Conceptual Engineering Products™ 6.2

Tutorials

aspentech

driving process profitability

Copyright Notice

Table of Contents

1 Crude Pre-Heat Train Network

1.1 Introduction

In this tutorial, you will design a heat exchanger network for a crude pre-heat train. A network will be created based on a process flow diagram proposed by a contractor. The concepts introduced here are used throughout the tutorial:

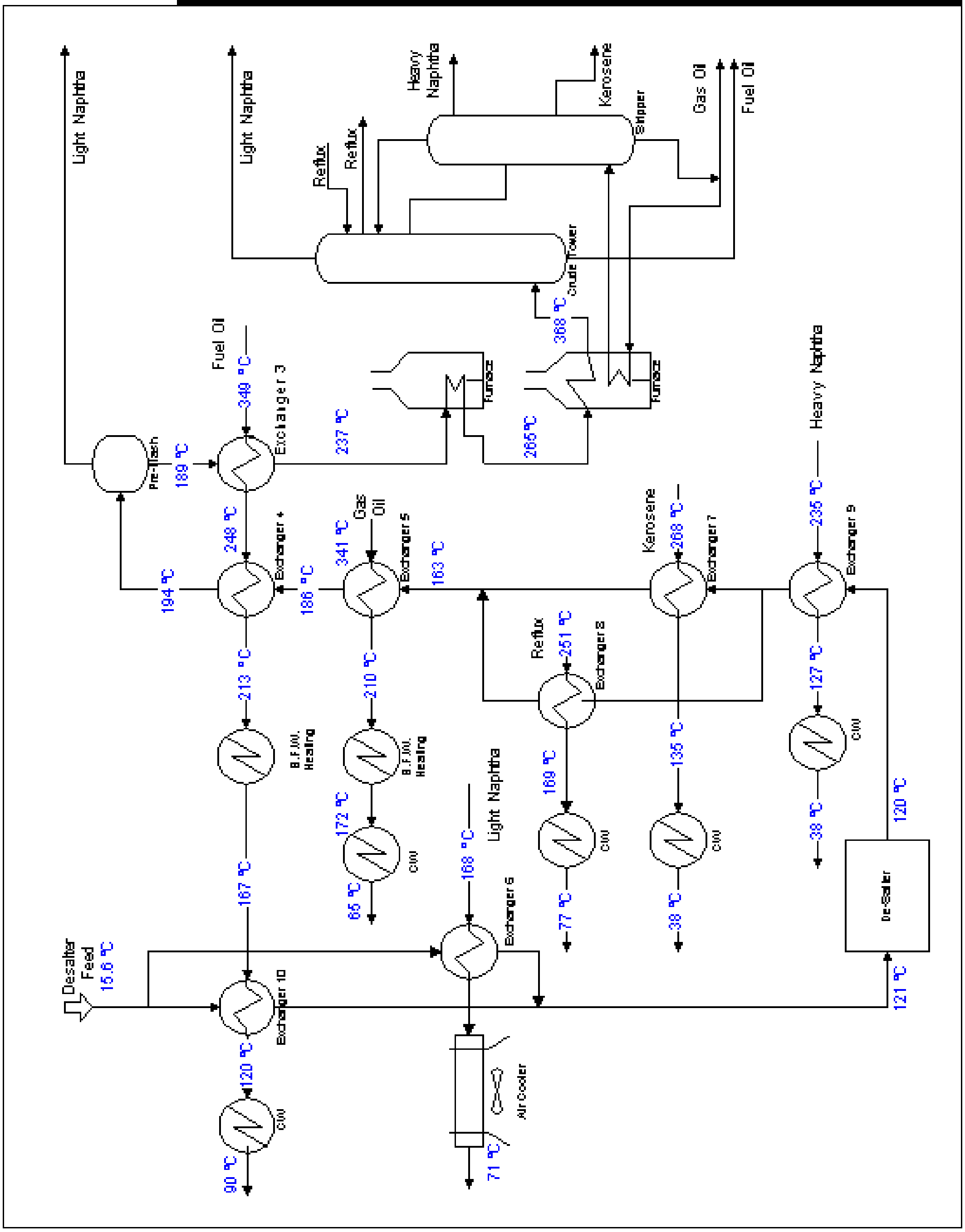
- creating process and utility streams,
- adding heat exchangers, and
- using the worksheet to manipulate the network.

Crude oil is often fractionated to produce saleable products such as heavy and light naphtha, kerosene, gas, and fuel oils. The proposed flowsheet was adapted from B. Linnhoff, D.W. Townsend, et al[1] and appears in **Figure 1.1**. The crude oil is split and heated in two heat exchangers, 10 and 6, by the Fuel Oil and Light Naphtha product streams, respectively. The crude feed is then mixed back together and passed through the desalter unit.

Effluent leaving the desalter is heated up further in heat exchangers 9, 7, 8, 5, and 4 by the product streams Heavy Naphtha, Kerosene, Reflux, Gas Oil, and Fuel Oil, respectively. An air cooler, coolers using cooling water, and boiler feed water heaters are used to further cool the product streams down to their target temperatures.

After the crude oil has been heated by the products, it is passed through a Pre-Flash operation to remove the Light naphtha cut. The heavier components from the Pre-Flash operation are heated by the hottest portion of the fuel oil in heat exchanger 3, and then passed through two furnaces. The crude tower takes the heated feed, and separates the Light Naphtha and the Fuel Oil cuts. The remaining cuts pass to the second column, the stripper. The stripper column also produces three other products: Gas Oil, Heavy Naphtha, and Kerosene.

Figure 1.1: Process Flowsheet



1.2 Entering Process Information

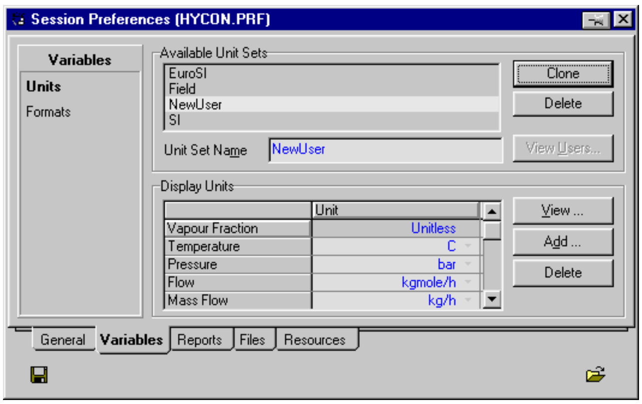
In order to analyze and create the heat exchanger network described earlier, all of the information from the pre-heat train must be entered into HX-Net. This includes all process stream information and utility stream information.

1.2.1 Setting Unit Preferences

Before you begin, verify that the units currently selected in the preferences are the ones you want to use. For this example, the desired unit for energy is MW.

1. On the **Tools** menu, click **Preferences**. The Session Preferences view appears.
2. Click on the **Variables** tab, and select the **Units** page.
3. Select **EuroSI** from the Available Unit Sets list, and click the **Clone** button. You have created a custom unit set.

Figure 1.2



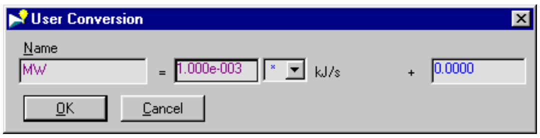
4. In the **Unit Set Name** field, change the name of the custom unit set to Energy Integration-Euro SI.

5. In the Display Units group, scroll through the table until you find **Energy**. The current energy flow unit is set to kcal/h.

There is no MW unit in the Energy unit drop-down list, only kW. Since HX-Net does not have the required unit, the MW unit will be created using the User Conversion.

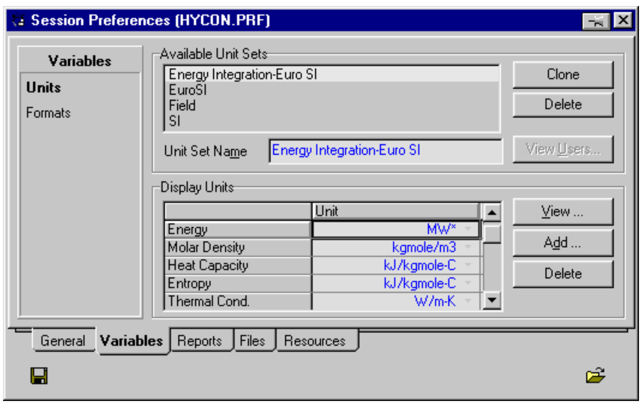
6. Click on the **Unit** cell for Energy, then click the **Add** button. The User Conversion view appears.
7. In the **Name** field, enter MW.
8. In the second field, enter the conversion factor of 1000, as shown in the figure below.

Figure 1.3



9. Click the **OK** button. The User Conversion view closes, and you return to the **Units** page. The **Energy** unit cell now displays the new unit **MW***.

Figure 1.4



Close icon

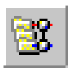
10. Click the **Close** icon to close the Session Preferences view.

1.2.2 Creating a HI Case

Now, you can create the Heat Integration (HI) Case.

