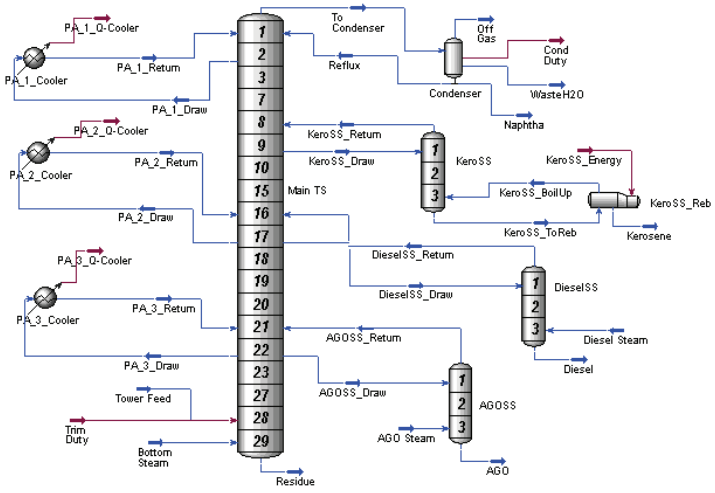
flowsheet appears below.

Figure 2.2



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.

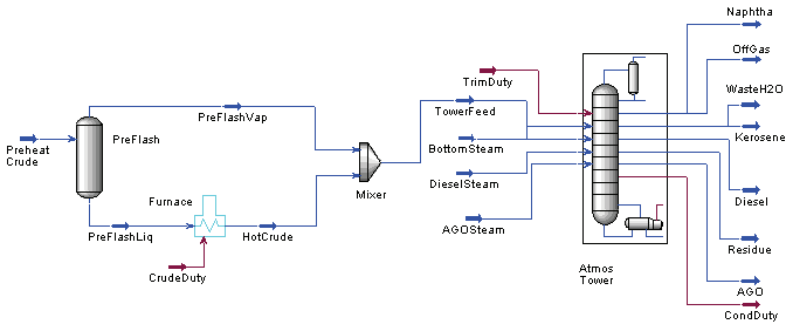
2.2 Steady State Simulation

2.2.1 Process Description

This example models a crude oil processing facility consisting of a pre-fractionation train used to heat the crude liquids, and an atmospheric crude column to fractionate the crude into its straight run products. The

Main Flowsheet for this process appears in the following figure.

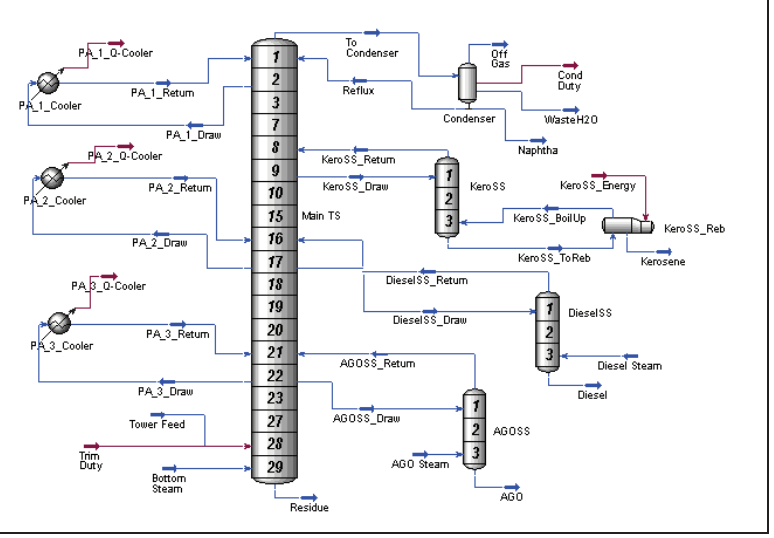
Figure 2.3



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

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.

Figure 2.4



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

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 steam-stripped DieselSS and AGOSS side strippers, respectively.

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

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.

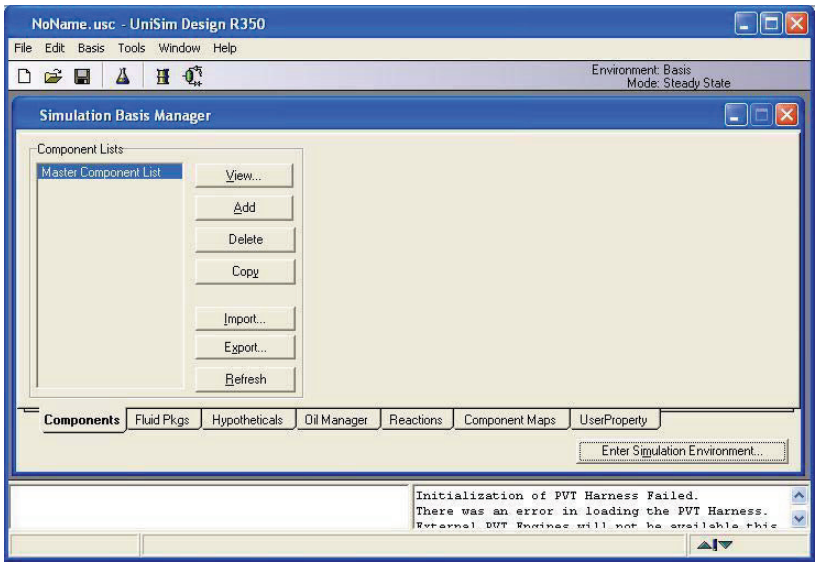
The Workbook is used to build the first part of the flowsheet, from specifying the feed conditions through to installing the pre-flash separator. The PFD is then used to install the remaining operations, from the crude furnace through to the column.

2.2.2 Setting Your Session Preferences

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

Manager view appears.

Figure 2.5



Your first task is to set your Session Preferences.

2. From the **Tools** menu, select **Preferences**.

   The Session Preferences view appears.

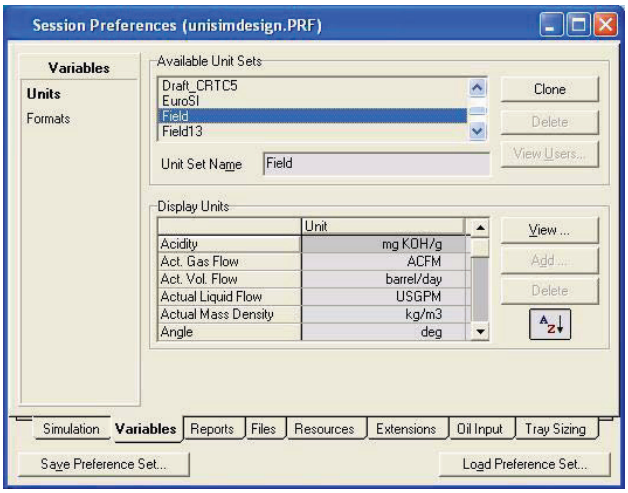
   The most important preference you will set is the unit set. UniSim Design does not allow you to change any of the default unit sets listed, however, you can create a new unit set by cloning an existing one. In this tutorial you will create a new unit set based on the UniSim Design Field set and customize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

Figure 2.6



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.

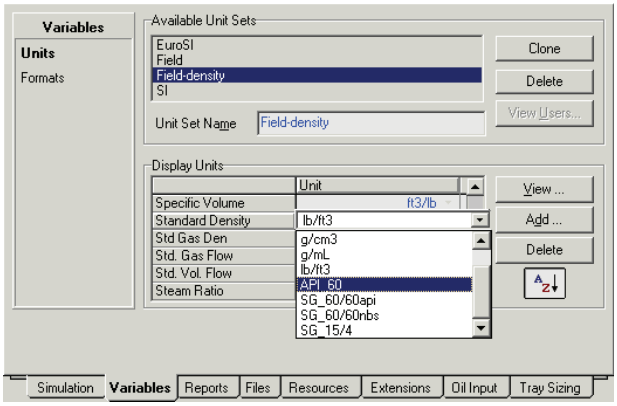
   The current default unit for **Standard Density** is lb/ft3. A more appropriate unit for this example is API_60.

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.

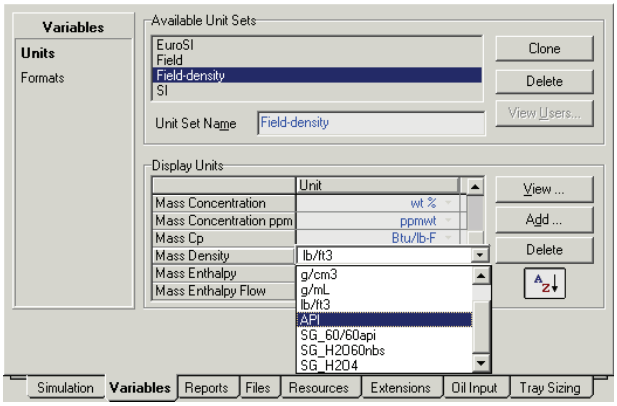
10. In the unit list, select **API_60**.

Figure 2.7



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

Figure 2.8



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

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

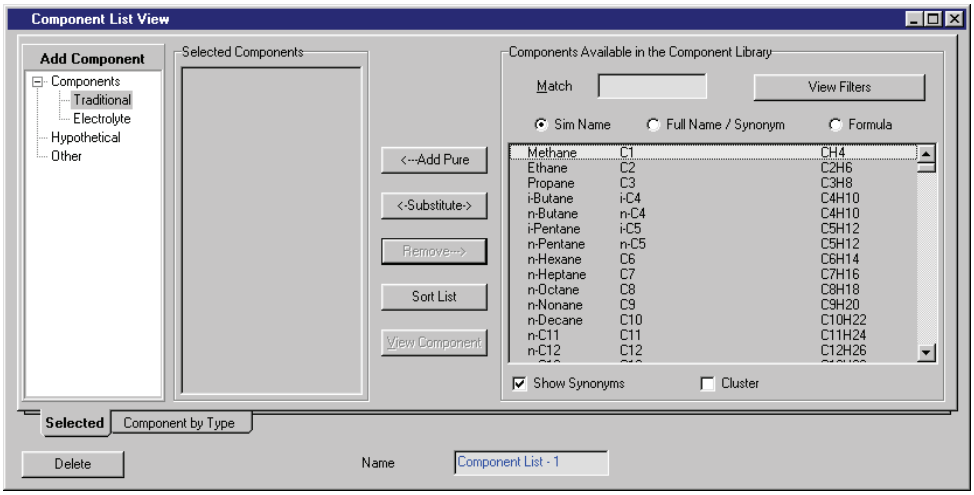
Selecting Components

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

Design pure component library into the component list.

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

Figure 2.9



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.

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.

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.

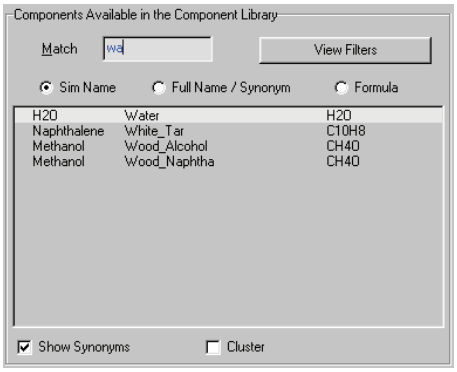
   First, you will add **H2O** using the match feature.

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

You can also move to the Match field by pressing **ALT M**.

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.

Figure 2.10



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.

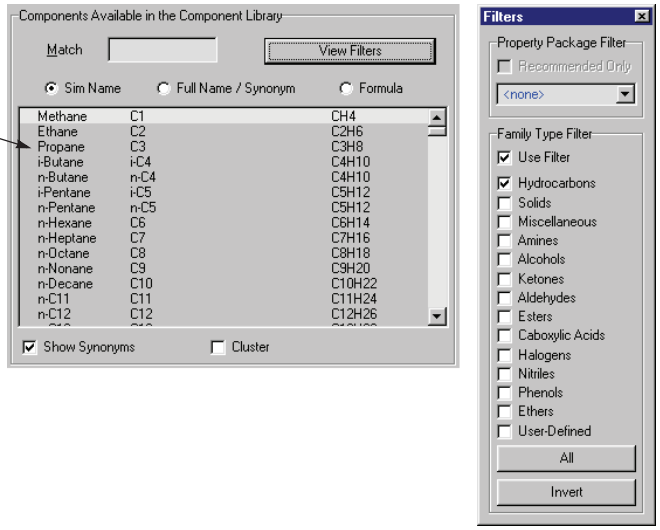
   You can also use the Family Filter to display only those components belonging to certain families. Next, you will add Propane to the component list using a Family Filter:
7. Ensure the **Match** field is empty, and click the **View Filter** button. The Filters view appears as shown on the left.
8. On the Filters view, check the **Use Filter** checkbox to activate the Family Filter.

Filters view

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

Figure 2.11

On the Component View view, 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.

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:

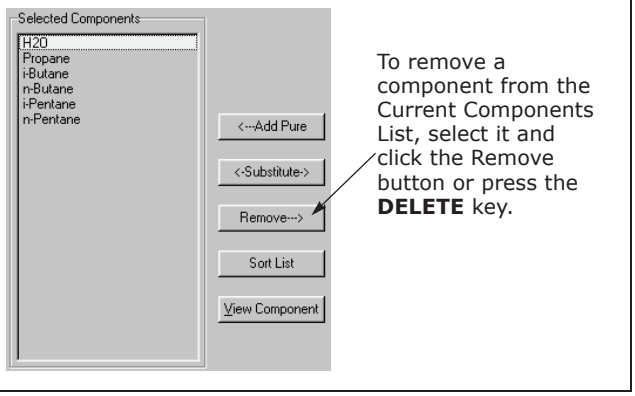
    - Hold down the **SHIFT** key and click on the last component, in this case n-C5. All components i-C4 through n-C5 are now selected. Release the **SHIFT** key.
    - Click and drag from i-C4 to n-C5. Components i-C4 through to n-C5 are selected.

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

To select non-consecutive components, use the **CTRL** key.

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

Figure 2.12



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.

The Simulation Basis Manager allows you to create, modify, and otherwise manipulate fluid packages in your simulation case. Most of the time, as with this example, you require only one fluid package for your entire simulation.

UniSim Design has created a fluid package with the default name Basis-1. You can change the name of this fluid package by typing a new name in the **Name** field at the bottom of the view.

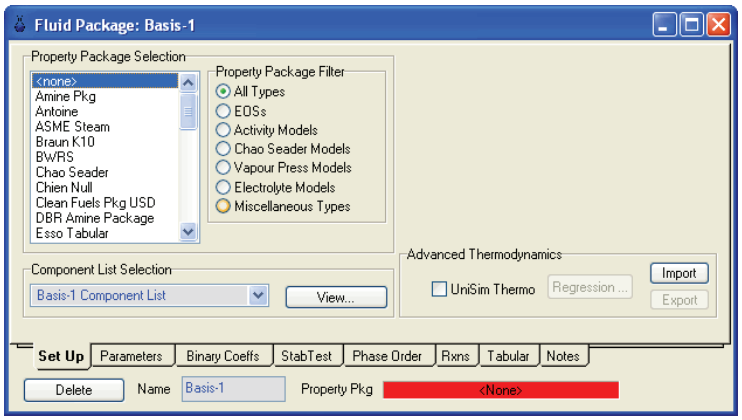
A fluid package contains the components and property methods that UniSim Design will use in its calculations for a particular flowsheet. Depending on what is required, a fluid package can also contain other information, such as a petroleum fluid characterization.

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

Package: Basis-1 view appears.

Figure 2.13



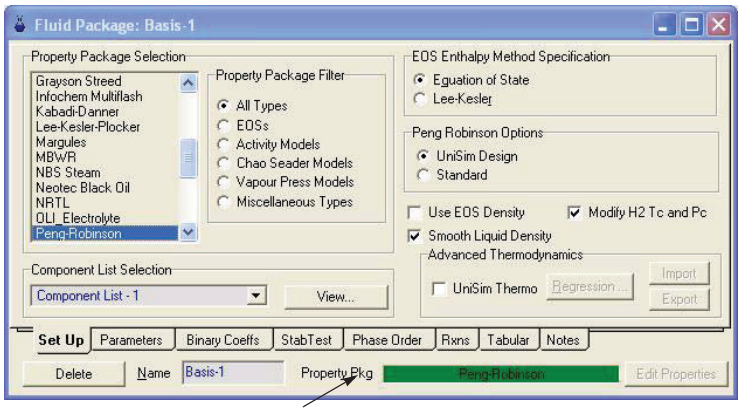
This view is divided into a number of tabs that allow you to supply all the information necessary to completely define the fluid package. For this tutorial, however, only the Set Up tab is used.

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.

Figure 2.14



The Property Pkg indicator now indicates **Peng Robinson** is the current property package for this fluid package.

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 drop-down list to find the name of any component lists you had created (currently empty).

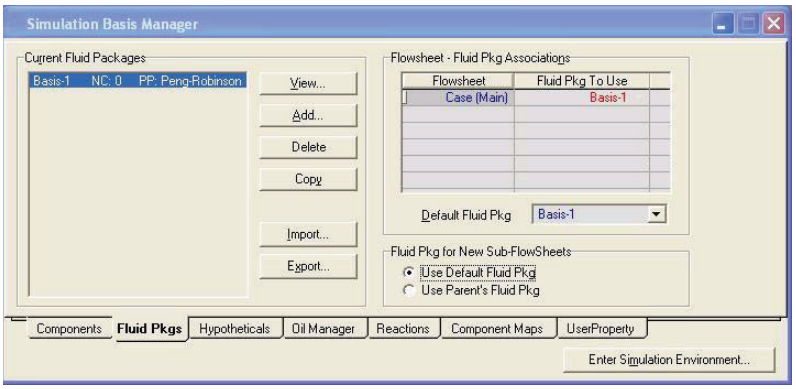
The View button opens the Component List View view of the selected component list.

