> K

2-100

- 4. Click the **Record** button to record the variables in their current state. The New Solved State view appears, prompting you for the name of the new state.
- 5. Change the Name for New State from the default State 1 to 3500 **O.F.** (denoting 3500 bbl/day Overflash). Click the **OK** button and you return to the Databook.
- 6. In the **Available Display** group, select the **Table** radio button.
- 7. Click the **View** button and the Data Recorder appears showing the values of the key variables in their current state.

State	3500 O.F.			
Reflux Ratio	0.7720		 	
Trim Duty [Btu/hr]	3.179e+007		 	
ASTM 1160 - Vac 5%	389.6			

Now you can make the necessary flowsheet changes and these current values remain as a permanent record in the Data Recorder unless you choose to erase them.

8. Click the **Minimize** icon to reduce the **Data Recorder** to an icon.

#### Changing the Overflash Specification

The value of the Overflash specification is going to be changed in the column and the changes is viewed in the process data table:

- 1. Click the **Object Navigator** icon on the toolbar.
- 2. Select the **UnitOps** radio button in the **Filter** group.
- 3. Select **Atmos Tower** and click the **View** button. The **Atmos Tower** property view appears.
- 4. Go to the **Design** tab and select the **Monitor** page.
- 5. Scroll down to the bottom of the **Specifications** table so the **Overflash** specification is visible.

A typical range for the Overflash rate is 3-5% of the tower feed. A slightly wider range is examined: 1.5-7.5%, which translates to 1500-7500 bbl/d.

- 6. Change the **Specified Value** for the **Overflash** specification from its current value of **3500** barrel/day to **1500** barrel/day. UniSim Design automatically recalculates the flowsheet.
- 7. Double-click on the **Key Variables Data** icon to restore the view to its full size. The updated key variables are shown below.

Object Var		Key Variables Data 📃 🗖					
UDJECI Val	riable	Value	Units				
Atmos Tower	Reflux Ratio	0.5858					
Trim Duty	Trim Duty	1.883e+007	Btu/hr				
Residue BP Cur   ASTM 1160	·Vac 5% Resi	377.8	F				

As a result of the change:

- the Trim Duty has decreased
- the Residue D1160 Vacuum Temperature 5% cut point has decreased
- the column reflux ratio has decreased
- 8. Press **CTRL D** to make the **Databook** active again. You can now record the key variables in their new state.
- 9. Move to the **Data Recorder** tab in the Databook.
- 10. Click the **Record** button, and UniSim Design provides you with the default name State 2 for the new state.
- 11. Change the name to 1500 O.F. and click the **OK** button to accept the new name.



Object Navigator icon

2-102

12. Click the **View** button and the **Data Recorder** appears, displaying the new values of the variables.

igure 2.143					
Data Recorder -	Main				
Data frecorder - I	main				
State	3500 O.F.	1500 O.F.			
Reflux Ratio	0.7722	0.5857			
Trim Duty [Btu/hr]	3.172e+007	1.883e+007			
	389.5	377.8			

 Record the process variables for **Overflash** rates of **5500** and **7500** barrels/day. Enter names for these variable states of **5500 O.F.** and **7500 O.F.**, respectively. The final **Data Recorder** appears below.

State	3500 O.F.	1500 O.F.	5500 O.F.	7500 O.F.		
Reflux Ratio	0.7722	0.5857	0.9417	1.099		
Trim Duty [Btu/hr]	3.172e+007	1.883e+007	4.304e+007	5.326e+007		
ASTM 1160 - Vac 5%	389.5	377.8	398.6	405.7		

14. Save your case by doing one of the following:

- press CTRL S.
- from the File menu, select Save.
- click the **Save** icon.

# 2.3 Dynamic Simulation

In this tutorial, the dynamic capabilities of UniSim Design are incorporated into a basic steady state oil refining model. A simple fractionation facility produces naphtha, kerosene, diesel, atmospheric gas oil, and atmospheric residue products from a heavy crude feed. In the steady state refining tutorial, preheated crude was fed into a preflash drum which separated the liquid crude from the vapour. The liquid crude was heated in a furnace and recombined with the vapour. The combined stream was then fed to the atmospheric crude column for fractionation. The dynamic refining tutorial only considers the crude column. That is, the crude preheat train is deleted from the flowsheet



This complete dynamic case has been pre-built and is located in the file **DynTUT2.usc** in your UniSim Design\Samples directory.