To access the main HI Case view, do one of the following.

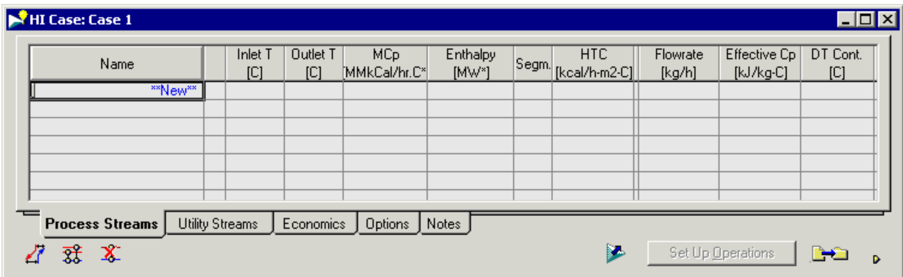
- From the **Features** menu, select **HI Case**.
- From the **Managers** menu, select **Heat Integration Manager**. The Heat Integration Manager view appears. In the left column, select **HeatIntegrationCase**, then click the **Add** button.
- On the tool bar, click the **Heat Integration Manager** icon. The Heat Integration Manager view appears. In the left column, select **HeatIntegrationCase**, then click the **Add** button.



Heat Integration Manager icon

When the HI Case view appears, and you should see the Process Streams tab as shown in the figure below.

Figure 1.5



1.2.3 Entering Process Stream Data

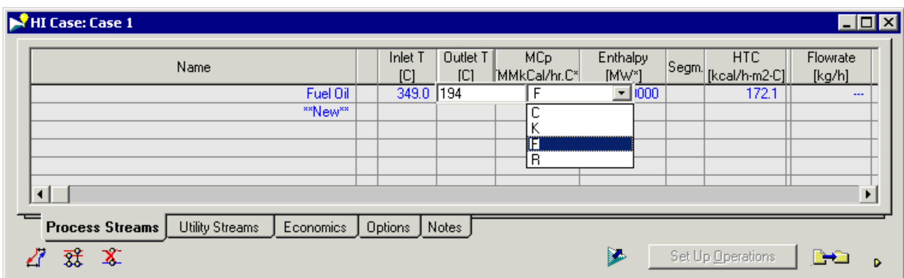
You will now begin entering the stream information, starting with the Fuel Oil stream.

1. Click in the **Name** column.
2. Type in the name Fuel Oil, then press the **ENTER** key. The cursor automatically moves to the **Inlet T** cell.
3. This stream exits from the bottom of the crude tower at a temperature of 349°C.
   Ensure that the **Inlet T** cell is active, and type **349**.
   The default unit that appears in the unit drop-down list is already degrees Celsius, so it does not need to be changed.
4. The desired outlet of this stream is 194°F. The default units are in degrees Celsius.
   Ensure that the **Outlet T** cell is active and type **194**.
   To change the units, do one of the following:
   - Click on the drop-down list that appears and select **F**.

   OR

   - Press the **SPACE BAR** and type **F**, then press **ENTER.** HX-Net automatically converts the value to the default units.

Figure 1.6



The temperature value in the Outlet T cell automatically changes from 194°F to 90°C, because degrees Celsius is the default unit.

Once the inlet and outlet temperatures are entered, HX-Net determines the stream type as hot or cold. The stream type is indicated in the second column by a red arrow for hot streams or a blue arrow for cold streams.

Segmenting Process Streams

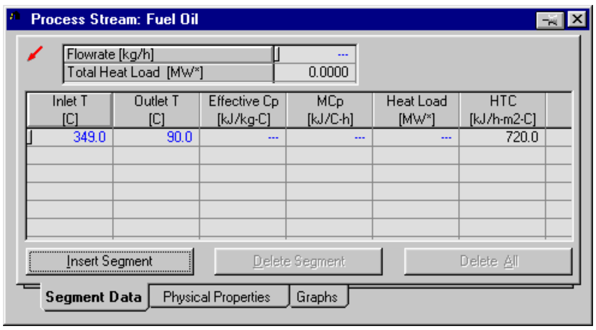
To complete this stream's information, the enthalpy or flow heat capacity must be entered. All other information is optional.

For this example, the Fuel Oil stream must be segmented. Stream segmentation is extremely useful for streams that change phase or have non-linear variations in enthalpy as the temperature changes.

Double-clicking in the HTC column opens the HTC Default Values view, which displays a list of default heat transfer coefficients for various materials.

1. Double-click in any cell of the **Fuel Oil** row (except for the HTC column) to open the Process Stream view.

Figure 1.7



2. To add a segment, click in the target outlet temperature cell (90°C) and click the **Insert Segment** button.
   A blank row appears above the target outlet temperature.
3. The outlet temperature of the first segment is 243°C.
   Click in the empty **Outlet T** cell and type 243.
4. The enthalpy for this section is 22.8 MW.
   Click in the **Heat Load** cell and type 22.8.

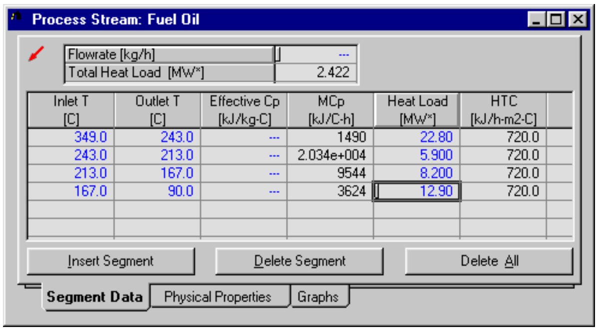
Always click the target Outlet T cell before clicking the Insert Segment button. If you insert the segment values in the wrong order, a warning appears.

5. Repeat steps #2,#3 and #4 to add more segments with the following information:

Segment Outlet T (°C)	Heat Load (MW)
213	5.9
167	8.2
90	12.9

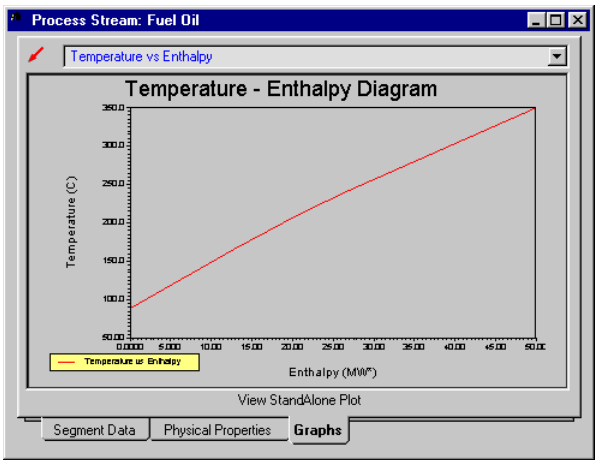
6. After entering the information for the last segment (167°C to 90°C), the process stream is complete as should appear as shown in the figure below.

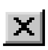
Figure 1.8



7. To view the temperature versus enthalpy plot, click on the **Graphs** tab. The plot appears similar to the figure below.

Figure 1.9





Close icon

8. Click the **Close** icon to return to the **Process Streams** tab of the HI Case view.
The red arrow beside the stream name indicates it is a hot stream.

Now that you know how to successfully enter process stream information and create segmented streams, enter the following stream information.

Enter the stream name, first Inlet T value and the target (last) Outlet T value on the Process Streams tab before accessing the Process Stream view to enter the segment information.

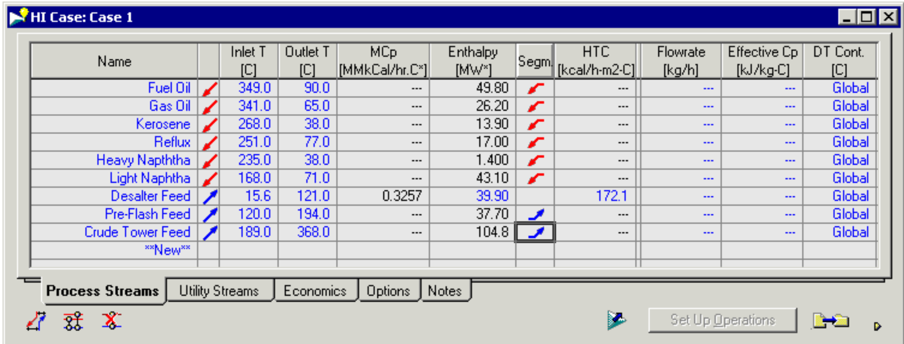
Enter only the Outlet T values and the Heat Load/Enthalpy values; the Inlet T values are calculated for you.

If you try to enter the segment values in the wrong order, a warning appears.