If 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.

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.

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

Figure 2.15



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 Volume 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

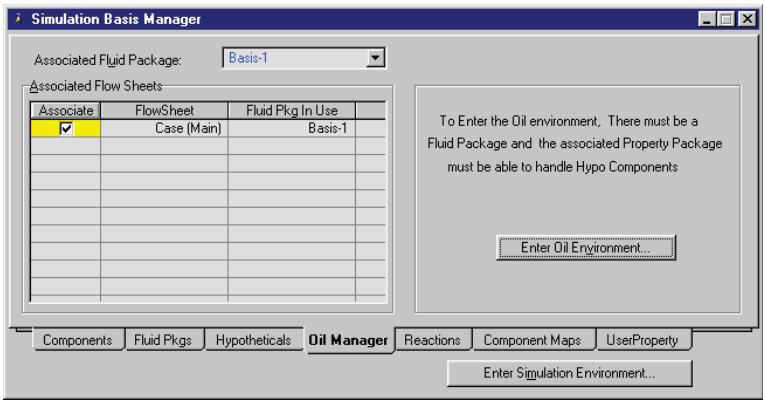
Accessing the Oil Environment

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

1. On the Simulation Basis Manager view, click the **Oil Manager** tab.

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.

Figure 2.16



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.

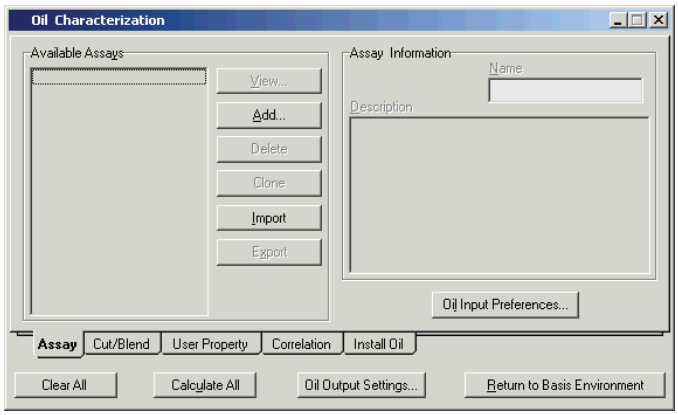
Oil Environment icon

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.

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.

The Oil Characterization view appears.

Figure 2.17



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

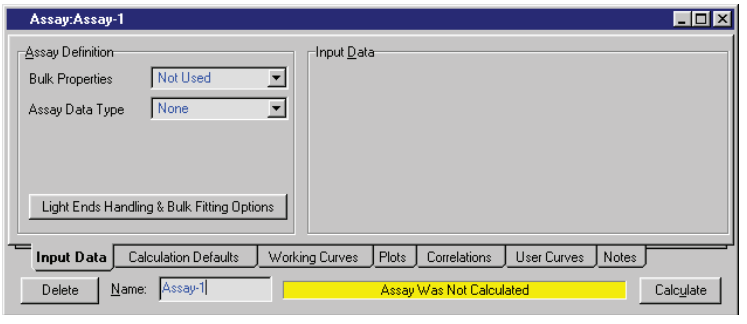
1. Supply data to define the Assay.
2. Cut the Assay into hypothetical components by creating a Blend.
3. Install the hypothetical components into the fluid package.

Defining the Assay

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

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.

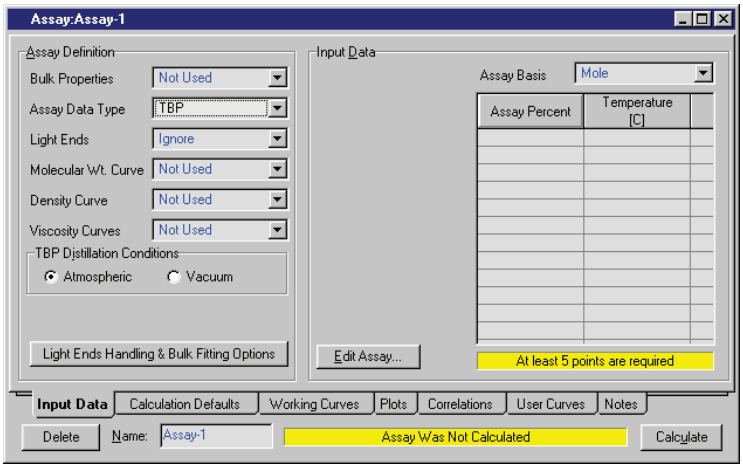
Figure 2.18



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.

2. From the Assay Data Type drop-down list, select TBP. The view is customized for TBP data.

Figure 2.19

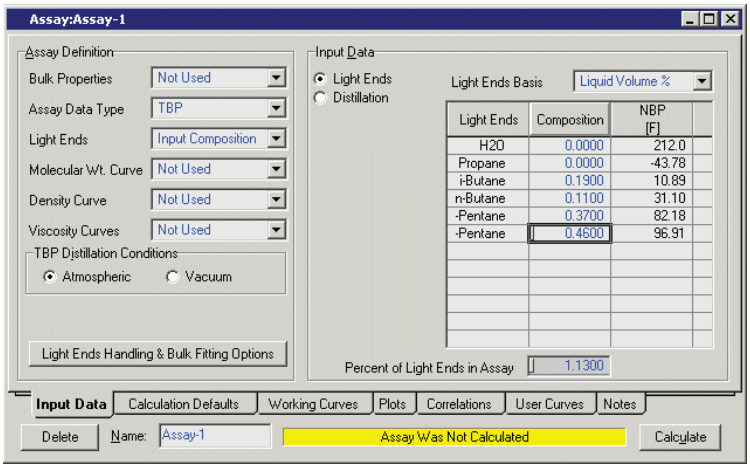


The next task is to enter the composition of the Light Ends in the Assay.

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.

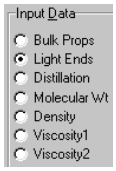
Figure 2.20



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.



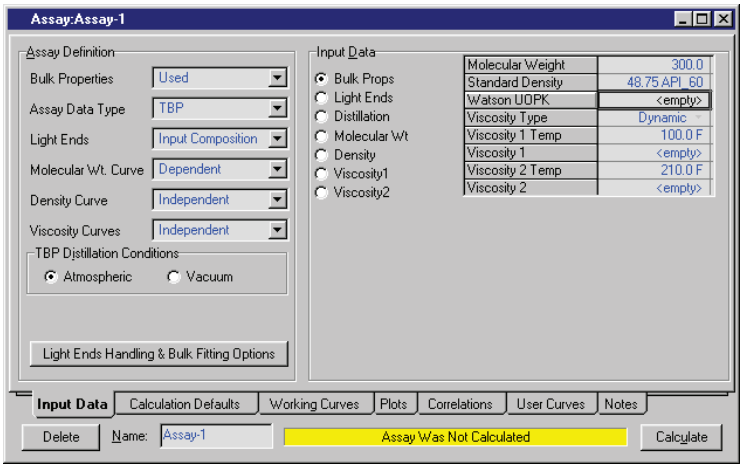
In the next few sections, you will enter the following laboratory assay data:

- 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)

Entering Bulk Property Data

1. Select the **Bulk Props** radio button, and the bulk property table appears to the right of the radio buttons.
2. 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.

Figure 2.21



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.

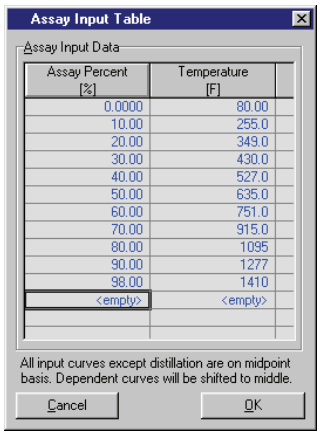
Entering Boiling Temperature (TBP) Data

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.

Figure 2.22



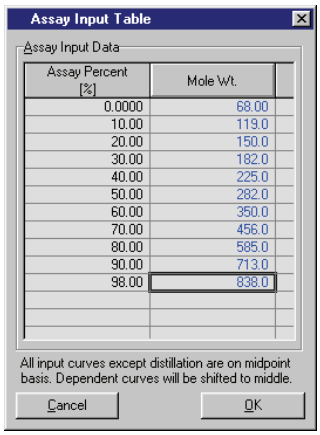
11. Click the **OK** button to return to the Assay property view.

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.

Figure 2.23

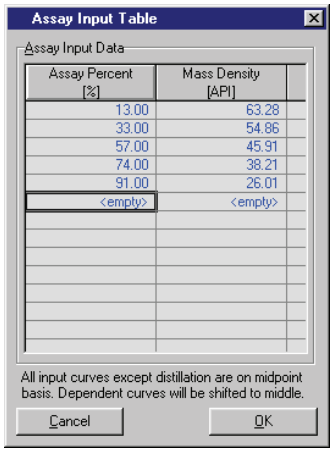


6. Click the **OK** button when you are finished.

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.

Figure 2.24



Entering Viscosity Data

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

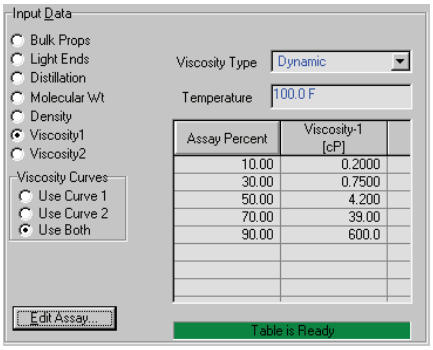
matrix appears.

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.

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

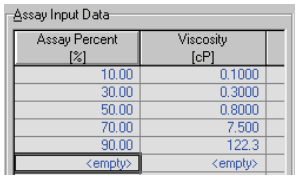
4. Input the Viscosity 1 assay data as shown here. This viscosity curve corresponds to Temperature 1, 100°F.

Figure 2.25



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

Figure 2.26

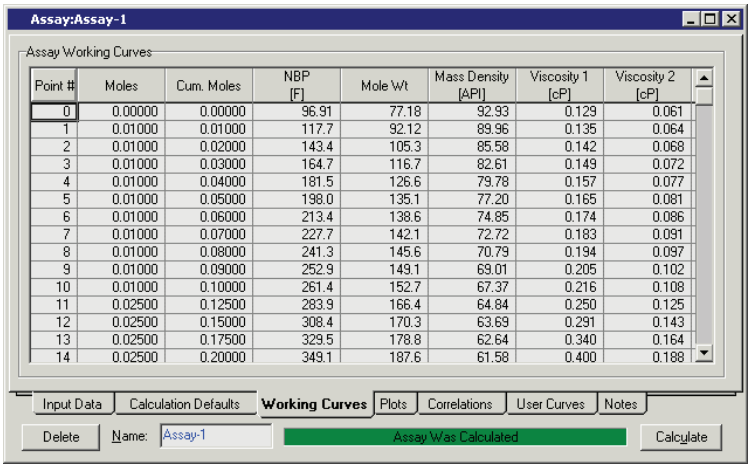


The Assay is now completely defined based on our available data.

7. 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.

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

Figure 2.27



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

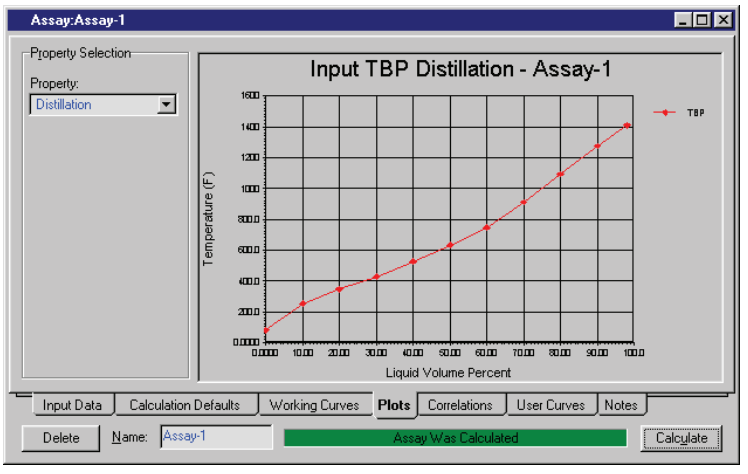
The plot view can be re-sized to make the plot more readable. To re-size the view, do **one** of the following:

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

Maximize icon

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.

Figure 2.28



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

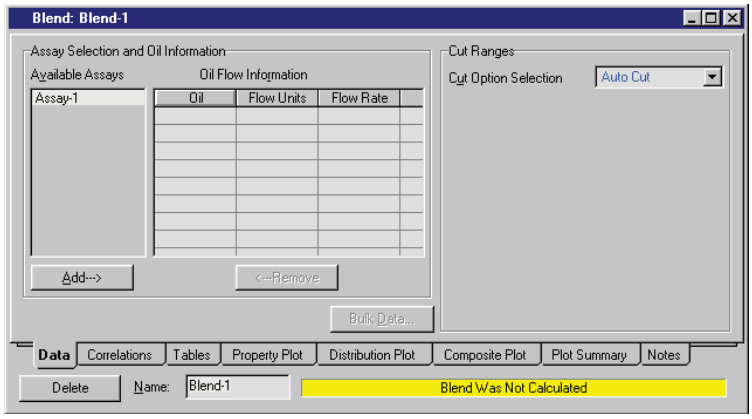
10. Close the Assay view to return to the Oil Characterization view.

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.

Figure 2.29



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:

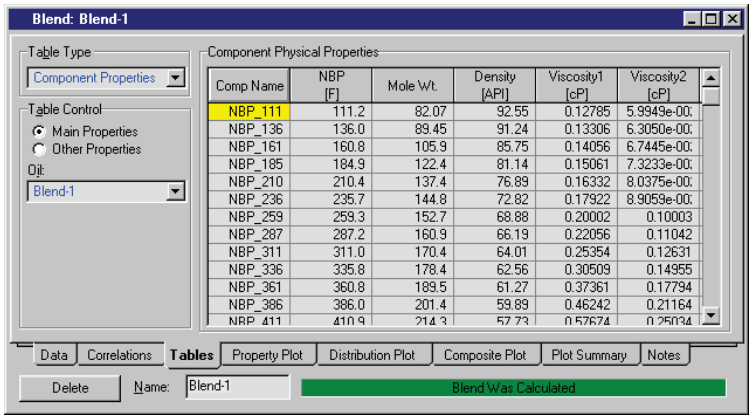
   - IBP to 800°F: 25°F per cut, generating [(800-IBP)/25] hypocomponents
   - 800 to 1200°F: 50°F per cut, generating 8 hypocomponents
   - 1200 to 1400°F: 100°F per cut, generating 2 hypocomponents

   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.

Figure 2.30



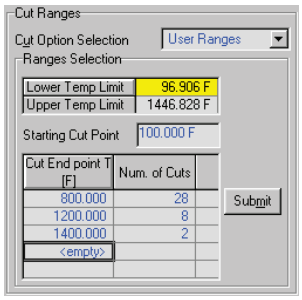
These components could be used in the simulation. Suppose, however, that you do not want to use the IBP as the starting point for the first temperature range. You could specify another starting point by changing the Cut Option to **User Ranges.** For illustration purposes, 100°F is used as the initial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

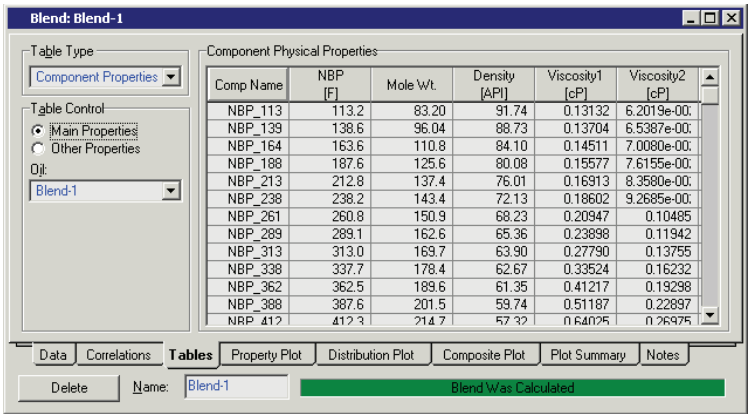


12. Once you have entered the data, click the **Submit** button to calculate the Blend based on the current initial cut point and range values. The message Blend Was Calculated appears in the status bar.

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.

13. Click the **Tables** tab to view the properties of the petroleum hypocomponents.

Figure 2.32



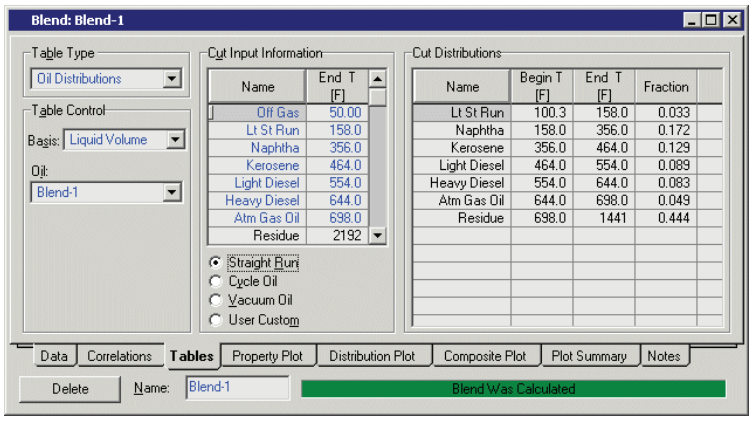
Use the vertical scroll bar to view the components which are not currently visible in the **Component Physical Properties** table.

Viewing the Oil Distributions

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

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

Figure 2.33



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.

2. 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

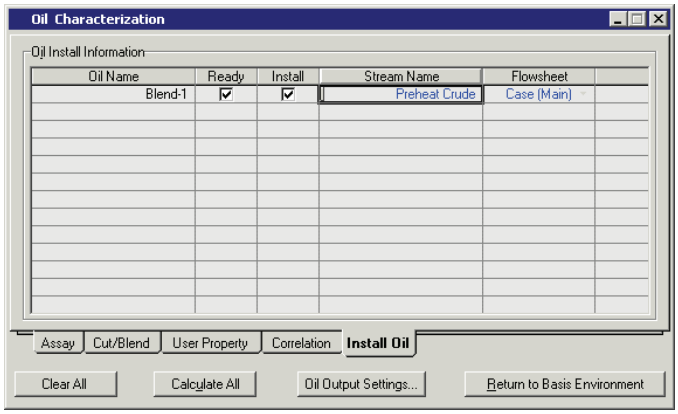
The last step in the oil characterization procedure is to install the oil, which accomplishes the following:

- 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.

Figure 2.34



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.

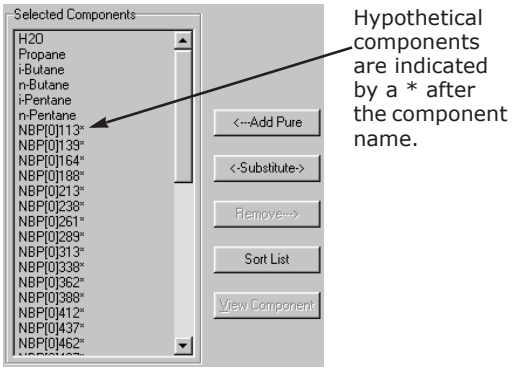
The characterization procedure is now complete.

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.

Leave Oil Environment icon

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

Figure 2.35



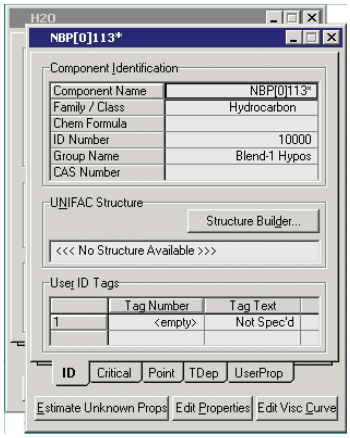
Viewing Component Properties

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.

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

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



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

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

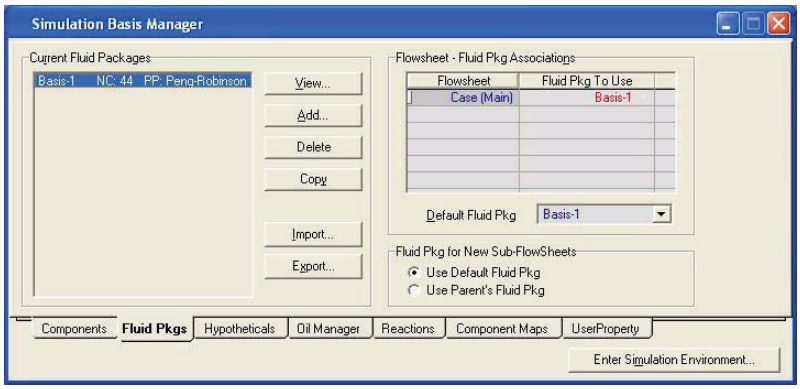
The petroleum hypocomponent shown here is an example of a hypothetical component. You can modify any of the parameters listed for this component. For this example, the properties of the hypothetical components generated during the oil characterization are not changed.

3. Close each of these two component property views.

Basis Manager icon

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.

Figure 2.37



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

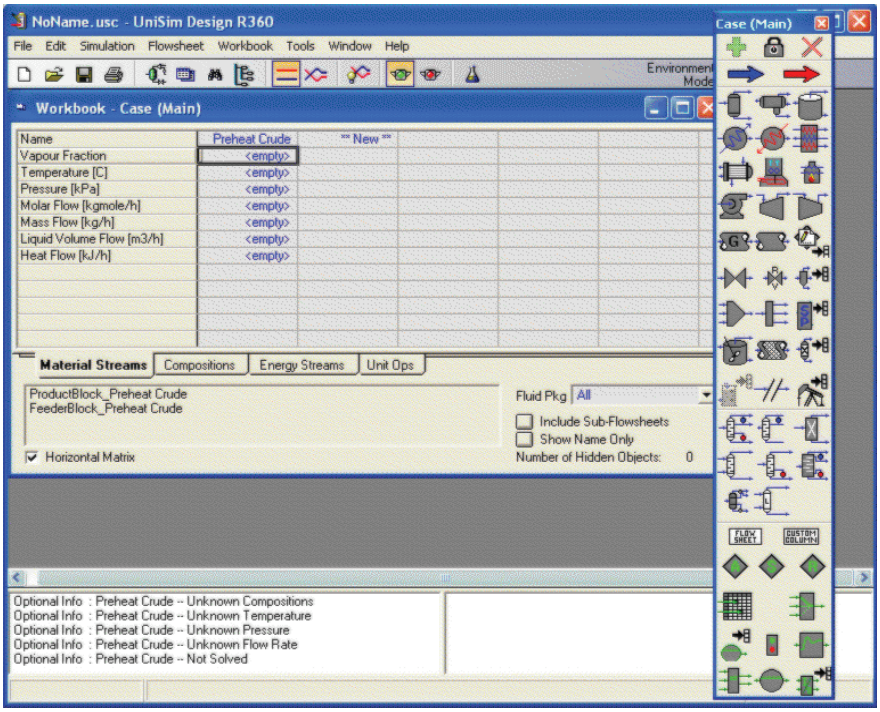
environment, do **one** of the following:

- 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**.

Figure 2.38



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.

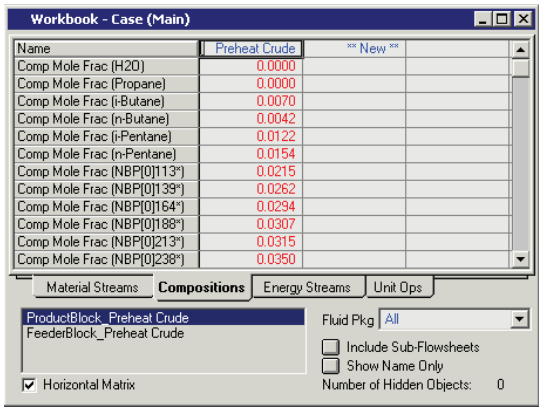
You can toggle the palette open or closed by pressing **F4**, or by selecting Open/Close Object Palette from the Flowsheet menu.

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.

Figure 2.39



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.

Save 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.

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

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

Workbook icon

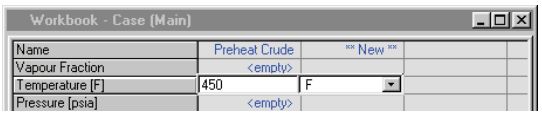
Click the Workbook icon on the toolbar to ensure the **Workbook view** is active.

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.

Figure 2.40



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.

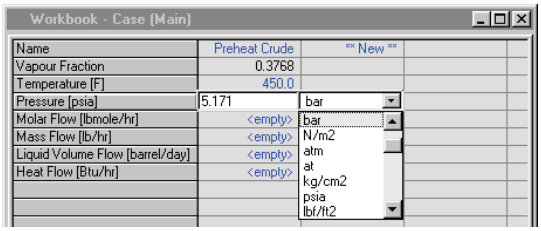
3. Since this is the correct unit, press the **ENTER** key. UniSim Design accepts the temperature. UniSim Design advances to the **Pressure** cell.

   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

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.

Figure 2.41



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

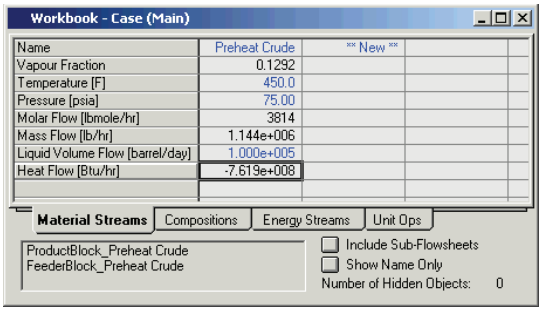
7. Once 'bar' is selected, press the **ENTER** key. UniSim Design accepts the pressure and automatically converts to the default unit, psia.
8. Click in the **Liquid Volume Flow** cell, then type 1e5. The stream flow is entered on a volumetric basis, in this case 100,000 barrel/day.
9. Press the **ENTER**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

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

Solver Active icon

Figure 2.42



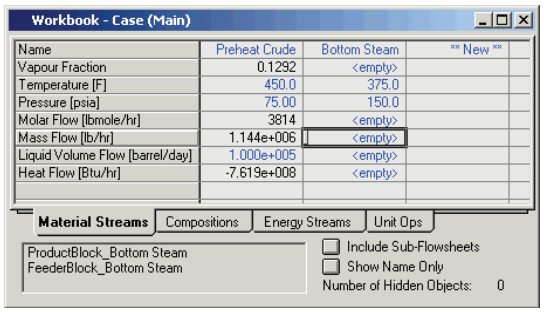
The next task is to install and define the utility steam streams that will be attached to the fractionation tower later.

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.

UniSim Design accepts blank spaces within a stream or operation name.

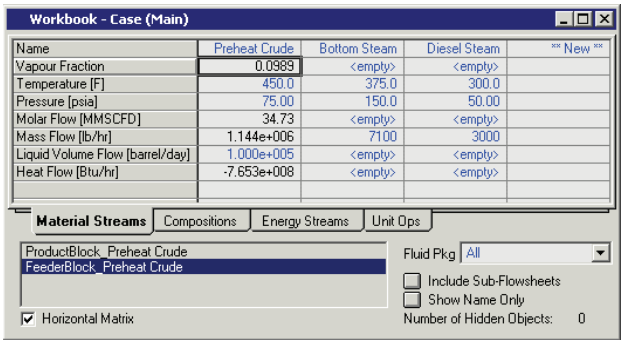
Figure 2.43



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.

Figure 2.44



Providing Compositional Input

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

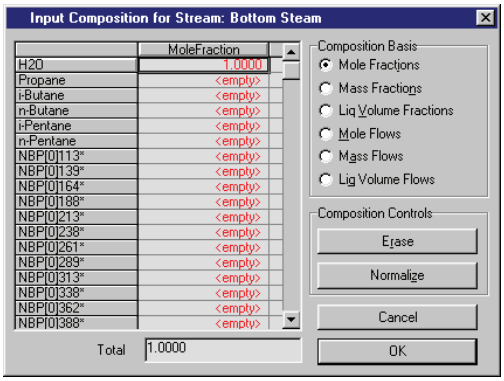
task is to input the compositions.

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.

Figure 2.45



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.
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: • Input the fractions of all components, including any zero components, such that their total is exactly 1.0000, then click the OK button. • 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. • Input the flows or relative number of parts of all components, including any zero components, then click the OK button.

These are the default colours; yours can appear differently depending on your settings on the Colours page of the Session Preferences view.

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

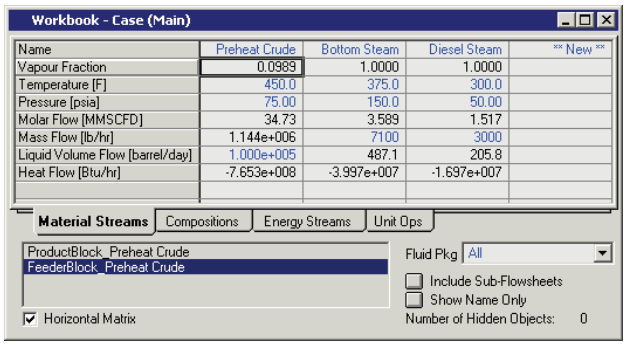
4. Click the **Normalize** button and all other component fractions are forced to zero.
5. Click the **OK** button. UniSim Design accepts the composition and you are returned to the Workbook view.

   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.

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.

Figure 2.46



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**.

Material Stream icon

Add Object icon

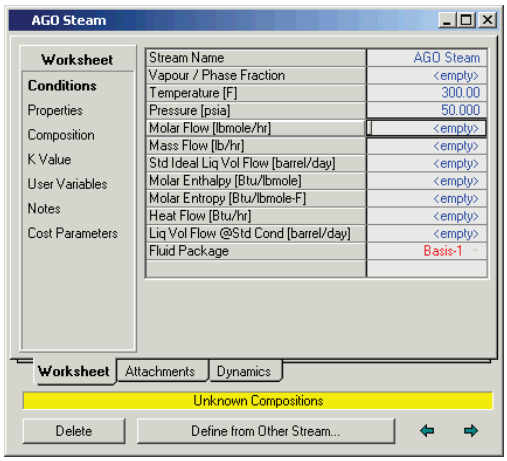
9. In the stream property view, click in the **Stream Name** cell and rename the stream **AGO Steam**.
10. Press **enter**.
11. In the **Temperature** cell, enter **300.**

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.

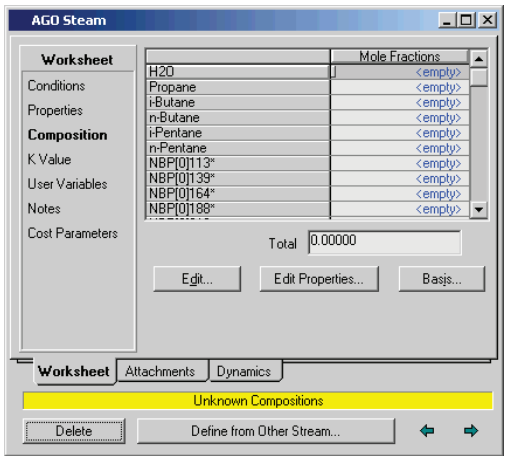
12. In the **Pressure** cell, enter **50**.

Figure 2.47



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

Figure 2.48



14. Click the **Edit** button. The Input Composition for Stream view appears.

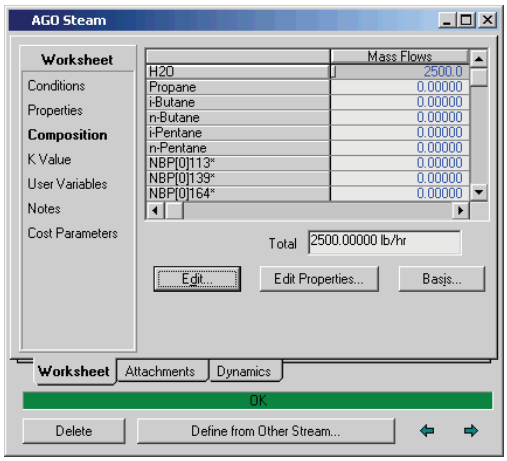
The current Composition Basis setting is set to the Preferences Default of Mole Fractions. The stream composition is entered on a mass basis.

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.

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.

Figure 2.49



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

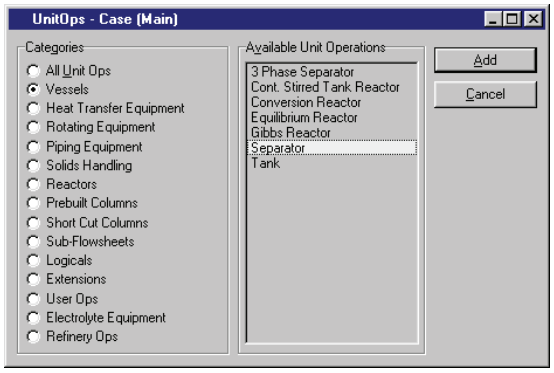
The first operation is a Separator, used to split the feed stream into its liquid and vapour phases. As with most commands in UniSim Design, installing an operation can be accomplished in a number of ways. One

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.

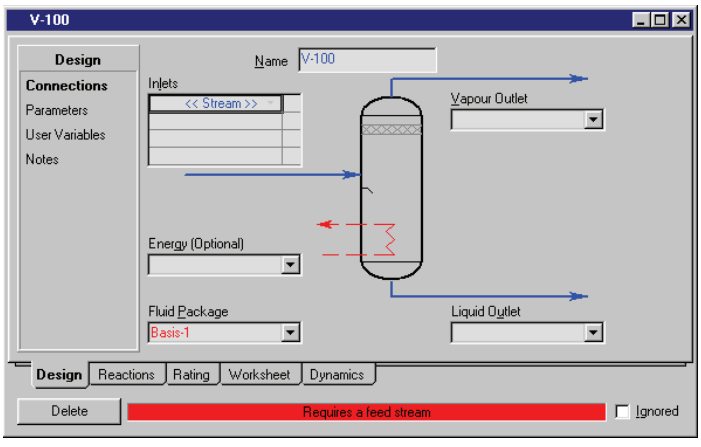
Figure 2.50



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.

Figure 2.51



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.

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.

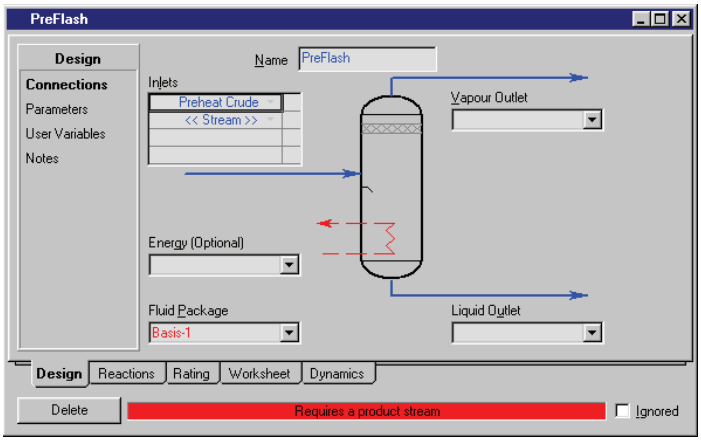
Many operations, like the separator, accept multiple feed streams. Whenever you see a matrix like the one in the **Inlets group**, the operation accepts multiple stream connections at that location. When the matrix is active, you can access a drop-down list of available streams.