Stream Name	Inlet T (°C)	Outlet T (°C)	Heat Load/Enthalpy (MW)
Gas Oil	341	210	13.8
	210	172	3.6
	172	111	5.3
	111	65	3.5
Kerosene	268	135	8.7
	135	38	5.2
Reflux	251	169	8.6
	169	77	8.4
Heavy Naphtha	235	127	0.8
	127	38	0.6
Light Naphtha	168	136	19.2
	136	118	8.6
	118	108	4.1
	108	71	11.2
Desalter Feed	15.6	121	39.9
Pre-Flash Feed	120	122	0.8
	122	163	17.3
	163	186	13.8
	186	194	5.8
Crude Tower Feed	189	237	22.9
	237	265	13.9
	265	368	68

9. After entering all the information in the above table, the **Process Streams** tab in the HI Case view appears similar to the figure below.

Figure 1.10

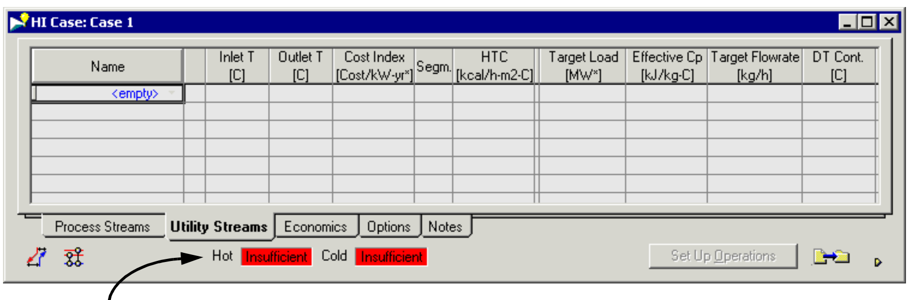


1.2.4 Entering Utility Stream Data

Now that all of the process stream information has been specified, the utilities to be used for heating and cooling must be specified.

1. On the HI Case view, click on the **Utility Streams** tab.
   The utilities for this example will be selected from the list of default utilities included with HX-Net.
2. In the **Name** column, move the cursor over the **<empty>** cell to activate the down arrow .

Figure 1.11

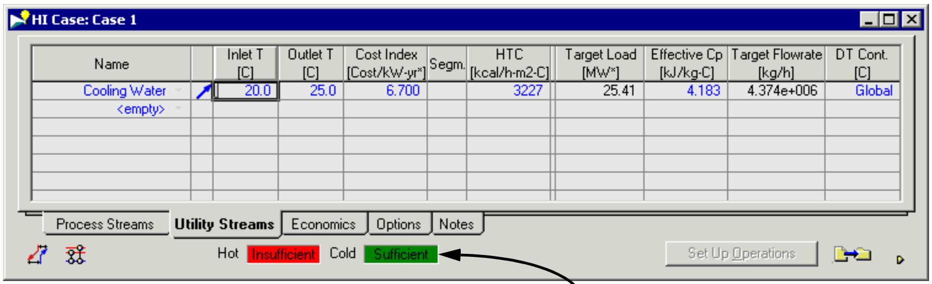


The Hot and Cold status bars appear below the tab when the Utility Streams tab is selected. The status bars indicate that there are not enough hot and cold utilities to satisfy the process streams.

3. Click the down arrow to open the drop-down list.

4. Scroll through the list until you find **Cooling Water,** then select it.

Figure 1.12



The Cold status bar indicates that cold utilities are now sufficient. This means that the cold utility entered can be used to cool all of the hot process streams.

5. Repeat steps #2 - #4 to add the following utilities:
   - LP Steam Generation
   - Air
   - Fired Heat (1000)

The **Utility Streams** tab should now appear similar to the figure below.

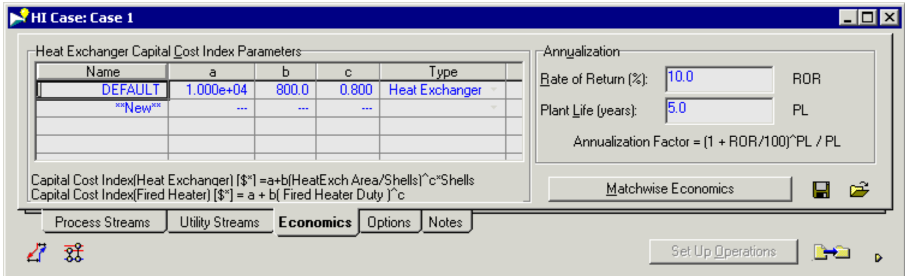
Figure 1.13

The Hot status bar now displays a sufficient status.

All utilities have default costs associated with them. This cost information is required to calculate the operating cost targets for the system.

6. Click the **Economics** tab.

Figure 1.14

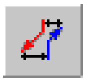


A default set of economic parameters is supplied by HX-Net.

Since at least one set of economic data must be available for the calculation of the capital cost targets and network capital costs, this information is left as is.

1.3 Examining the Targets

Now that all of the data required to create the crude pre-heat train network has been entered, you can examine the various engineering targets calculated by HX-Net. These targets represent the performance of an ideal heat exchanger network.

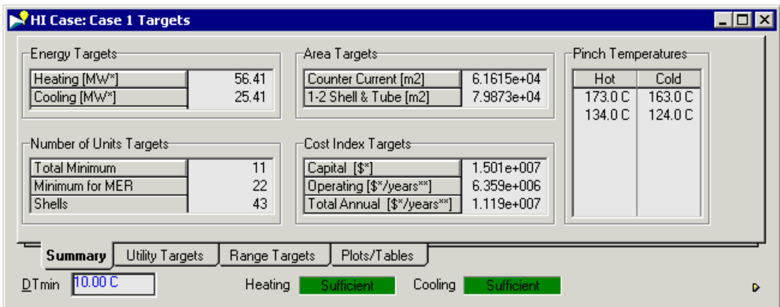


Open Targets View icon

To open the Targets view:

1. On the HI Case view, click on the **Open Targets View** icon that appears at the bottom of the view for all tabs.
2. The Targets view appears.

Figure 1.15



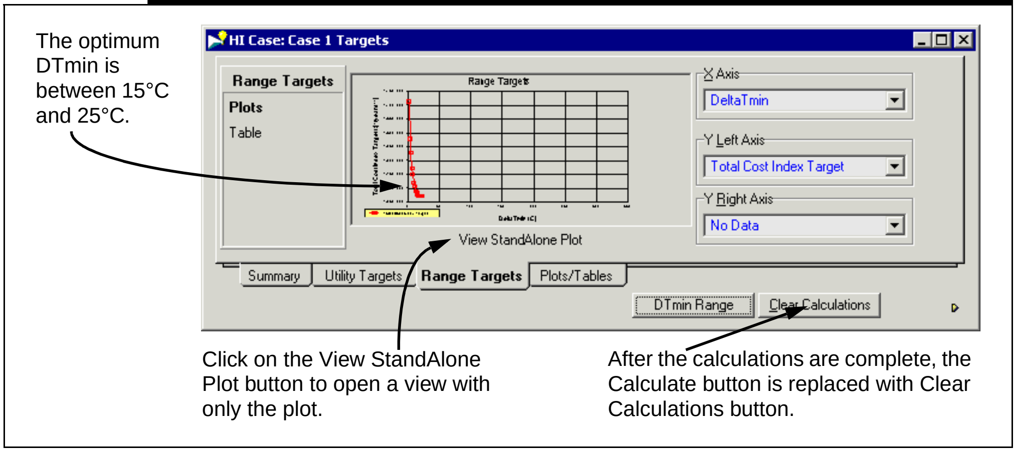
The minimum number of units required to build this heat exchanger network is 11. The network you will build is based on **Figure 1.1: Process Flowsheet**, which has 19 exchangers (including heaters and coolers). Therefore, the network that will be built will be over the unit targets. The network, however, may require less energy from the utility streams than the targets shown in the Energy Targets group.

1.3.1 Range Targeting

One of the main objectives when creating a heat exchanger network is to minimize the capital and operating costs. The minimum approach temperature, DTmin, is a key parameter in defining the balance between capital and operating costs. You can find the optimal value for DTmin by performing a Range Targeting calculation.

1. On the Targets view, click on the **Range Targets** tab.
2. Click the **Calculate** button, located below the **Range Targets** tab, to perform the calculations.
3. Click on the **Plots** page to view the plot, which displays the Total Cost Index Target vs. DTmin, as shown in the figure below.

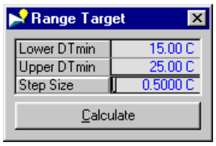
Figure 1.16



4. To find a better approximation of the optimal DTmin value, you need to narrow the calculations.
   Click the **Clear Calculations** button to clear the plot before you perform a new calculation.
5. Click the **DTmin Range** button.
   The Range Targets view appears. This view allows you to specify the range of calculations.
6. In the **Lower DTmin** cell, enter 15.
7. In the **Upper DTmin** cell, enter 25.
8. In the **Interval Size** cell, enter 0.5. This is the step size that will be used between the lower and upper DTmin values.