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.

Alternatively, you could have made the connection by typing the exact stream name in the cell, and pressing **ENTER**.

9. 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**'.

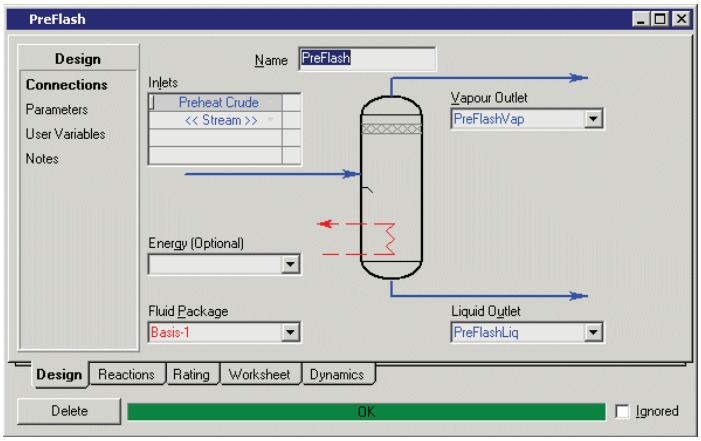
Figure 2.52



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.**

Figure 2.53

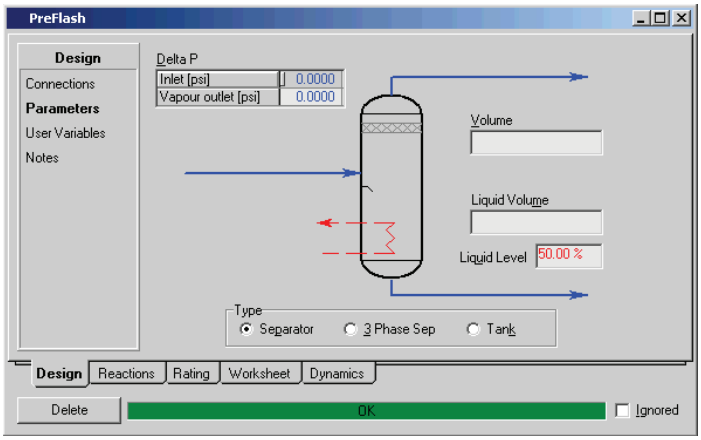


An **Energy** stream could be attached to heat or cool the vessel contents, however, for the purposes of this example, the energy stream is not required.

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.

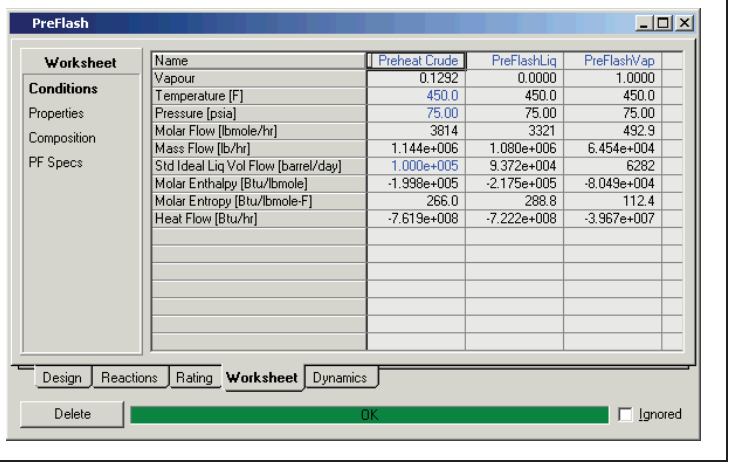
Figure 2.54



Since there is no energy stream attached to the **separator**, no **Optional Heat Transfer** information is required.

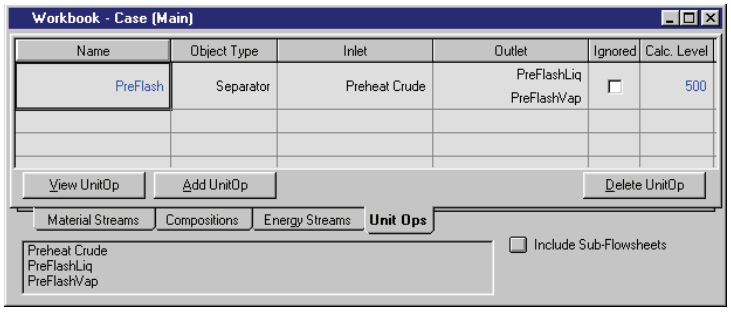
14. To view the calculated outlet streams, click the **Worksheet** tab. This is a condensed Workbook displaying only those streams attached to the operation.

Figure 2.55



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.

Figure 2.56



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

Return to the Material Streams tab of 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.

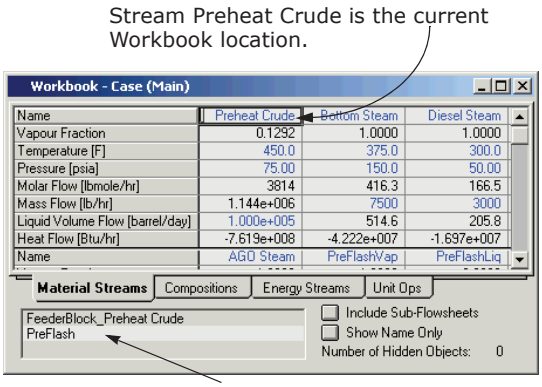
Any utilities attached to the stream with the Workbook active are also displayed in (and are accessible through) this display field.

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

property view appears.

Figure 2.57



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.

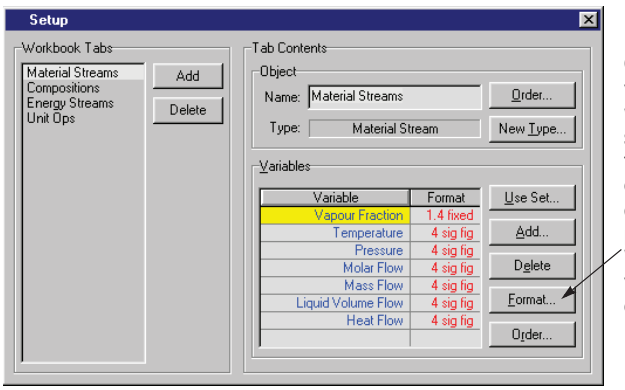
In this section, you will create a new Workbook tab that displays only stream pressure, temperature, and flow.

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.

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.

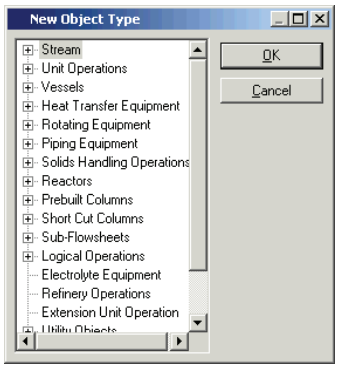
Figure 2.58



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.

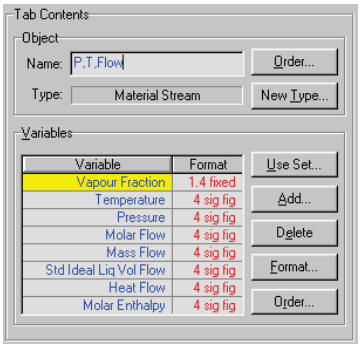
2. In the Workbook Tabs group list, select **Compositions**.
3. Click the **Add** button. The **New Object Type** view appears.

Figure 2.59



4. 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.

Figure 2.60



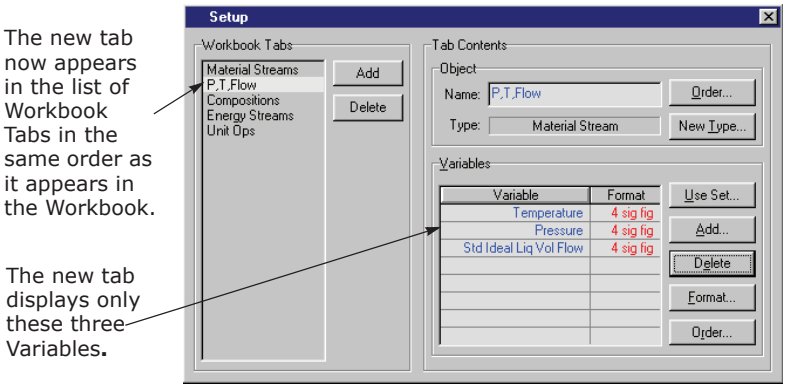
The next task is to customize the tab by removing the variables that are not required.

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.

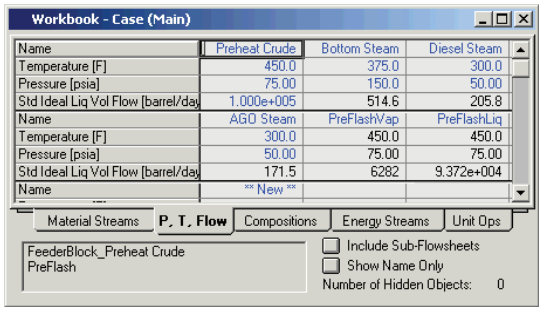
11. Click the **Delete** button to remove them from this **Workbook** tab. The finished **Setup** view appears below.

Figure 2.61



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

Figure 2.62



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**.

Save icon

2.2.8 Using the PFD

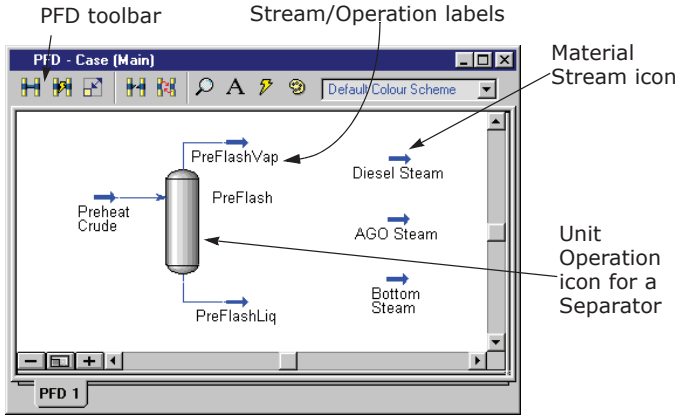
PFD icon

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.

Figure 2.63



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.

Size Mode icon

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

Zoom Out 25% icon

Display Entire PFD icon

Zoom In 25% icon

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.

- 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:

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

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.

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

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.

Heater icon (Red)

Cooler icon (Blue)

Installing the Crude Furnace

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

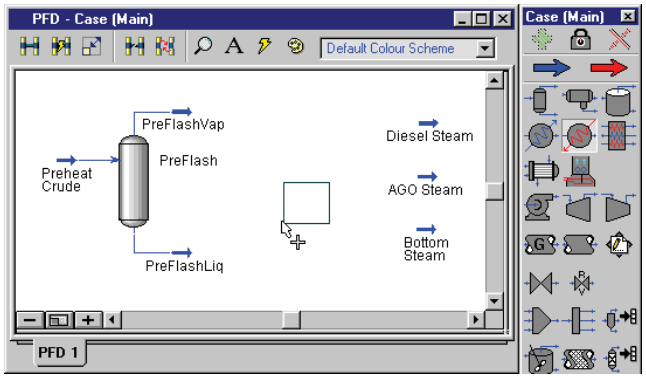
1. Ensure the **Object Palette** is visible (if it is not, press **F4**).

   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

horizontal scroll bar.

2. 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**.

Figure 2.64

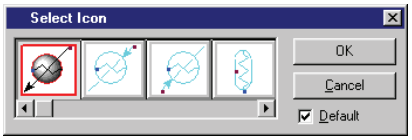


Notice the **heater** has red status (colour), indicating that it requires feed and product streams.

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.

Figure 2.65



Furnace icon

Attach Mode icon

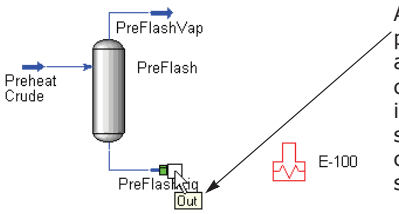
7. Click the **WireFrameHeater5** icon (scroll to the right), then click the **OK** button. The new icon appears in the **PFD**.

Attaching Streams to the Furnace

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.

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.

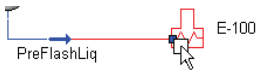
Figure 2.66



At the square connection point, a pop-up description appears attached to the cursor. The pop-up "Out" indicates which part of the stream is available for connection, in this case, the stream outlet.

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.

Figure 2.67



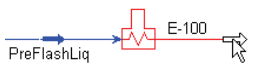
Break Connection icon

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.

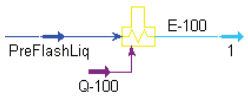
6. Release the mouse button, and the connection is made to the **heater** inlet.
7. Position the cursor over the right end of the **heater** icon. The connection point and pop-up 'Product' appears.
8. With the pop-up visible, click and hold the mouse button. The white box again becomes black.
9. Move 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



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.

Figure 2.69



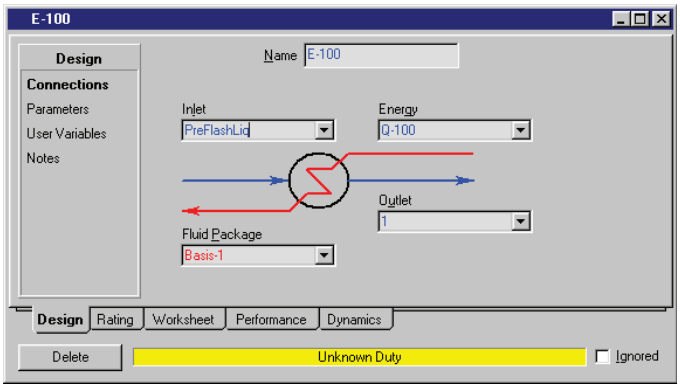
12. Click the **Attach** icon again to return to **Move** mode.

    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.
2. Click the **Design** tab, then select the **Connections** page. The names of the Inlet, **Outlet,** and **Energy** streams appear in the appropriate fields.

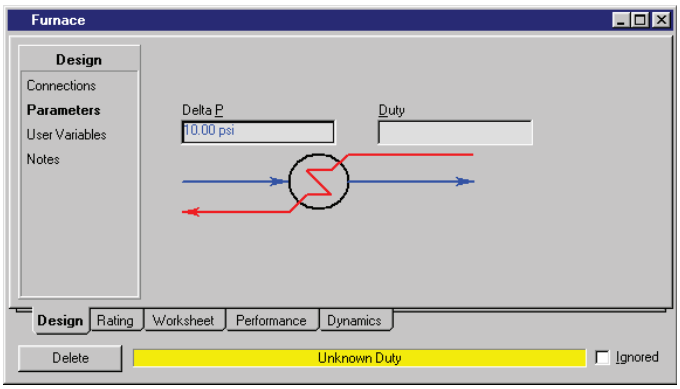
Figure 2.70



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.

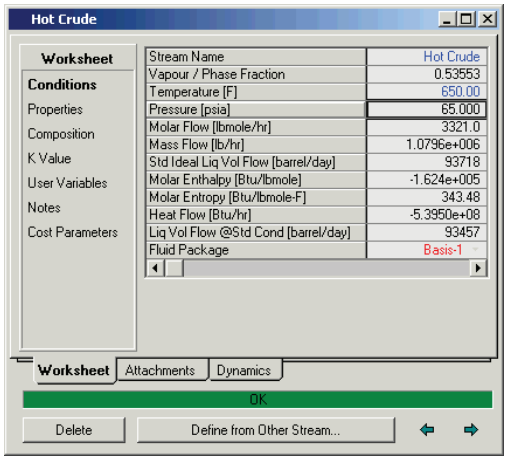
Figure 2.71



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 n**ame** to **Hot Crude**.
8. In the **Temperature** field, specify a temperature of **650**°F.

Figure 2.72

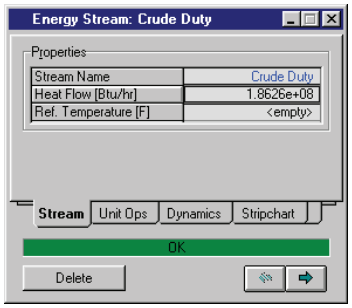


The remaining degree of freedom in the Furnace has now been used, so UniSim Design can flash **Hot Crude** and determine its remaining properties.

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



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.

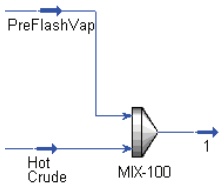
1. Make some empty space available to the right of the **Furnace** using the horizontal scroll bar. Move other objects if necessary.

Mixer icon

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.
7. 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.

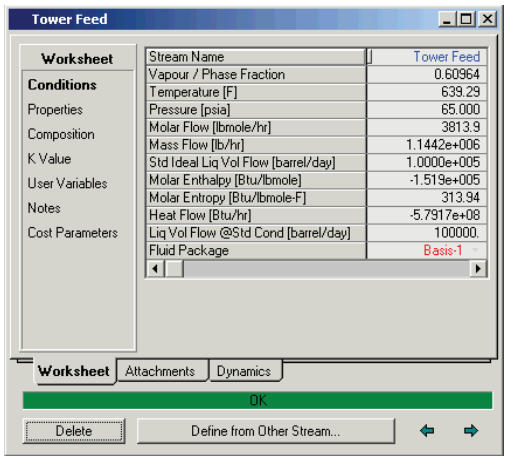
Multiple connection points appear because the **Mixer** accepts multiple feed streams.

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.



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.

Figure 2.74



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

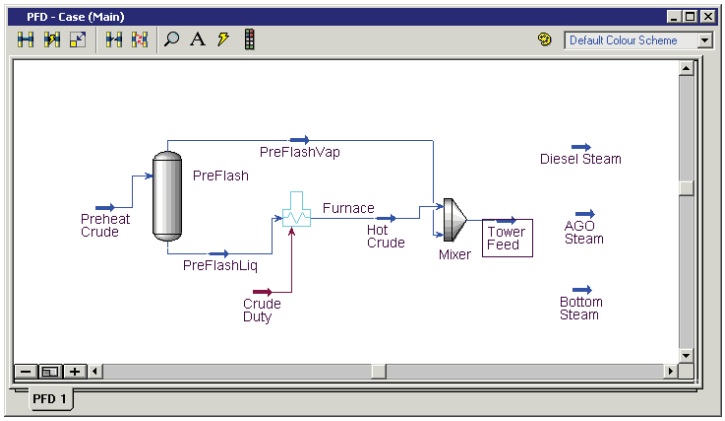
Resizing Icons in the PFD

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

Zoom All icon

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.

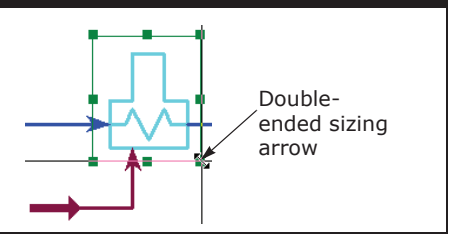
Figure 2.75



Size Mode icon

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.

Figure 2.76



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.

Energy Stream icon

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

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.

Refluxed Absorber icon

Save icon

Installing the Column

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 you are comfortable working with the side equipment, try using the templates. Instructions on using the crude column template are given in an annotation on the next page.

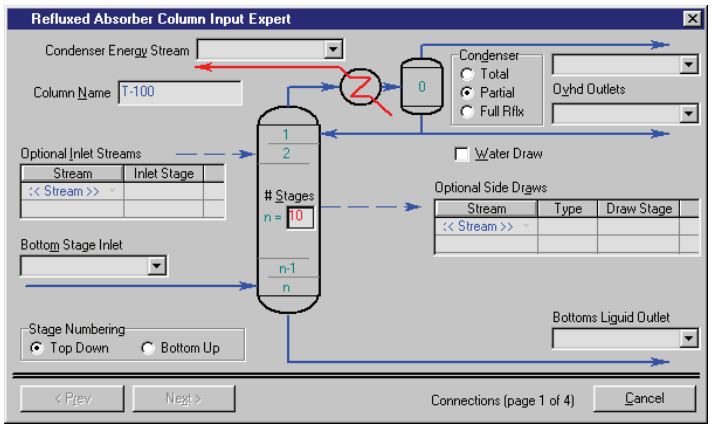
UniSim Design has a number of pre-built column templates that you can install and customize by changing attached stream names, number of stages and default specifications, and adding side equipment. One of these templates is going to be used for this example (a crude column with three side strippers), 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.

To install this column using the pre-built crude column templates:

1. Double-click on the **Custom Column** icon on the Object Palette.
2. On the view that 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.

Figure 2.77



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

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:

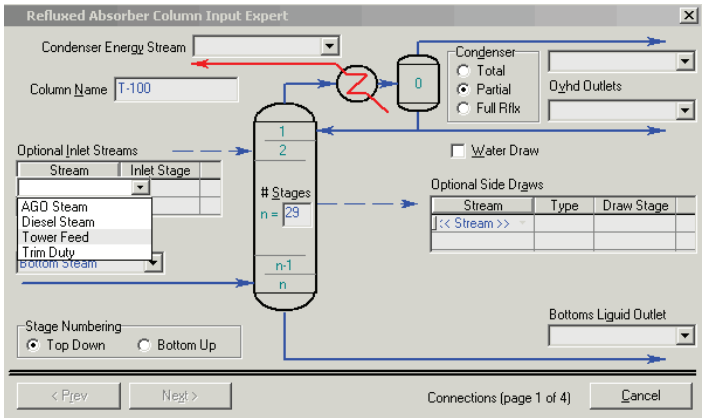
- These are theoretical stages, as the UniSim Design default stage efficiency is one.
- If present, the Condenser and Reboiler are considered separate from the other stages, and are not included in the # Stages field.

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**.
3. Click the down arrow to open the drop-down list of available feeds.

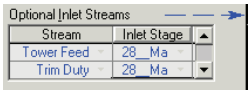
Figure 2.78



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 drop-down 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.

Figure 2.79

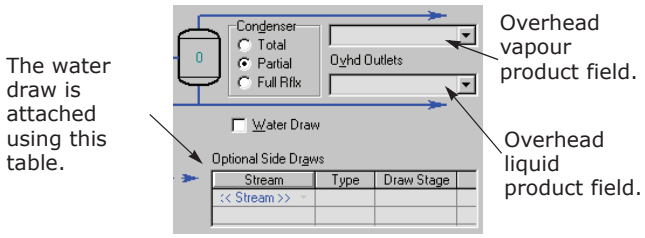


9. Advance to the **Bottom Stage Inlet** field by clicking on it or by pressing **tab**.
10. In the **Bottom Stage Inlet** field, click the down arrow to open the drop-down list of available feeds.
11. From the list, select **Bottom Steam** as the bottom feed for the column.

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.

Figure 2.80



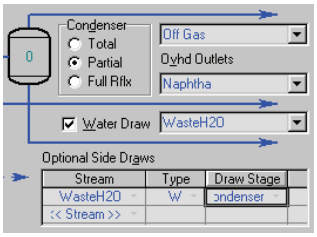
Although the overhead vapour product has zero flow, do not change the condenser to **Total**. At this time, only the **Partial** radio button allows you to specify a three-phase condenser.

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.
3. 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.
5. 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

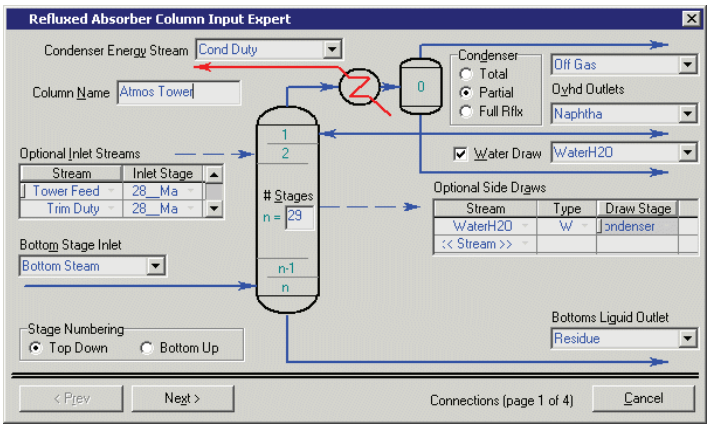


10. In the **Column Name** field, enter **Atmos Tower**.
11. In the **Bottoms Liquid Outlet** field, type **Residue** to create a new stream.
12. In the **Condenser Energy Stream** field, type **Cond Duty** to define a new stream. Press **ENTER**.

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



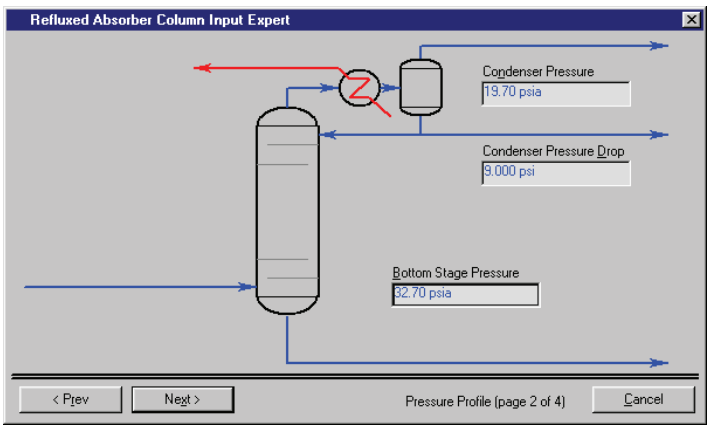
The Next button now becomes available, indicating sufficient information has been supplied to advance to the next page of the **Input Expert**.

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

Entering the Initial Estimate Values

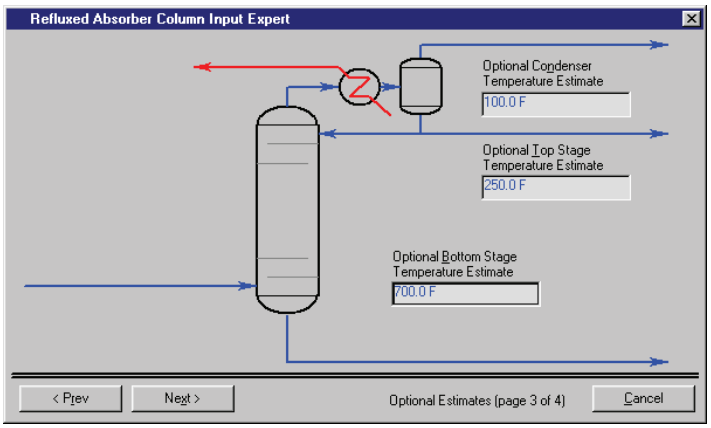
1. On the **Pressure Profile** page, specify the following:
   - Condenser Pressure 19.7 psia
   - Condenser Pressure Drop 9 psi
   - Bottom Stage Pressure 32.7 psia

Figure 2.83



2. 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

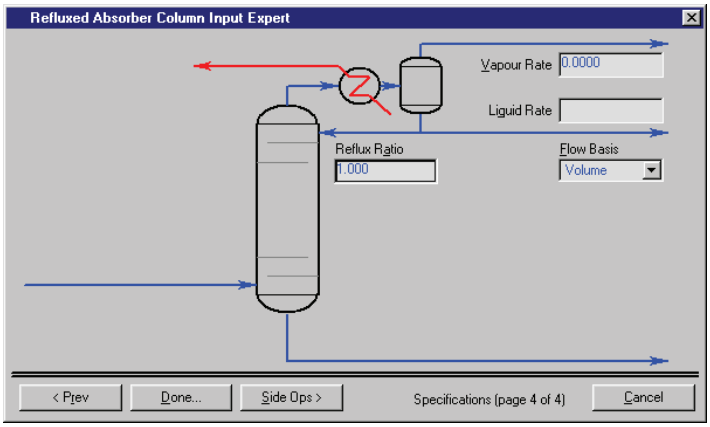


4. Click the **Next** button to advance to the fourth and final page of the **Input Expert**. This page allows you to supply values for the default column specifications that UniSim Design has created.

   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**

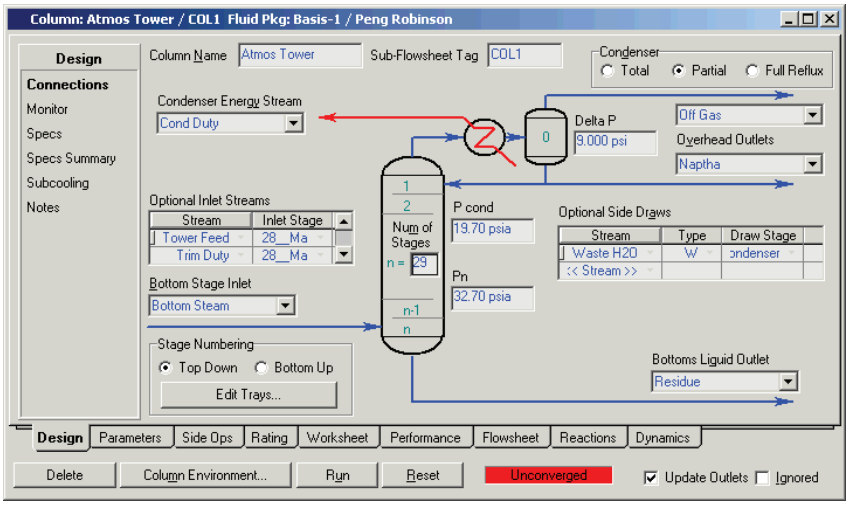
- **Reflux Ratio 1.0**.

Figure 2.85



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

Figure 2.86



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.

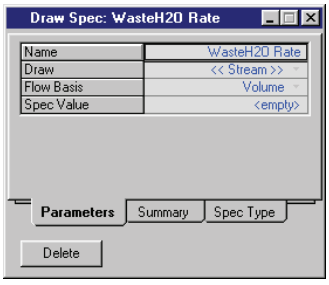
The current **Degrees of Freedom** is one, indicating that only two specifications are **active**. As noted earlier, a Refluxed Absorber with a partial condenser has two degrees of freedom and, therefore, requires two **active** specifications. In this case, however, a third degree of freedom was created 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.
6. Select Column Draw Rate and click the **Add Spec(s)** button. The Draw Spec property view appears.

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.

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



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**.

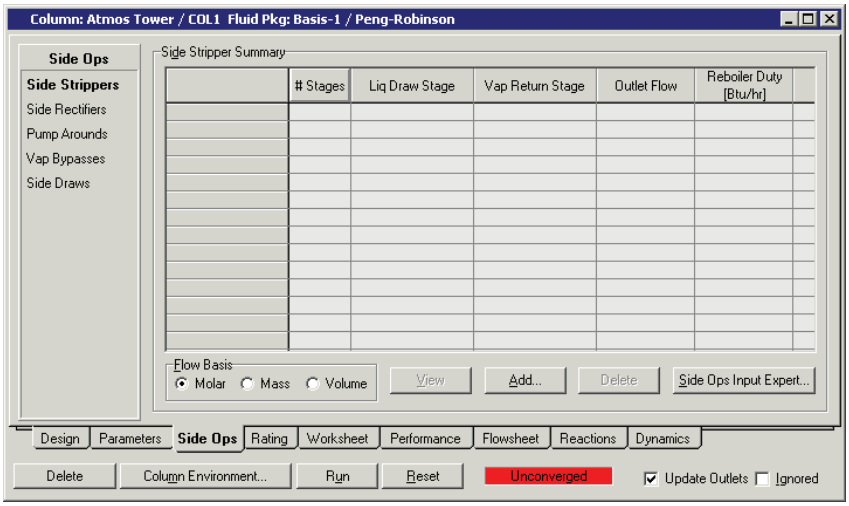
   The Connections page is similar to the first page of the **Input Expert**. Currently, the column is a standard type, so this page shows a column schematic with the names of the attached streams. When the side equipment is added to the column, the page becomes non-standard. There are a large number of possible non-standard 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.

Figure 2.88

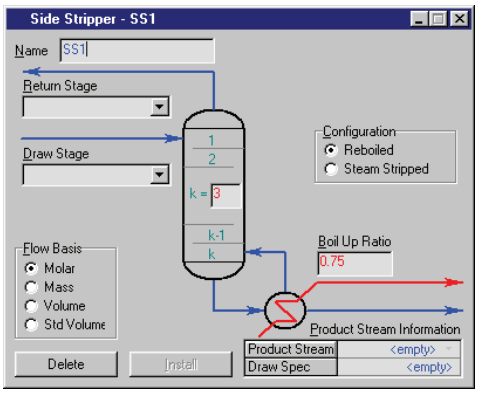


On this tab, you can Install, View, Edit, or Delete all types of Side Equipment. The table displays summary information for a given type of side operation, depending on the page you are currently on.

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

When you install side equipment, it resides in the Column sub-flowsheet. 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.

Figure 2.89

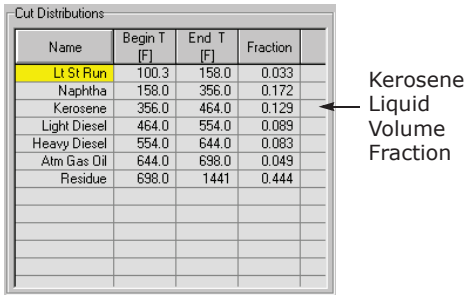


This is a **reboiled** 3-stage stripper with a 0.75 boil up ratio, so leave the **Configuration** radio button at Reboiled, and the **k** = and **Boil Up Ratio** fields at their defaults.

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 fi**eld, enter **Kerosene**.

   The straight run product distribution data calculated during the Oil Characterization appears in the figure below.

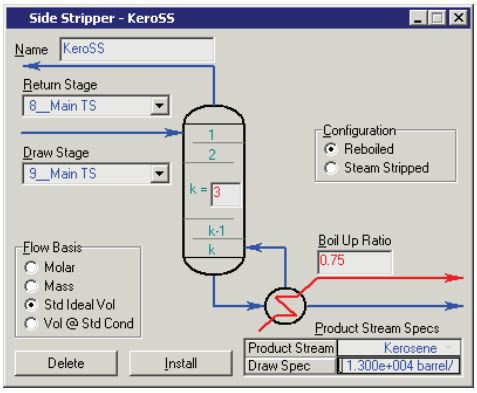
Figure 2.90



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

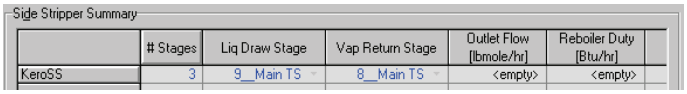


10. Click the **Install** button, and a view summarizing your input appears.


Close icon

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.