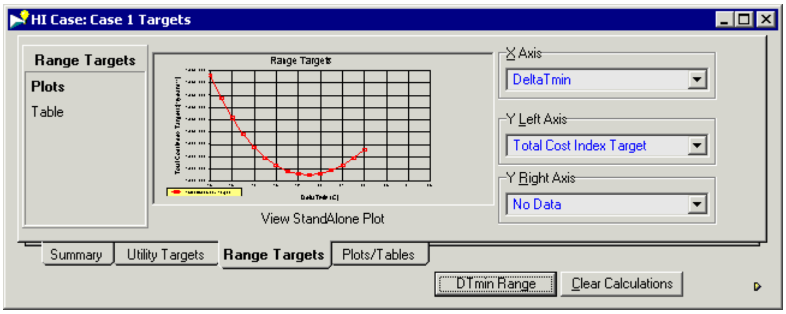
The Range Target view should appear like the figure below:

Figure 1.17



9. Click the **Calculate** button. HX-Net automatically closes the Range Target view and performs the new calculation. The results indicate that the optimal DTmin value is 19.5°C.

Figure 1.18



10. To verify this value, click on the **Table** page.
11. Examine the values in the **Total Cost Index** column for the minimum value. Move across the row to find the corresponding DTmin value. It is 19.5°C.

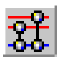
The new plot shows that the optimal DTmin value is 19.5°C. For the purpose of this application, however, leave the DTmin value at 10°C.

Close icon

12. Click the **Close** icon to close the Targets view and return to the main HI Case view.

1.4 Building the Heat Exchanger Network

1.4.1 Accessing the HEN Design View

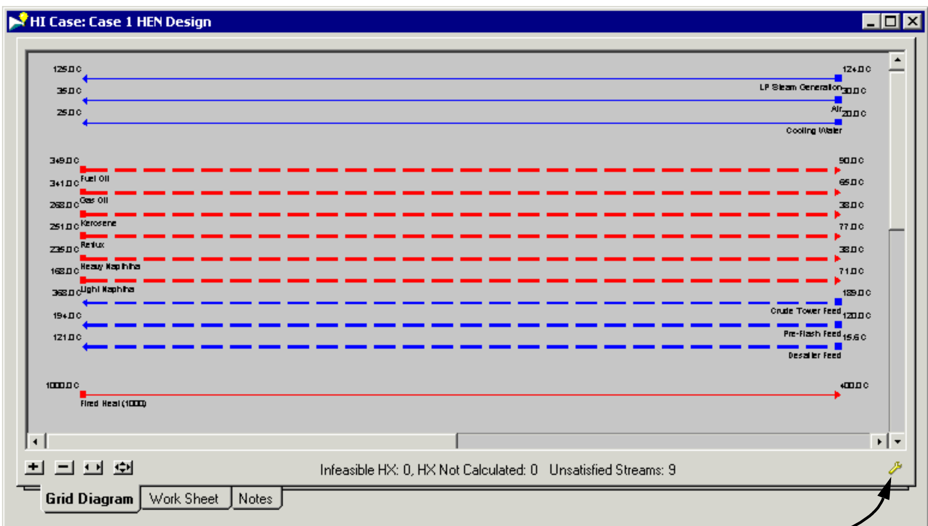


Open HEN Grid Diagram icon

To access the heat exchanger network (HEN) design:

1. In the HI Case view, click the **Open HEN Grid Diagram** icon.
2. The Heat Exchanger Network (HEN) Design view appears with the **Grid Diagram** tab active, as shown in the figure below.

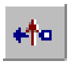
Figure 1.19



The Open Palette View icon allows you to access the Design Tools palette.

1.4.2 Modifying HEN Diagram Properties

Open Palette View icon



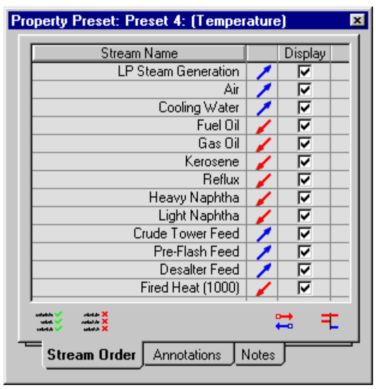
Open HEN Diagram Properties View icon

1. To open the Property Presets view, do one of the following:
   - On the HEN Design view, click the **Open Palette View** icon. The Design Tools palette appears. On the Design Tools palette, click the **Open HEN Diagram Properties View** icon.

   OR

   - Right-click on the Grid Diagram, and select **Properties** from the Object Inspect menu.
2. From the list in the Property Presets view, select **Preset 4: (Temperature)**. This property preset sorts the streams by their temperatures and displays both process and utility streams.
3. Click the **Edit** button. The Property Preset: Preset 4 view appears.

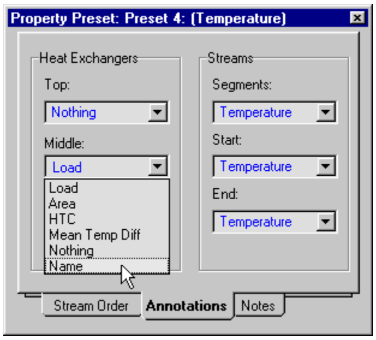
Figure 1.20



4. Click the **Annotations** tab.

5. In the **Heat Exchangers** group, click the **Middle** drop-down list and select **Name** as shown in the figure below.
   The heat exchanger name will now appear above the heat exchanger object in the Grid Diagram.

Figure 1.21



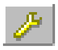
Close icon

6. Click the **Close** icon to close the Property Preset 4:(Temperature) view and the Property Presets view.

1.4.3 Adding a Splitter

To maintain a logical order, you will place the heat exchangers in a position similar to that shown on the Process Flowsheet. The crude oil feed is split before it enters heat exchanger 10 and 6 on the Process Flowsheet, so we must split the Desalter Feed stream.

In this procedure, you will add a splitter to the Desalter Feed stream in the flowsheet.

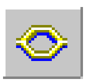


Open Palette View icon

To add heat exchangers, mixers and splitters, the Design Tools palette must be available.

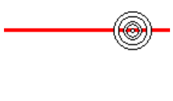
1. Click the **Open Palette View** icon, located in the bottom right corner of the **Grid Diagram** tab.
   The Design Tools palette appears.

Figure 1.22

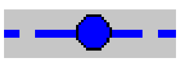


Add Split icon

2. Right-click and hold on the **Add Split** icon.
3. Drag the cursor over the Desalter Feed stream until the **Bull's eye** icon appears.
4. Now release the mouse button. The splitter appears as a solid blue dot.
5. To expand the splitter, click the blue dot once. The stream will now appear as shown in the figure below.

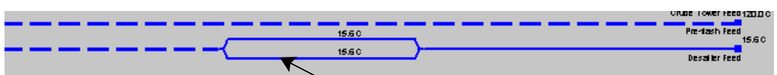


Bull's eye icon



Blue Dot icon

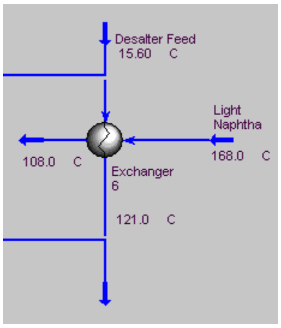
Figure 1.23



Each segment of the split is called a branch.

The Desalter Feed stream exchanges heat with the Light Naphtha stream in exchanger 6 and the Fuel Oil stream in exchanger 10. You will now place the first heat exchanger on the Desalter Feed stream.

1.4.4 Adding Heat Exchangers

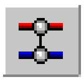


Heat Exchanger 6

Adding Heat Exchanger 6

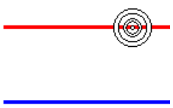
In this procedure, you will place an exchanger between the Light Naphtha stream and the Desalter Feed stream. This is Exchanger 6 on the Process Flowsheet.

The streams on the Grid Diagram appear dashed because the energy in the streams has not been satisfied. As heat exchangers are placed and stream energy is satisfied, the lines representing the streams changes from dashed to solid.

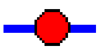


Add Heat Exchanger icon

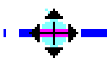
1. In the Design Tools palette, right-click and hold on the **Add Heat Exchanger** icon.
2. Drag the cursor to the top branch of the split on the Desalter Feed stream until the **Bull's eye** icon appears.
3. Release the mouse button. The heat exchanger appears as a sold red dot.
4. Click and hold on the red dot, then drag the cursor to the Light Naphtha stream. A light blue dot will appear underneath the cursor as you drag it to the new stream.
5. Release the mouse button. The heat exchanger appears.



Bull's eye icon



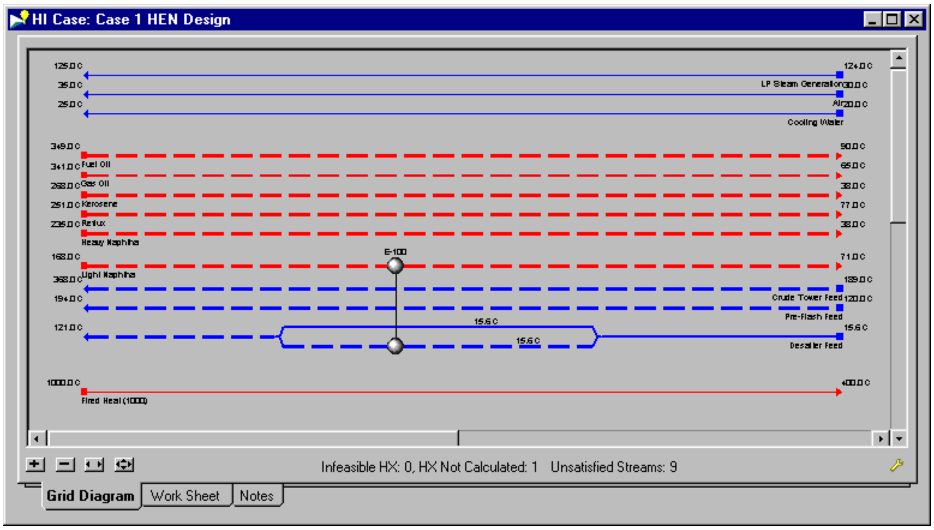
Red Dot icon



Light blue dot icon
(under four arrows)

The **Grid Diagram** tab should now appear as shown in the figure below.

Figure 1.24

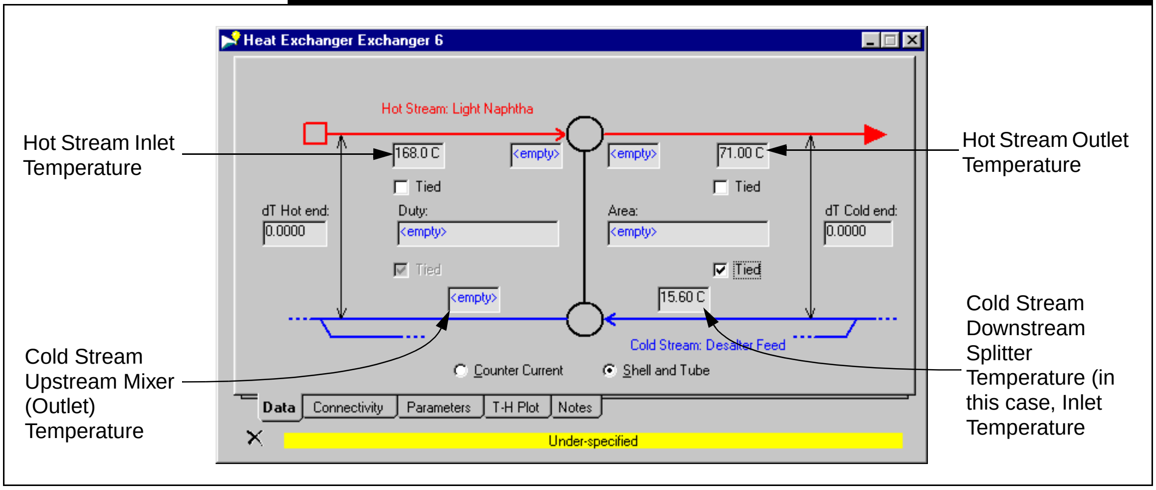


HX-Net may appear to move the heat exchanger to the bottom branch. If this happens, adding a heat exchanger to the second branch later in the tutorial will result in Exchanger 6 showing on the top branch (see **Figure 1.28**).

6. Double-click either end of the heat exchanger (the grey circles) to open the Heat Exchanger Editor view.
7. Click the **Notes** tab.
8. In the **Name** field, type **Exchanger 6**.
9. Click the **Data** tab.

10. Click the **Tied** checkbox by the Desalter Feed stream (cold stream) inlet temperature field. The view now appears as shown in the figure below.

Figure 1.25



The Desalter Feed stream passes through the hottest part of the Light Naphtha stream, therefore, you can tie the inlet temperature of the hot stream entering the heat exchanger.

Since the Desalter Feed stream is being heated from its initial inlet temperature, you can “tie” this value to the inlet temperature value found on the **Process Streams** tab on the main HI Case view.

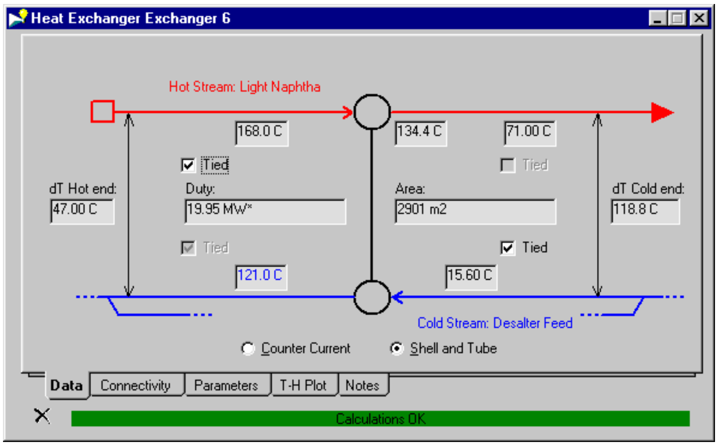
11. Click the **Tied** checkbox by the hot stream inlet temperature field. Before the calculation can occur, you must specify the cold stream outlet temperature.
12. In the cold stream outlet temperature field, enter **121°C**. This value comes from the Defaulter Stream outlet target temperature on the Process Flowsheet.

All values in blue have been entered by the user and can be altered.

All values in black have been calculated by HX-Net and cannot be altered.

The heat exchanger now solves and appears as shown in the figure below.

Figure 1.26

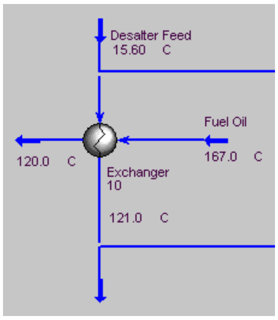


Close icon

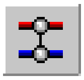
13. Click the **Close** icon to close the Exchanger 6 property view.

Adding Heat Exchanger 10

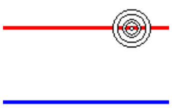
Figure 1.27



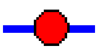
Heat Exchanger 10



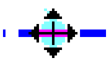
Add Heat Exchanger icon



Bull's eye icon



Red Dot icon

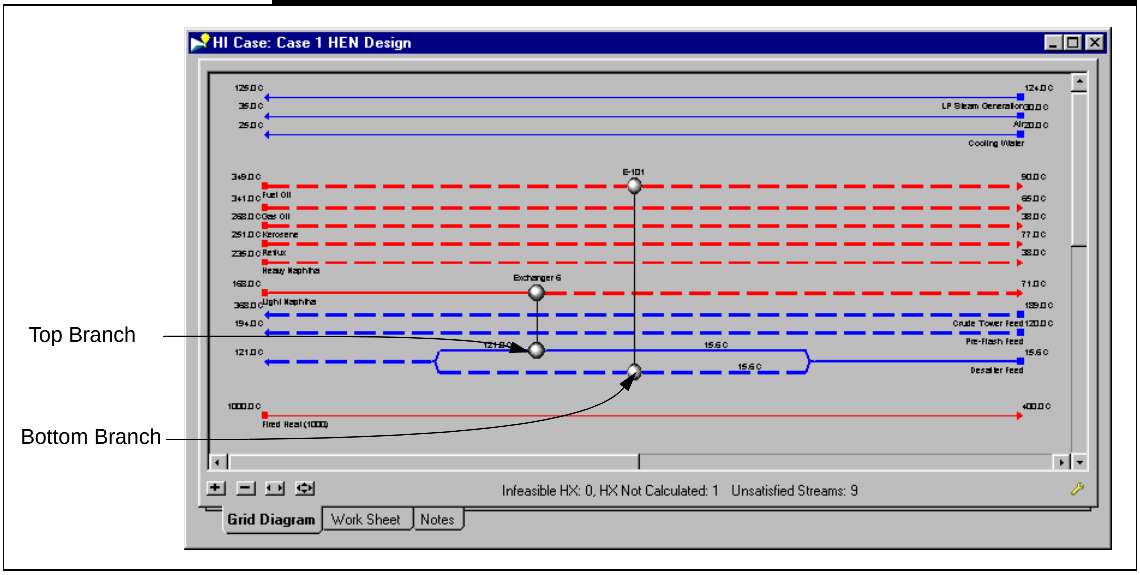


Light blue dot icon
(under four arrows)

1. In the Design Tools palette, right-click and hold on the **Add Heat Exchanger** icon.
2. Drag the cursor to the empty branch of the split on the Desalter Feed stream until the **Bull's eye** icon appears.
3. Release the mouse button. The heat exchanger appears as a sold red dot.
4. Click and hold on the red dot, then drag the cursor to the Fuel Oil stream. A light blue dot will appear underneath the cursor as you drag it to the new stream.

5. Release the mouse button. The heat exchanger appears.
   The **Grid Diagram** tab should now appear as shown in the figure below.