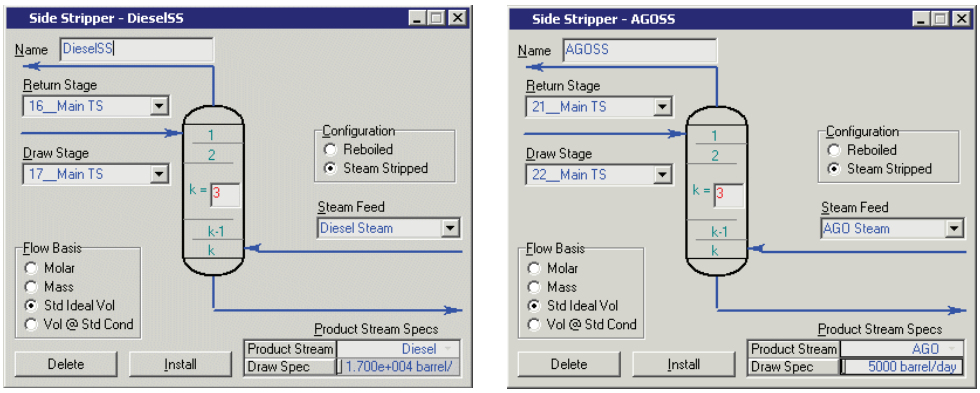
Figure 2.92



12. Use the previous steps to install the two remaining side strippers **DieselSS** 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 **DieselSS and AGOSS side stripper** views appear in the following figure.

Figure 2.93



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 sub-flowsheet, are unknown at this point. The conditions of the Main Flowsheet streams are duplicated into these sub-flowsheet streams when the stream attachments are performed.

The completed **Side Stripper Summary** table appears below.

Figure 2.94

Side Stripper Summary

	# Stages	Liq Draw Stage	Vap Return Stage	Outlet Flow [lbmole/hr]	Reboiler Duty [Btu/hr]
KeroSS	3	9_Main TS	8_Main TS	<empty>	<empty>
DieselSS	3	17_Main TS	16_Main TS	<empty>	<empty>
AGOSS	3	22_Main TS	21_Main TS	<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.

Figure 2.95

	Specified Value	Current Value	Wt. Error	Active	Estimate	Current
Reflux Ratio	1.000	<empty>	<empty>	☑	☑	☑
Distillate Rate	<empty>	<empty>	<empty>	☑	☑	☑
Vap Prod Rate	0.0000 barrel/day	<empty>	<empty>	☐	☑	☐
WasteH2O	<empty>	<empty>	<empty>	☑	☑	☑
KeroSS Prod Flow	1.300e+004 barrel/day	<empty>	<empty>	☑	☑	☑
KeroSS BoilUp Ratio	0.7500	<empty>	<empty>	☑	☑	☑
DieselSS Prod Flow	1.700e+004 barrel/day	<empty>	<empty>	☑	☑	☑
AGOSS Prod Flow	5000 barrel/day	<empty>	<empty>	☑	☑	☑

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.

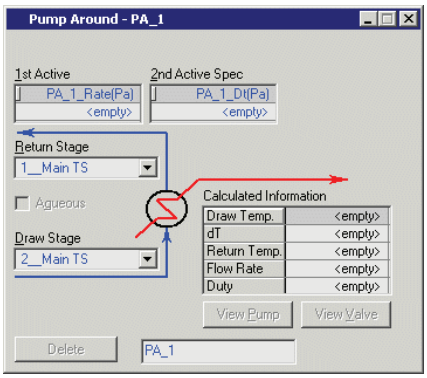
The installation of the side strippers created four additional degrees of freedom, so UniSim Design created a **Prod Flow** (product flow) specification for each side stripper, plus a **BoilUp Ratio** specification for the Kerosene side stripper. The new specifications were automatically made **Active**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

Installing the Pump Arounds

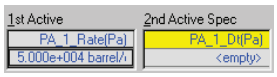
1. Click the **Side Ops** tab and select the **Pump Arounds** page.
2. Click the **Add** button. The initial **Pump Around** view appears.
3. In the **Return Stage** drop-down list, select stage 1 (**1_Main TS**).
4. In the **Draw Stage** drop-down list, select stage 2 (**2_Main TS**).

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

Figure 2.96



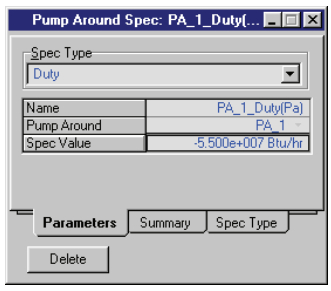
Each cooled pump around circuit has two specifications associated with it. The default **Pump Around Specifications** are circulation rate and temperature drop (**Dt**) between the liquid draw and liquid return. For this example, the **Dt** specification is changed to a Duty specification for the pump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



6. In the empty cell under the **PA_1_Rate(Pa)** specification, enter **5e4**.
7. 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**.

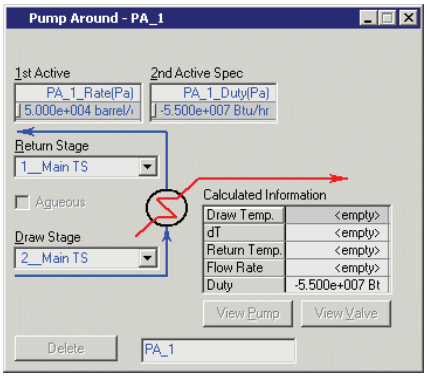
Notice the negative sign convention indicates cooling.

Figure 2.97



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

Figure 2.98



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.

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 drop-down list.
7. Enter the Spec Value.
8. Close the view.

Figure 2.99

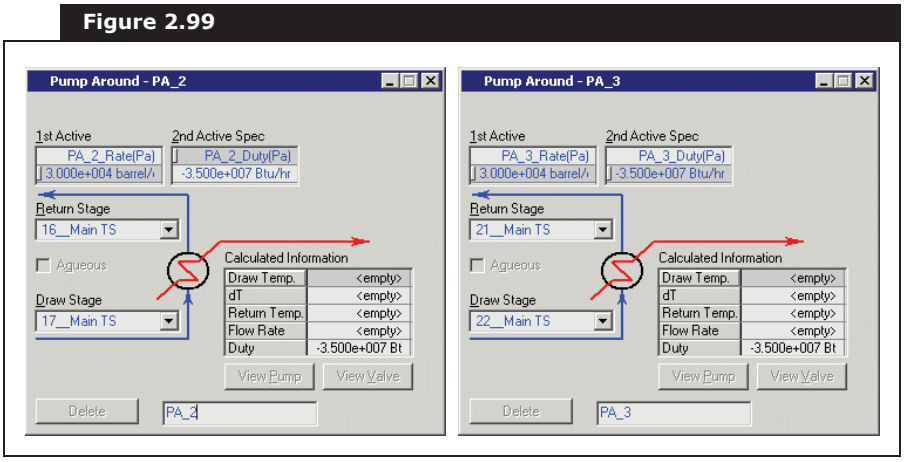
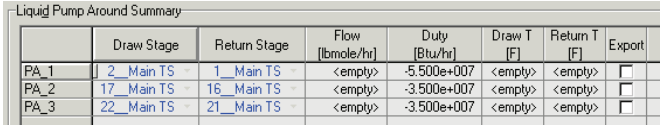
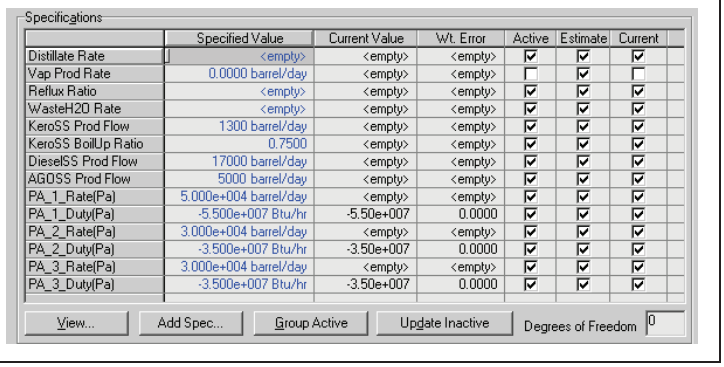


Figure 2.100



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

Figure 2.101

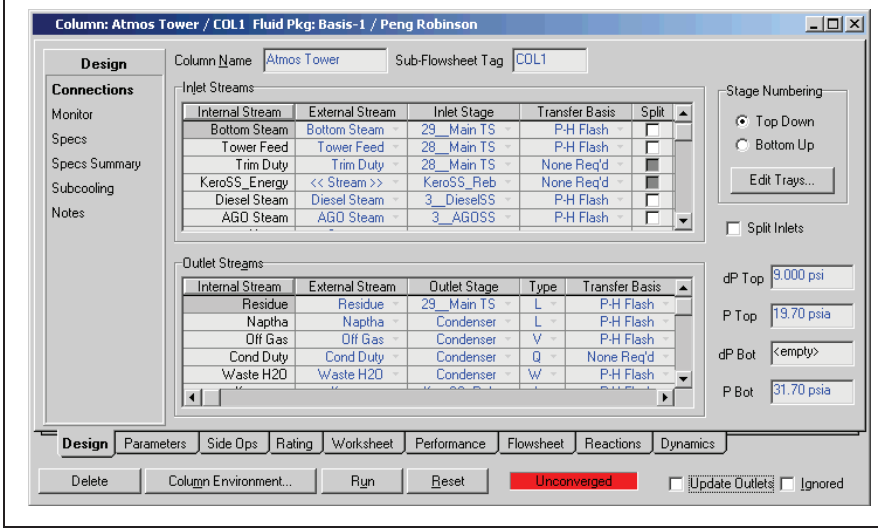


The addition of the pump arounds has created six more degrees of freedom, resulting in a total of 13 available degrees of freedom. Currently, 13 Specifications are active, so the overall Degrees of Freedom is zero.

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.

Figure 2.102



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

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

The column has 40 **Total Theoretical** Stages:

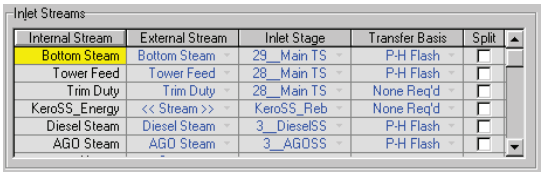
- 29 in the main tray section
- 1 condenser for the main column
- 9 in the side strippers (3 side strippers with 3 stages each)
- 1 reboiler for the Kerosene side stripper

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 sub-flowsheet 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**.

Figure 2.103



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.

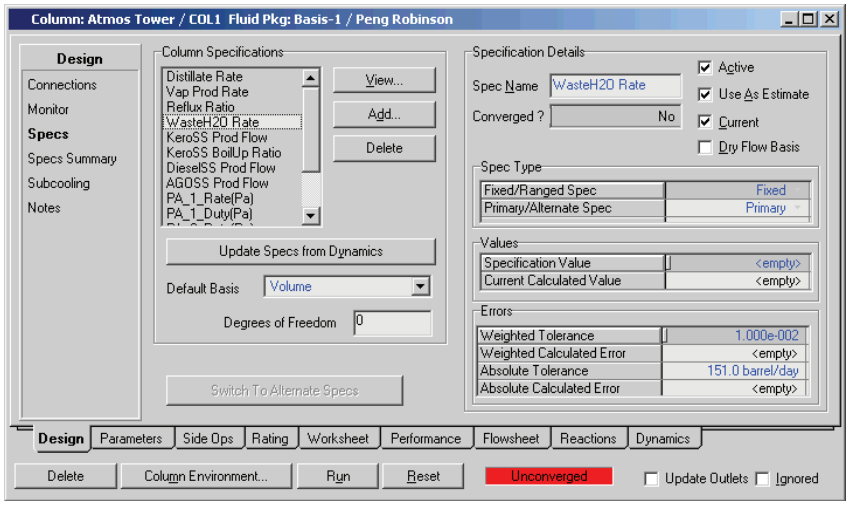
The current **Degrees of Freedom** is zero, indicating the column is ready to be solved. Before you run the column, however, you will have to replace two of the active specifications, **Waste H2O Rate** and **KeroSS BoilUp Ratio**, with the following new ones:

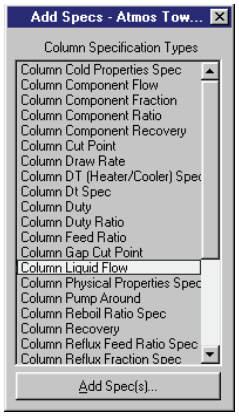
- Overflash specification for the feed stage (Tray Net Liquid Flow specification)
- Kerosene side stripper reboiler duty specification

Adding the Overflash Specification

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

Figure 2.104



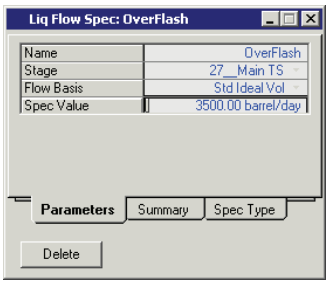


2. In the **Column Specifications** group, click the **Add** button. The **Add Specs** view appears.
3. Select **Column Liquid Flow** as the **Column Specification Type**.
4. Click the **Add Spec(s)** button, and the **Liq Flow Spec** view appears.
5. Change the **name** from its default to **Overflash**.
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.

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

Figure 2.105

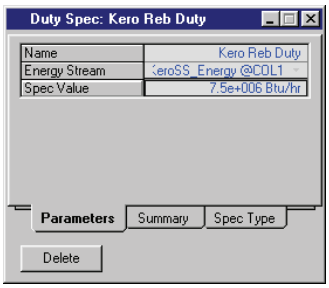


8. Close the view to return to the Column property view. The new specification appears in the list of **Column Specifications** group on the **Specs** page.

Adding the Duty Specification

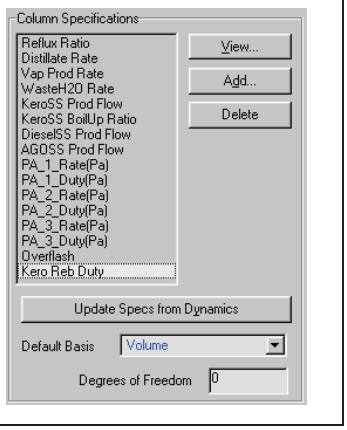
1. Click the **Add** button again to add the second new specification.
2. Select **Column Duty** as the **Column Specification** Type, then click the **Add Spec(s)** button. The **Duty Spec** view appears.
3. In the **Name** cell, change the **name** to **Kero Reb Duty**.
4. In the **Energy Stream**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

Figure 2.106



6. 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



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:

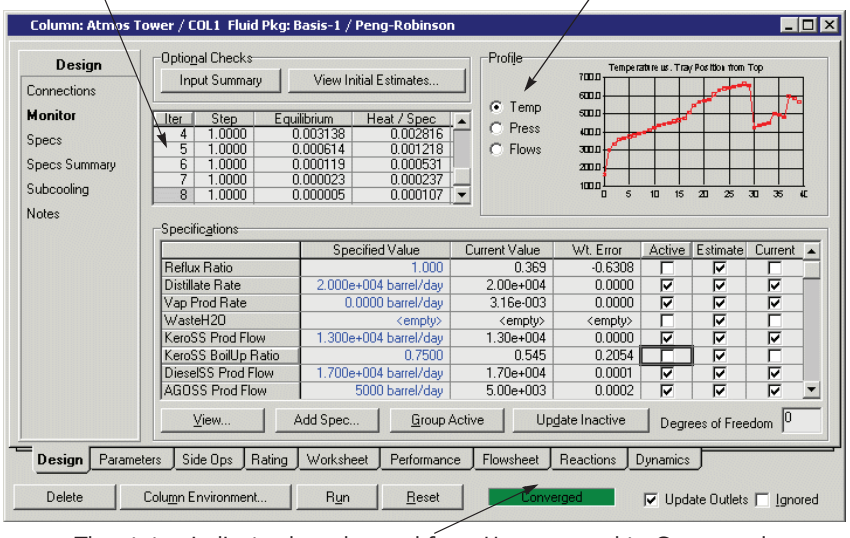
2. In the **Specified Value** cell for the **Distillate Rate** specification, enter **2e4** (barrel/day).
3. Activate the **Overflash** specification by clicking its **Active** checkbox.
4. Activate the **Kero Reb Duty** specification.
5. Activate the **Vap Prod Rate** specification.
6. Deactivate the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

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.

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

shown in the figure below.

Figure 2.108

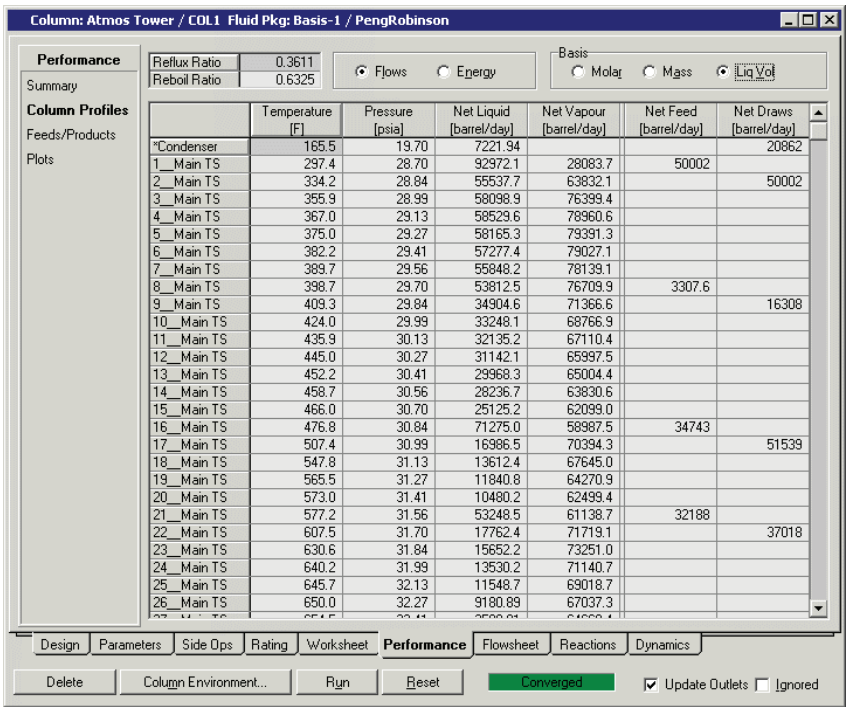


The converged temperature profile is currently displayed in the upper right corner of the view. To view the pressure or flow profiles, select the appropriate radio button.

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

The **Column Profiles** page appears below.

Figure 2.109



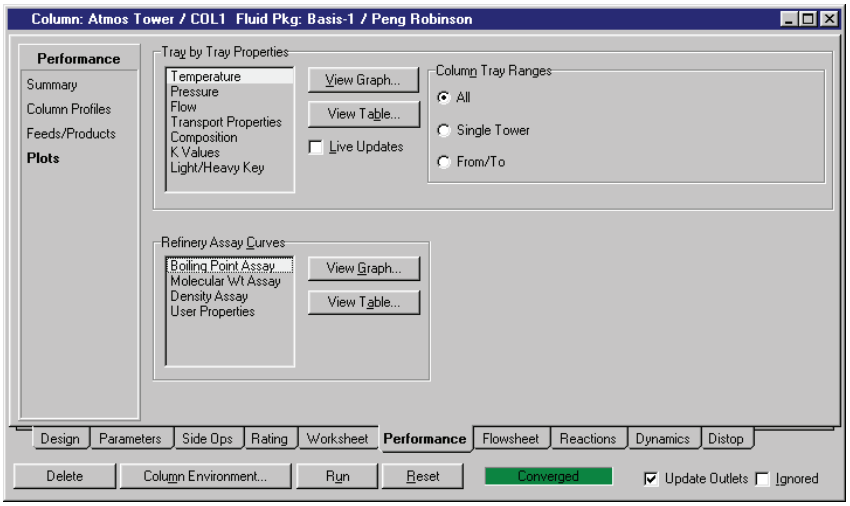
In the Basis group near the top of the view, select the **Liq Vol** radio button to examine the tray vapour and liquid flows on a volumetric basis.

Viewing Boiling Point Profiles for the Product Stream

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

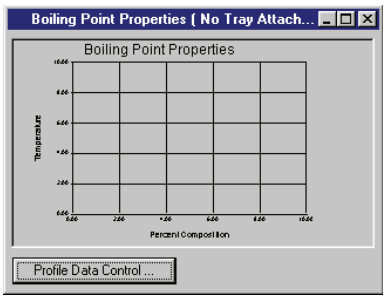
1. On the **Performance** tab, click on the **Plots** page.

Figure 2.110



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

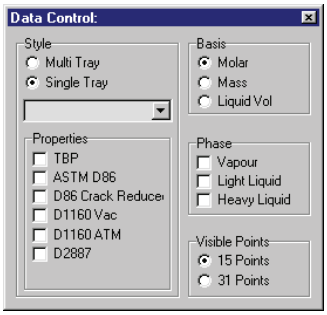
Figure 2.111



No data is plotted on the graph, since there is currently **No Tray Attached**, as shown in the title bar.

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

Figure 2.112



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.

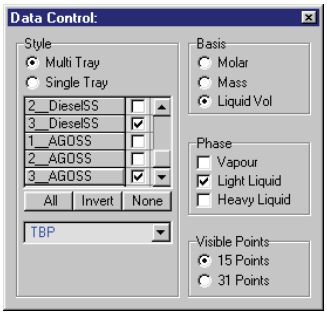
5. Select the **Multi Tray** radio button in the **Style** group. The **Data Control** view is modified, showing a matrix of column stages with a checkbox for each stage.
6. Activate the following stages by clicking on the corresponding checkboxes:
   - Condenser (Naphtha product stage)
   - 29_Main 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.
10. 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 (x-axis) variable is the Assay Volume Percent, while the dependent (y-axis) variable is the TBP in °C.

The completed **Data Control** view is shown below.

Figure 2.113

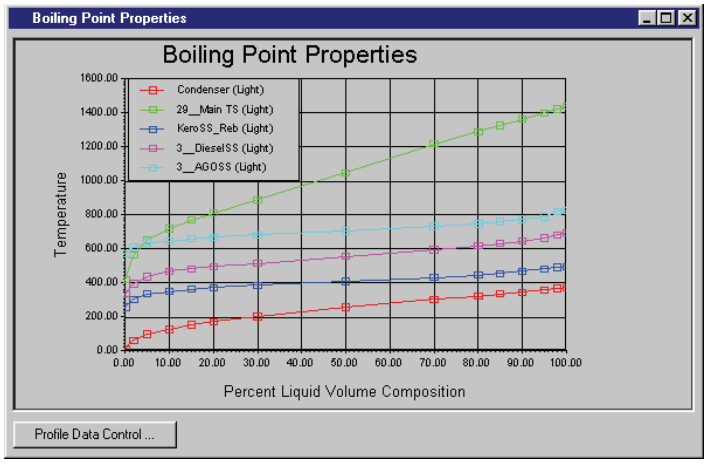


11. Click on the **Close** icon to close the Data Control view. You return to the Boiling Point Properties view, which now displays the TBP curves.
12. Make the **Boiling Point Properties** view more readable by clicking the **Maximize** icon in the upper right corner of the view, or by clicking and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

    The Boiling Point Properties view is shown below.

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

Figure 2.114



PFD icon

Workbook icon

Column Runner icon

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

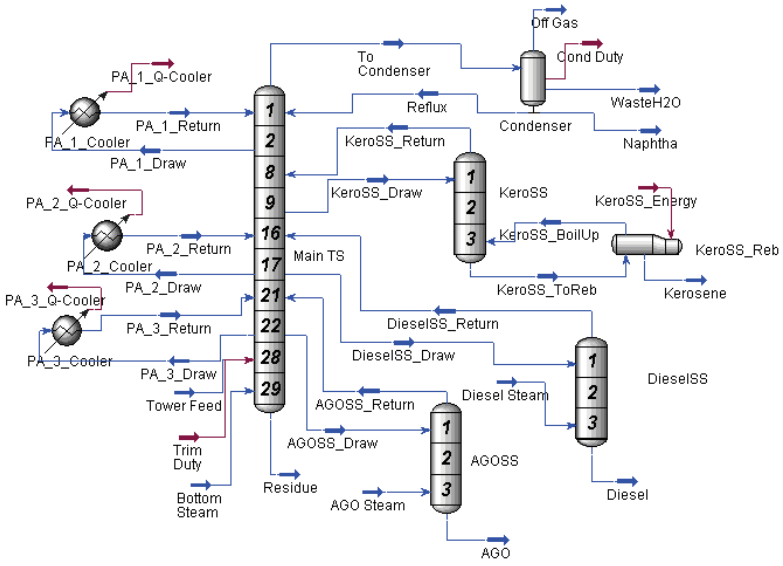
Moving to the Column Sub-Flowsheet

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

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.

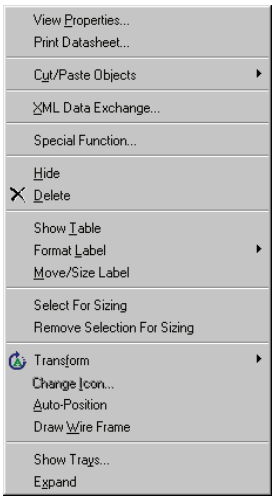
   The Column sub-flowsheet PFD is shown below.

Figure 2.115



Maximize icon

Zoom All icon



Object Inspect menu

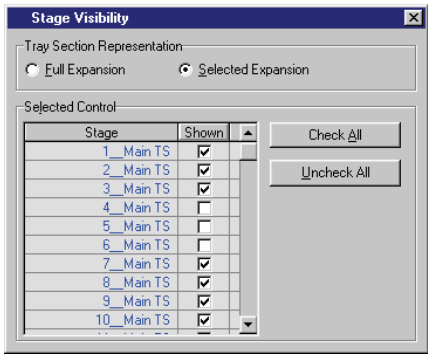
Customizing the Column PFD

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.

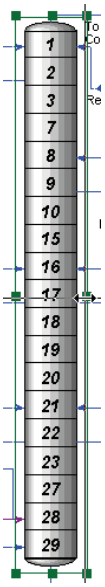
Figure 2.116



9. 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.

Size icon

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:

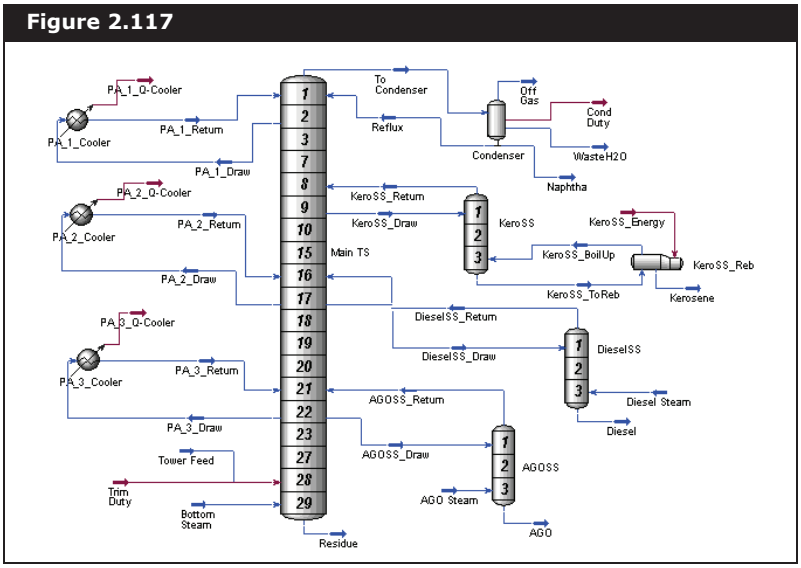
7. Click on the label you want to move.
8. Right-click and select **Move/Size Label**.
9. Move the label to its new position by clicking and dragging it, or by pressing the arrow keys.

   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 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**.

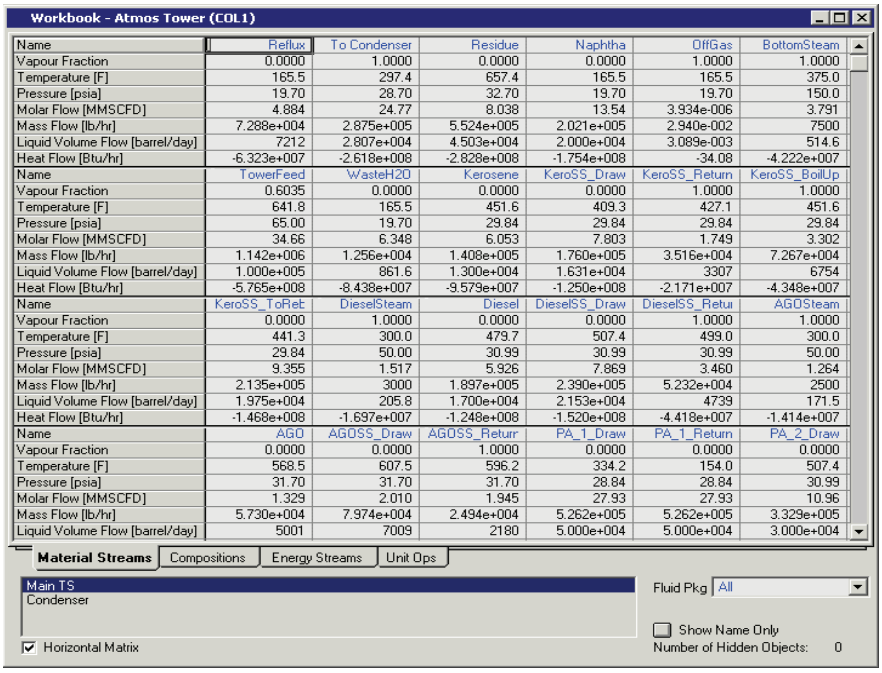
    The customized **PFD** appears below.

Figure 2.117



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

Figure 2.118



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	WasteH2O	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_ToRet	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_Returr	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

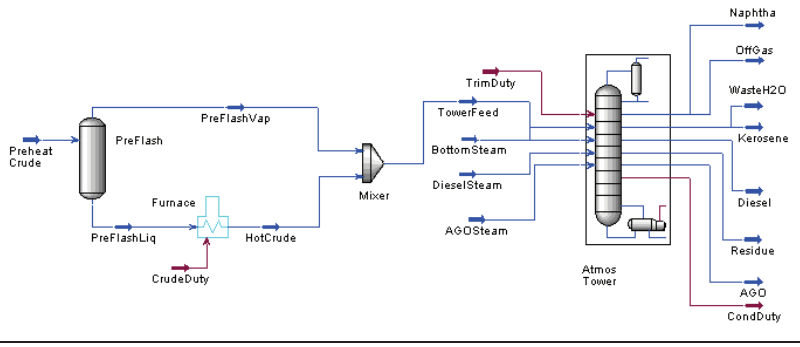
13. When you are finished working in the Column environment, return to the Main Flowsheet by clicking the **Enter Parent Simulation Environment** icon.

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.

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.

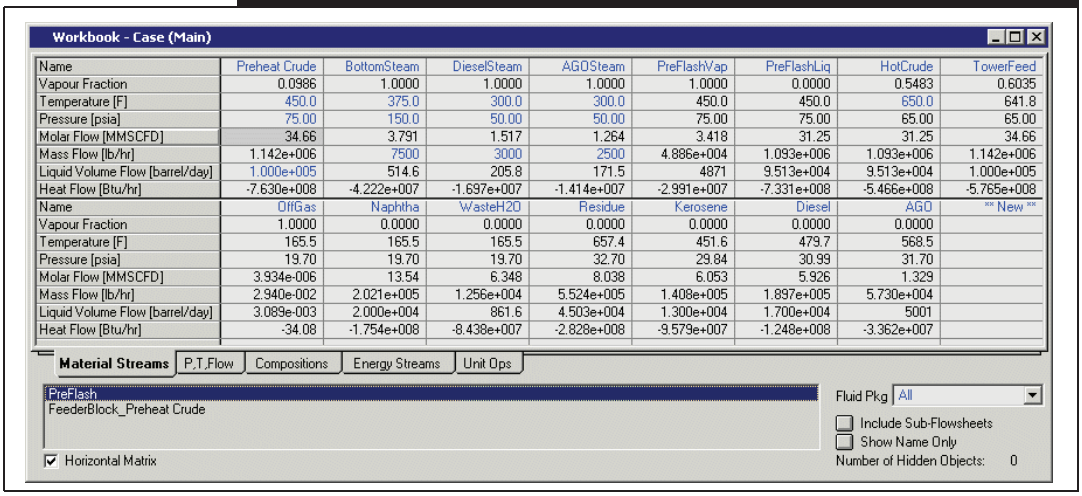
Figure 2.119



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.

Figure 2.120



Name	Preheat Crude	BottomSteam	DieselSteam	AGOSteam	PreFlashVap	PreFlashLiq	HotCrude	TowerFeed
Vapour Fraction	0.0986	1.0000	1.0000	1.0000	1.0000	0.0000	0.5483	0.6035
Temperature [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
Molar 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
Liquid 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	WasteH2O	Residue	Kerosene	Diesel	AGO	** New **
Vapour 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	
Liquid 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	

Using the Object Navigator

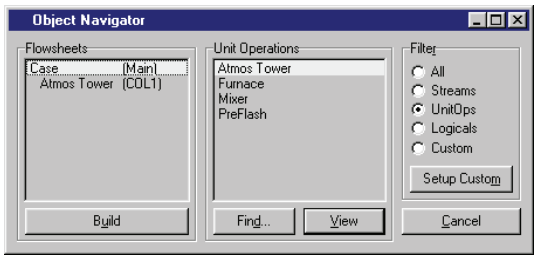
Now that results have been obtained, you can view the calculated properties of a particular stream or operation. The **Object Navigator** allows you to quickly access the property view for any stream or unit operation at any time during the simulation.

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

The **Object Navigator** view appears:

Figure 2.121



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 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.

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.

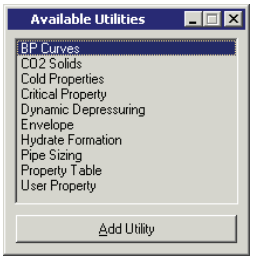
2.2.10 Installing a Boiling Point Curves Utility

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.

6. Click the **Create** button. The Available Utilities view appears, presenting you with a list of UniSim Design utilities.

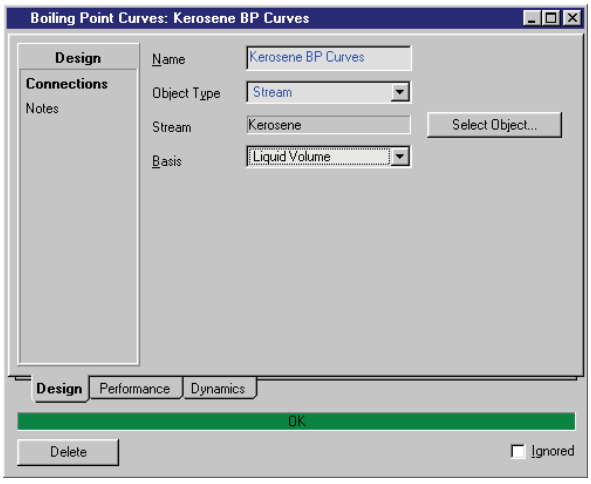
Figure 2.122



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.
9. 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.