Figure 1.28

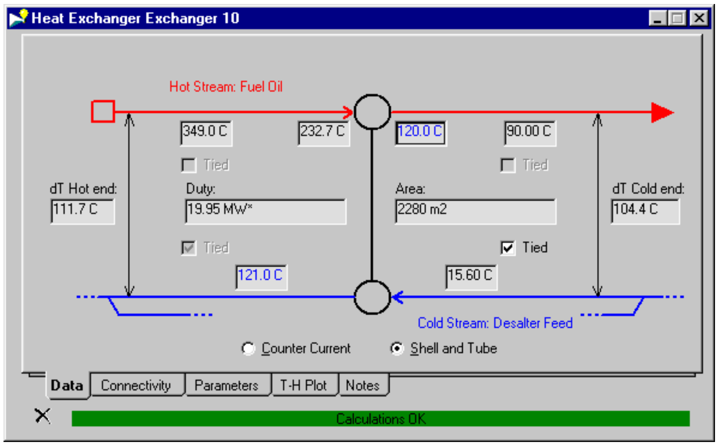


6. Double-click on either end of the heat exchanger (the grey circles) to open the Heat Exchanger Editor view.
7. Click the **Notes** tab.
8. In the **Name** field, type **Exchanger 10**.
9. Click the **Data** tab.
10. Click the **Tied** checkbox by the Desalter Feed cold stream inlet temperature field.
11. In the Desalter Feed cold stream outlet temperature field, enter **121°C**, then press the ENTER key.
12. In the empty hot stream outlet temperature field, type **120°C**, then press the ENTER key. The exchanger solves.

According to the Process Flowsheet, the Desalter Feed is being heated from its inlet temperature of 15.6°C to its target temperature, which is also the mixer temperature of 121°C.

We also know the Fuel Oil stream temperature exits the exchanger at 120°C.

Figure 1.29

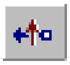


13. Click the **Close** icon to close the property view.

HX-Net automatically calculates the inlet temperature of the hot stream to be 232.7°C. From the Process Flowsheet, however, you know that the temperature of the hot stream entering the exchanger should be 167°C. In order to adjust this temperature, you must change the balance of flows going through the splitter on the Desalter Feed stream.

Adjusting the Split Ratio

Before you adjust the split ratio, ensure the split ratio values are visible on the Grid Diagram view.

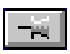


Open HEN Properties View icon

1. On the Design Tools palette, click the **Open HEN Diagram Properties View** icon. The Property Presets view appears.
2. Click the **New** button. The New Property Preset view appears.
3. In the **Name** field, type **Split Ratio**, then press ENTER.
   The Property Presets view reappears, and Split Ratio appears in the list.
4. In the list, select Split Ratio, then click the **Edit** button.
   The Property Preset: Split Ratio view appears.
5. Click the **Annotations** tab.
6. In the Streams group, click the Segments drop-down list and select **Split Fraction**.
7. Click the **Close** icon on both the Property Preset: Split Ratio view and the Property Preset view to close them.
   The split ratio values now appear on the Grid Diagram view for the Desalter Feed stream. The default ratio is 0.5:0.5.

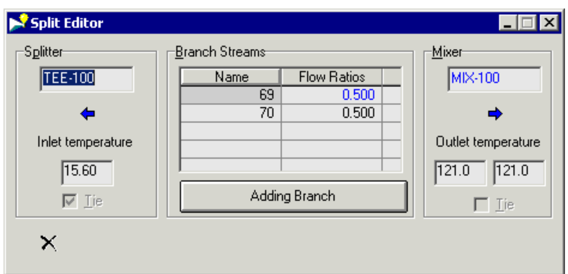
Now you will adjust the split ratio.

Remember, the heat exchanger views and Split Editor view are Modal views. To open all views, you have to click the **Pin** icon and change the Modal views to Non-Modal views.



1. Double-click on either end of Exchanger 10 to open the heat exchanger view. You want this view open so you can observe the changes in the hot stream inlet temperature.
2. Open the splitter view by double-clicking on either end of the splitter. The Split Editor view appears as shown in the figure below.

Figure 1.30



3. Arrange the heat exchanger view and the Split Editor view so you can see both views clearly.

To decrease the inlet temperature for Exchanger 10, you must transfer less energy in the heat exchanger. An effective way of doing this is to decrease the flow of the cold stream.

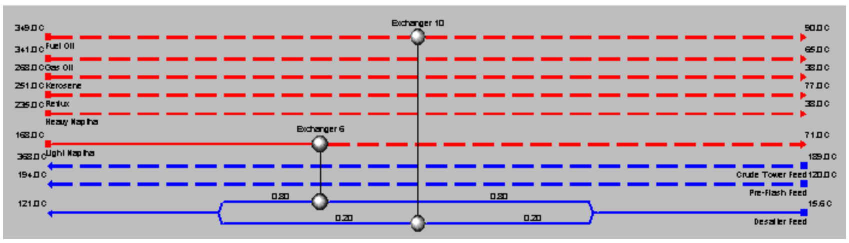
The rows in the Branch Streams table represent the two branches of the split.

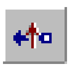
Examine the Grid Diagram to confirm which table row affects which branch of the split.

4. In the Split Editor view, Flow Ratios column, click in the top cell with the blue text. Change the flow ratio value from 0.5 to **0.75** and observe the inlet temperature change in the heat exchanger view.
5. Continue to adjust the split ratio until the hot stream inlet temperature for Exchanger 10 is about 167°C. The split ratio will be approximately 0.2 for Exchanger 10 and 0.8 for Exchanger 6.
6. Close the Split Editor view and the Exchanger 10 property views.

The line representing the Desalter Feed stream is now solid. This means that this stream's energy requirements have been satisfied.

Figure 1.31





Open HEN Properties View icon

7. Open the Property Presets view by clicking the **Open HEN Diagram Properties View** icon in the Design Tools palette.
8. Select the **Preset 4: Temperature** to display the temperature value above the streams for each segment.
9. Click the **Close** icon on the Property Preset view to close the view.

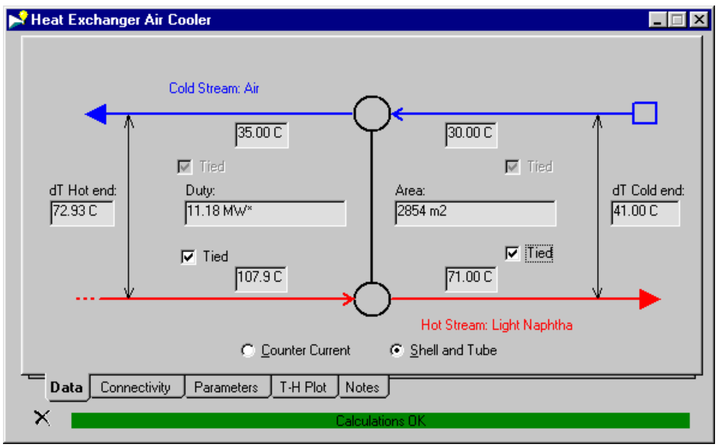
Adding Two Coolers

You will now place an air cooler on the Light Naphtha stream and a cooler on the Fuel Oil stream.

1. Follow steps #1 through #5 in **Section 1.4.4 - Adding Heat Exchangers** to add an air cooler between the Air stream and Light Naphtha stream, downstream from Exchanger 6.
2. Open the Heat Exchanger Editor view by double-clicking on either end of the newly inserted heat exchanger.
3. Click the **Notes** tab. In the **Name** field, type **Air Cooler**.
4. Click the **Data** tab.
5. Click the **Tied** checkbox by the hot stream inlet temperature field.
6. Click the **Tied** checkbox by the hot stream outlet temperature field. The heat exchanger now solves.

From the Process Flowsheet, you know the air cooler cools the Light Naptha stream from the Exchanger 6 outlet temperature of 107°C to the stream target temperature of 71°C.

Figure 1.32



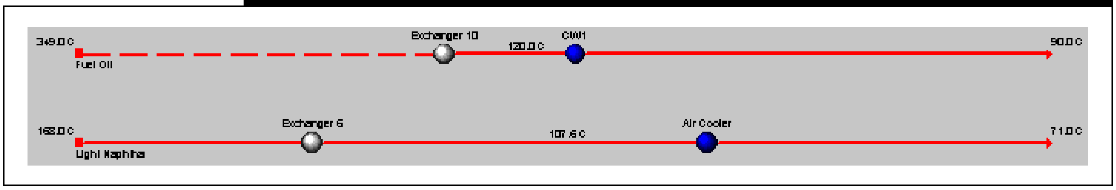
7. Add a cooler between the Fuel Oil stream and the Cooling Water utility stream, downstream from Exchanger 10 on the Fuel Oil stream.
8. Open the Heat Exchanger Editor view by double-clicking on either end of the newly inserted heat exchanger.
9. Click the **Notes** tab. In the **Name** field, type **CW1**.
10. Click the **Data** tab.

From the Process Flowsheet, you know the cooling water utility cools the rest of the Fuel Oil stream from the Exchanger 10 outlet temperature of 120°C to the stream target temperature of 90°C.

11. Click the **Tied** checkbox by the hot stream inlet temperature field.
12. Click the **Tied** checkbox by the hot stream outlet temperature field. The exchanger solves.