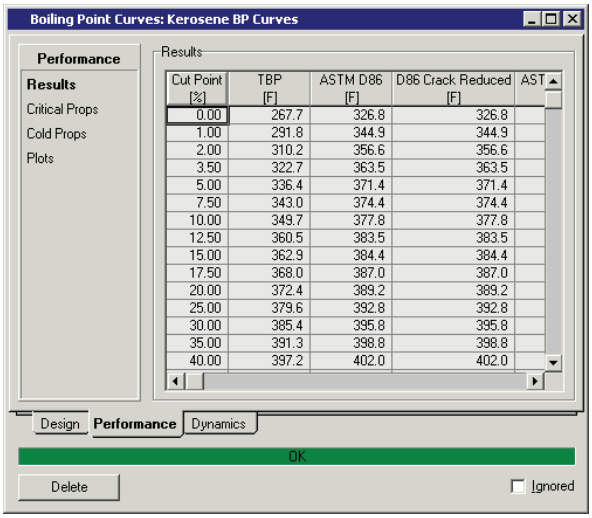
A Utility is a separate entity from the stream it is attached to; if you delete it, the stream is not affected. Likewise, if you delete the stream, the Utility remains but cannot display any information until you attach another stream using the Select Object button.

Figure 2.123



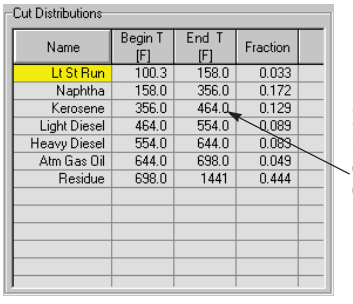
11. You can scroll through the matrix of data to see that the **TBP** ranges from 267°F to 502°F by going to the **Performance** tab and selecting the **Results** page.

Figure 2.124



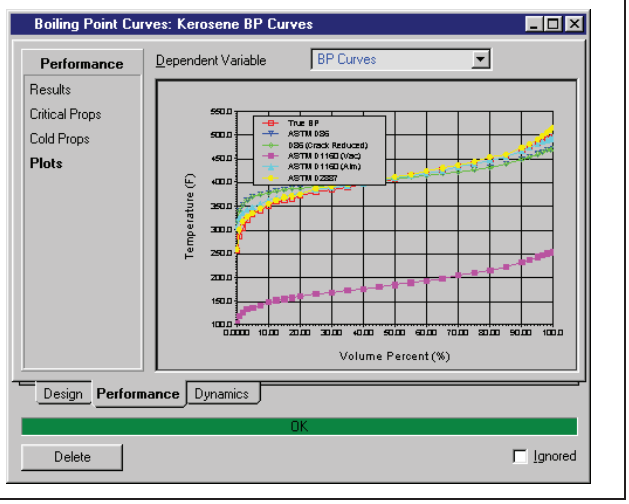
This boiling range predicted by the utility is slightly wider than the ideal range calculated during the Oil characterization procedure for Kerosene, 356°F to 464°F.

Figure 2.125



12. Select the **Plots** page on the **Parameters** tab of the utility property view to view the data in graphical format.

Figure 2.126



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

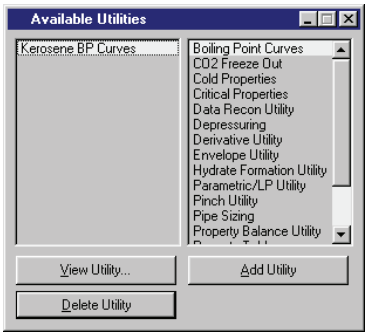
Alternative to using the **Utilities** page of a stream property view, you can also install a utility using the **Available Utilities** view. Another **BP** Curves utility is installed for stream **Residue**. This utility is used for the case study in the next section. To install the utility:

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

The Available Utilities view appears.

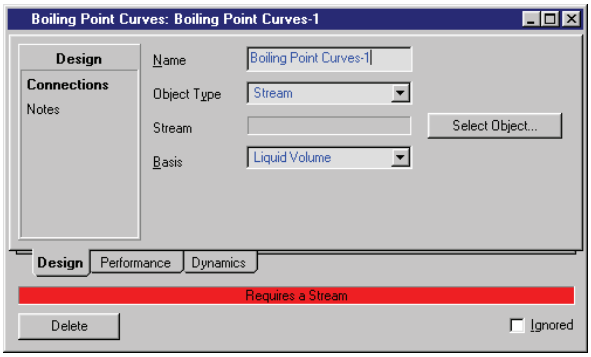
Notice the name of the utility created previously, Kerosene BP Curves, appears in the Available Utilities view.

Figure 2.127



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

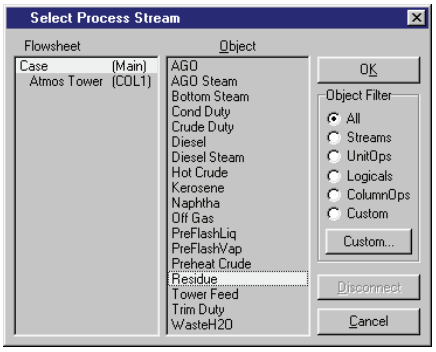
Figure 2.128



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 drop-down list. The next task is to attach the utility to a material stream.

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

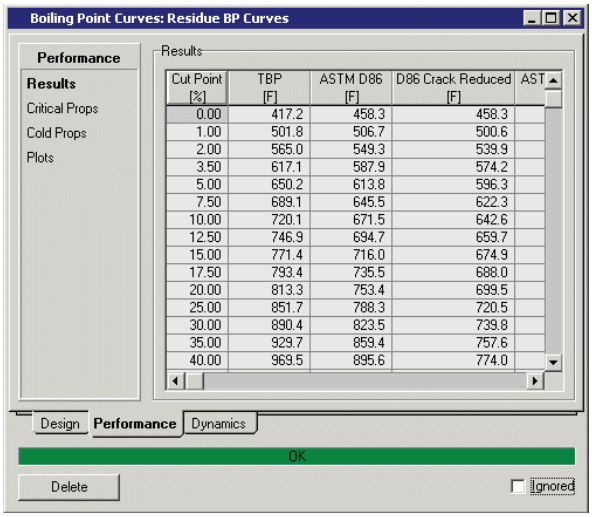
Figure 2.129



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

Notice that the stream name Residue now appears in the Stream cell.

Figure 2.130



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

2.2.11 Using the Databook

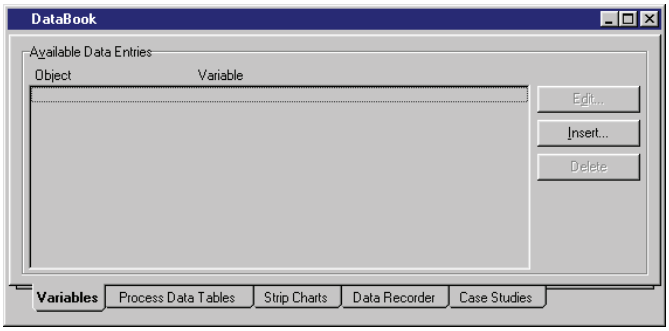
The UniSim Design **Databook** provides you with a convenient way to examine your flowsheet in more detail. You can use the **Databook** to monitor key variables under a variety of process scenarios, and view

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.

Figure 2.131



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:

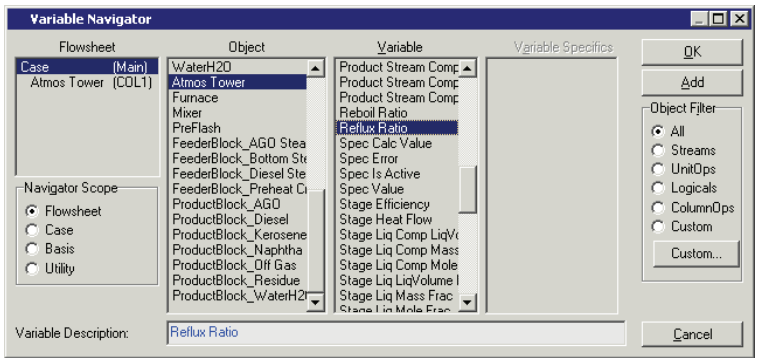
- D1160 Boiling Temperature for 5% volume cut point of stream Residue
- heat flow of energy stream Trim Duty
- column reflux ratio

1. Click the **Insert** button and the Variable Navigator view appears.
2. 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-to-right 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.

Figure 2.132

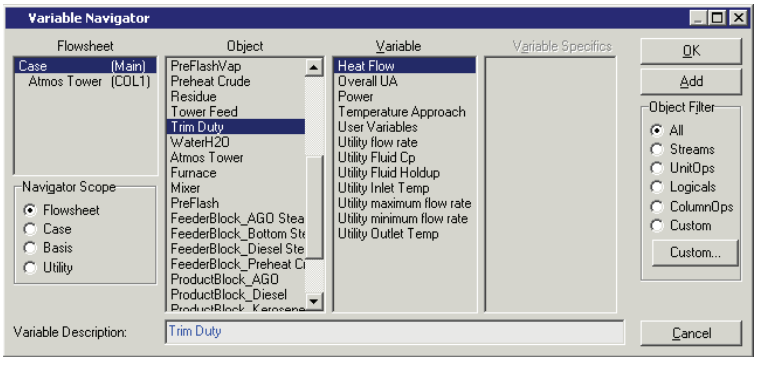


5. Click the **Add** button. The variable appears in the Databook and the Variable Navigator view remains open.
6. To add the next variable, select the **Streams** radio button in the **Object Filter** group. The **Object** list is filtered to show streams only.
7. Scroll down and click on **Trim Duty** 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:

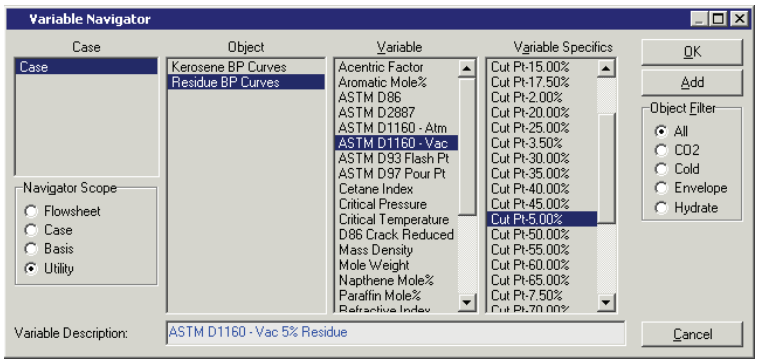
9. Click inside the **Variable Description** field and delete the default name.
10. Type a new description, such as **Trim Duty**, and click the **Add** button. The variable now appears in the Databook.

Figure 2.133



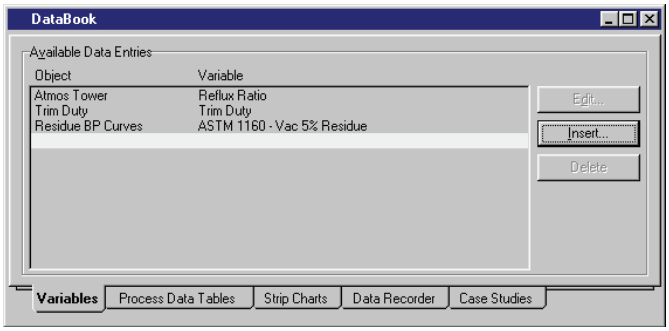
11. To add the third variable, the ASTM D1160 cut point from the **Residue BP Curves** utility, select the **Utility** radio button in the **Navigator Scope** group.
12. Select **Residue BP Curves** in the **Object** list.
13. Select **ASTM D1160 - Vac** in the **Variable** list.
14. Select Cut PT-5.00% 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.

Figure 2.134



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

Figure 2.135

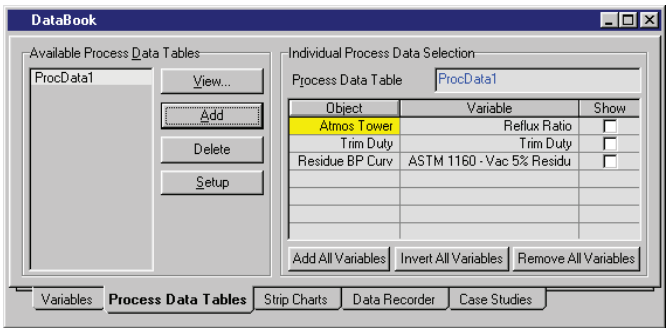


Create a Data Table

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

Figure 2.136

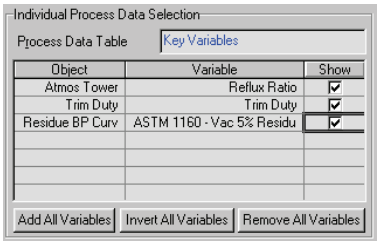


3. Change the default name from **ProcData1** to **Key Variables** by editing the **Process Data Table** field.

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

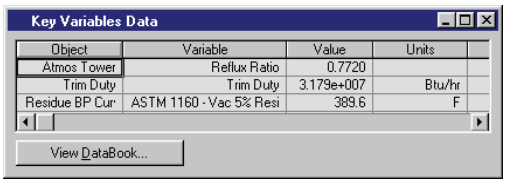
4. Activate each variable by clicking on the corresponding **Show** checkbox.

Figure 2.137



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

Figure 2.138



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

6. 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.

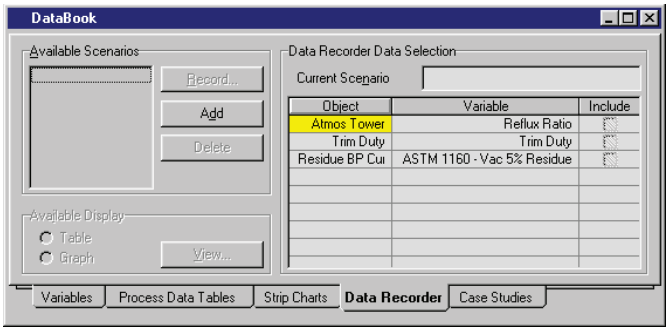
Recording Data

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.

Figure 2.139

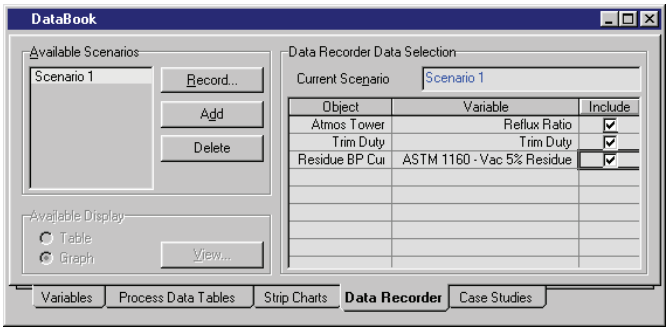


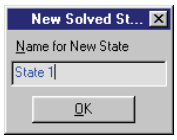
When using the Data Recorder, you first create a **Scenario** containing one or more of the key variables, then record the variables in their current state.

2. 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.

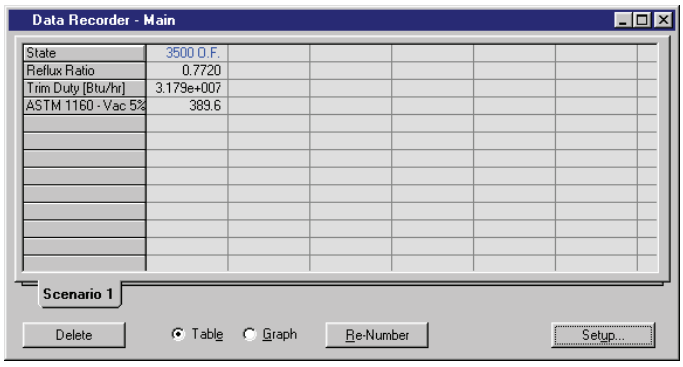
Figure 2.140





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.

Figure 2.141



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

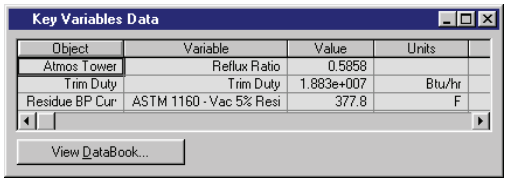
The value of the Overflash specification is going to be changed in the column and the changes is viewed in the process data table:

Object Navigator icon

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.

Figure 2.142



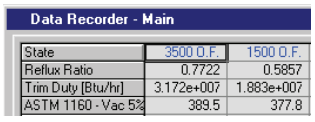
As a result of the change:

- 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.

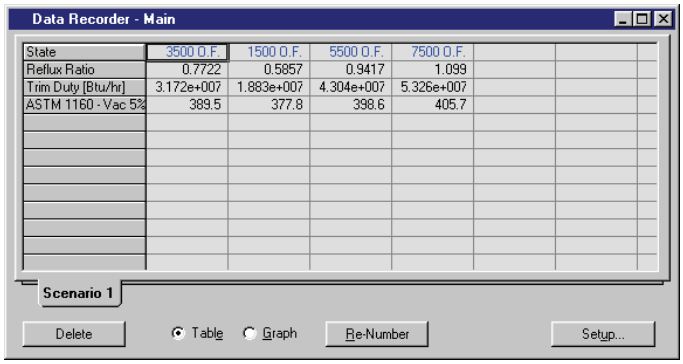
12. Click the **View** button and the **Data Recorder** appears, displaying the new values of the variables.

Figure 2.143



13. 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.

Figure 2.144



14. Save your case by doing one of the following:
    - press **CTRL S**.
    - from the **File** menu, select **Save**.
    - click the **Save** icon.

Save icon

2.3 Dynamic Simulation

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

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 pre-flash 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