The Fuel Oil and Light Naphtha streams should appear as shown in the figure below.

Figure 1.33



1.4.5 Using the Worksheet to Enter Heat Exchanger Information

After the desalter operation, the crude oil is represented by the Pre-Flash Feed stream. This stream will first pass through Exchanger 9 and transfer heat with the Heavy Naphtha stream.

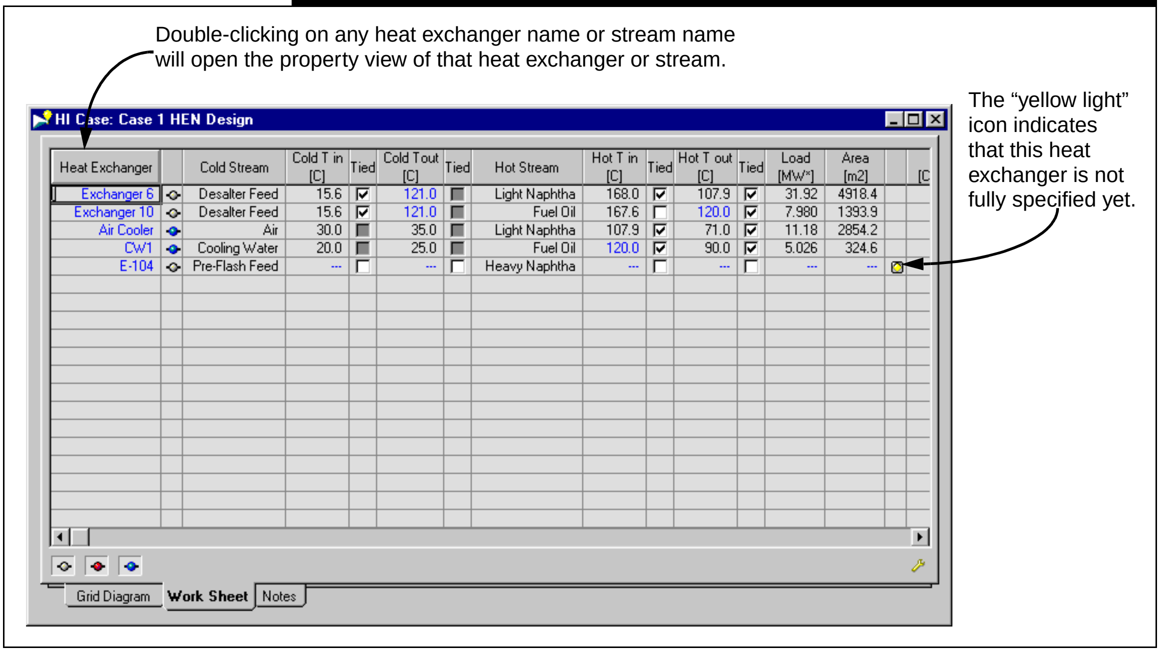
You can only add heat exchangers in the **Grid Diagram** tab.

In the following procedure, you will use the **Work Sheet** tab to modify the heat exchangers:

1. Close any property views that are open.
2. Follow steps #1 through #5 in **Section 1.4.4 - Adding Heat Exchangers** to add a heat exchanger between the Pre-Flash Feed stream and the Heavy Naphtha stream.

3. On the HEN Design view, click the **Work Sheet** tab (see figure below). The newly added heat exchanger is named E-104.

Figure 1.34



From the Process Flowsheet, you know that both the Heavy Naptha stream and the Pre-Flash Feed stream enter Exchanger 9 at their inlet temperatures.

You also know the Pre-Flash Feed stream exits the exchanger at 122°C.

4. In the **Heat Exchanger** column, click in the cell with E-104 and type **Exchanger 9**.
5. In the Exchanger 9 row, click the checkbox in the **Tied** column beside the **Cold T in** column.
6. Click the checkbox in the **Tied** column beside the **Hot T in** column.
7. Click in the **Cold T out** cell and enter **122°C**. The final values are calculated and appear on the worksheet. The "yellow light" icon disappears, indicating a fully solved heat exchanger.
8. Click on the **Grid Diagram** tab.
9. To satisfy the rest of the energy in the Heavy Naphtha stream, follow steps #1 through #5 in **Section 1.4.4 - Adding Heat Exchangers** to add a heat exchanger between the Heavy Naphtha stream and the Cooling Water utility stream.
10. Click the **Work Sheet** tab and rename the new exchanger **CW2**.
11. In the CW2 row, click the checkboxes in the **Tied** columns beside the **Hot T in** and **Hot T out** columns. The exchanger solves.

1.4.6 Completing the Pre-Flash Section

To complete the section between the desalter operation and the pre-flash operation, four more exchangers and five coolers must be added. You can do this by using just the **Grid Diagram** tab, or both **Grid Diagram** and **Work Sheet** tabs.

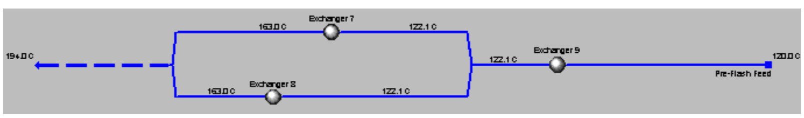
Satisfying the Kerosene and Reflux Streams

1. Add a splitter to the Pre-Flash Feed stream downstream from Exchanger 9. Refer to steps #1 through #5 in **Section 1.4.3 - Adding a Splitter**.

If necessary, refer to **Section 1.4.4 - Adding Heat Exchangers** to review the procedure for adding a heat exchanger.

2. Add a heat exchanger between the top branch of the split Pre-Flash Feed stream and the Kerosene stream.
3. Rename the new exchanger **Exchanger 7**.
4. Click the **Tied** checkbox for both the hot and cold stream inlet temperatures.
5. In the cold stream outlet temperature field, enter **163°C**. This is also the mixer temperature.
6. Add a heat exchanger on the other/empty branch of the Pre-Flash Feed and connect the exchanger to the Reflux stream.
7. Rename the new exchanger **Exchanger 8**.
8. Click the **Tied** checkbox for both the hot and cold stream inlet temperatures.
9. In the cold stream outlet temperature field, enter **163°C**. This is also the mixer temperature.
   After the addition of these two exchangers, the Pre-Flash Feed stream should appear similar to the figure below.

Figure 1.35

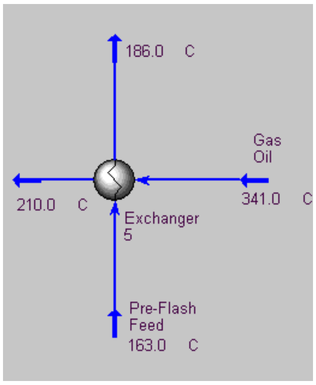


If necessary, refer to steps #1 through #6 of **Adjusting the Split Ratio**.

10. Adjust the split ratio of the Pre-Flash Feed stream, if required. Ensure that the hot stream outlet temperature of Exchanger 7 is 135°C**,** and the hot stream outlet temperature of Exchanger 8 is 169°C. The default split of 0.50 to each branch should be sufficient.

11. Add a heat exchanger between the Cooling Water utility and the Reflux stream, downstream from Exchanger 8.
12. Rename the new exchanger **CW3**.
13. Click the **Tied** checkboxes for both the hot stream inlet and outlet temperatures. The exchanger solves.
14. Add a heat exchanger between the Cooling Water utility and the Kerosene stream, downstream from Exchanger 7.
15. Rename the new exchanger **CW4**.
16. Click the **Tied** checkboxes for both the hot stream inlet and outlet temperatures. The exchanger solves.
17. Close any property views that are open.

Satisfying the Gas Oil Stream



Heat Exchanger 5

1. Add a heat exchanger between the Pre-Flash Feed stream, downstream from the splitter, and the Gas Oil stream.
2. Rename this heat exchanger **Exchanger 5**.
3. Click the **Tied** checkboxes for both inlet temperatures.
4. In the hot stream outlet temperature field, enter **210°C**. This value comes from the Process Flowsheet
5. Add another heat exchanger between the Gas Oil stream and the LP Steam Generation utility, downstream from Exchanger 5.
6. Rename this heat exchanger **BFW Heating 1**.
7. Click the **Tied** checkbox for the hot stream inlet temperature only.
8. In the hot stream outlet temperature field, enter **172°C**.
9. Add one more heat exchanger between the Gas Oil stream and the Cooling Water utility, downstream from BFW Heating 1.
10. Rename this heat exchanger **CW5**.
11. Click the **Tied** checkboxes for the hot stream inlet and outlet temperatures. The exchanger solves.

Satisfying the Pre-Flash Feed Stream

This is the last heat exchanger required to satisfy the Pre-Flash stream.

1. Add a heat exchanger between the Pre-Flash Feed stream and the Fuel Oil stream, upstream from Exchanger 10 on the Fuel Oil stream.
2. Rename the heat exchange **Exchanger 4**.
3. Click the **Tied** checkboxes for the Pre-Flash cold stream inlet and outlet temperatures.
4. In the hot stream inlet temperature field, enter **243°C**. This temperature come from the Process Flowsheet. The heat exchanger solves.
   The **Grid Diagram** tab and **Work Sheet** tab should now appear similar to the figures below.

Figure 1.36

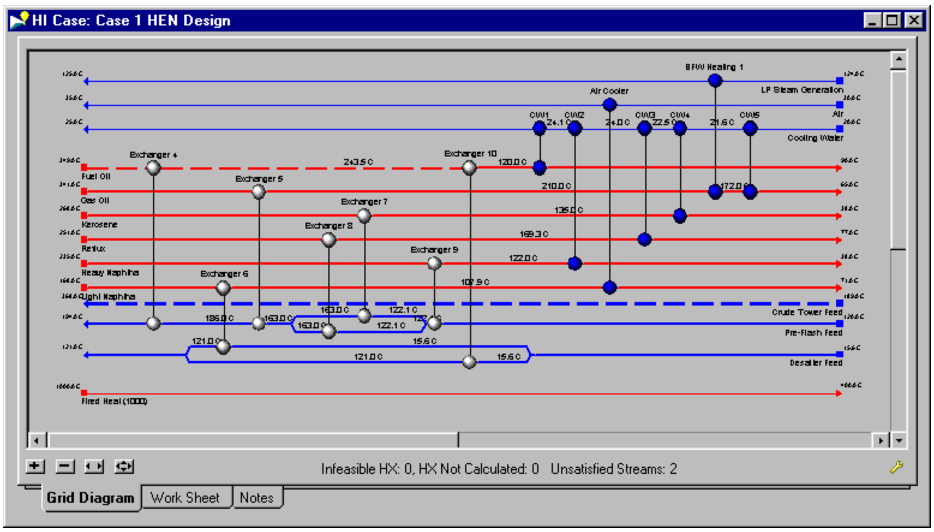
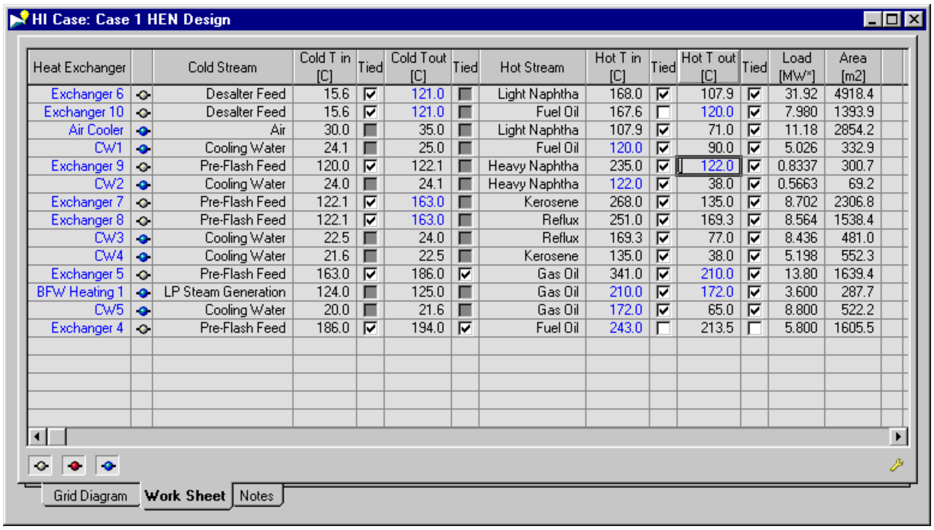


Figure 1.37



HI Case: Case 1 HEN Design

Heat Exchanger		Cold Stream	Cold T in [C]	Tied	Cold Tout [C]	Tied	Hot Stream	Hot T in [C]	Tied	Hot T out [C]	Tied	Load [MW*]	Area [m2]
Exchanger 6		Desalter Feed	15.6	☑	121.0		Light Naphtha	168.0	☑	107.9	☑	31.92	4918.4
Exchanger 10		Desalter Feed	15.6	☑	121.0		Fuel Oil	167.6	☐	120.0	☑	7.980	1393.9
Air Cooler		Air	30.0		35.0		Light Naphtha	107.9	☑	71.0	☑	11.18	2854.2
CW1		Cooling Water	24.1		25.0		Fuel Oil	120.0	☑	90.0	☑	5.026	332.9
Exchanger 9		Pre-Flash Feed	120.0	☑	122.1		Heavy Naphtha	235.0	☑	122.0	☑	0.8337	300.7
CW2		Cooling Water	24.0		24.1		Heavy Naphtha	122.0	☑	38.0	☑	0.5663	69.2
Exchanger 7		Pre-Flash Feed	122.1	☑	163.0		Kerosene	268.0	☑	135.0	☑	8.702	2306.8
Exchanger 8		Pre-Flash Feed	122.1	☑	163.0		Reflux	251.0	☑	169.3	☑	8.564	1538.4
CW3		Cooling Water	22.5		24.0		Reflux	169.3	☑	77.0	☑	8.436	481.0
CW4		Cooling Water	21.6		22.5		Kerosene	135.0	☑	38.0	☑	5.198	552.3
Exchanger 5		Pre-Flash Feed	163.0	☑	186.0	☑	Gas Oil	341.0	☑	210.0	☑	13.80	1639.4
BFW Heating 1		LP Steam Generation	124.0		125.0		Gas Oil	210.0	☑	172.0	☑	3.600	287.7
CW5		Cooling Water	20.0		21.6		Gas Oil	172.0	☑	65.0	☑	8.800	522.2
Exchanger 4		Pre-Flash Feed	186.0	☑	194.0	☑	Fuel Oil	243.0	☐	213.5	☐	5.800	1605.5

Grid Diagram | **Work Sheet** | Notes

1.4.7 Completing the Heat Exchanger Network

The rest of the heat exchanger network consists of heat exchangers to satisfy the Fuel Oil stream and two furnaces. The furnaces will be replaced with heaters on the Fired Heat (1000) stream.

1. Add a heat exchanger between the Fuel Oil stream and Crude Tower Feed stream.

Ensure that you place the hot end of the heat exchanger between the Fuel Oil stream inlet and Exchanger 4, not between Exchanger 4 and Exchanger 10.

2. Rename the heat exchanger **Exchanger 3**.
   The Fuel Oil stream should appear similar to the figure below.

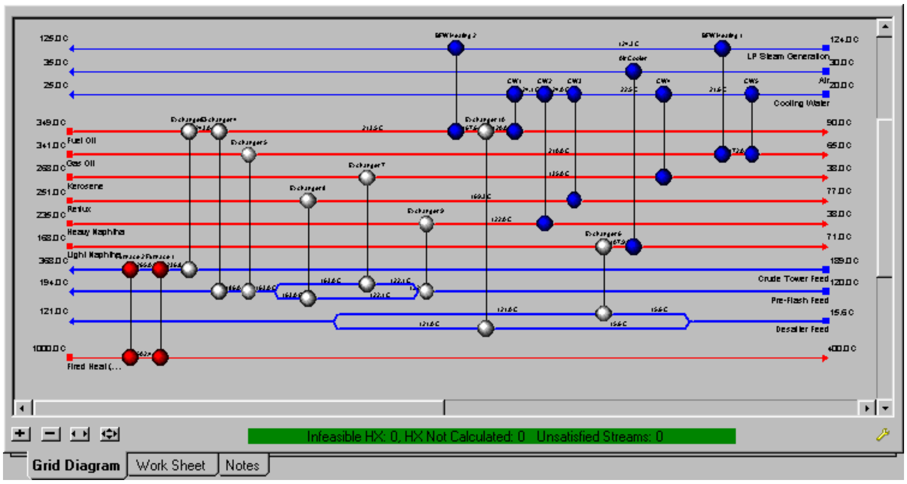
Figure 1.38

3. Click the **Tied** checkboxes for the hot stream inlet and outlet temperatures, and for the cold stream inlet temperature. The heat exchanger solves.
4. Add a heat exchanger between the Fuel Oil stream and the LP Steam Generation utility, between Exchanger 4 and Exchanger 10.
5. Rename the exchanger **BFW Heating 2**.
6. Click the **Tied** checkboxes for the hot stream inlet and outlet temperatures. The exchanger solves.
7. Add the first of the fired heaters by placing a heat exchanger between the Crude Tower Feed and Fired Heat (1000) utility, downstream of Exchanger 3.
8. Rename this exchanger **Furnace 1**.
9. Click the **Tied** checkbox by the cold stream inlet temperature.
10. In the cold stream outlet temperature field, enter **265°C**. The exchanger solves.
11. Add the final heat exchanger between the same streams as in step #4 above, downstream from Furnace 1.
12. Rename the exchanger **Furnace 2**.
13. Click the **Tied** checkbox for both cold stream temperatures. The exchanger solves and all streams are satisfied.

The heat exchanger network is complete. The status bar on the **Grid Diagram** tab will appear green, indicating that there are no unsatisfied streams, and no uncalculated heat exchangers.

Confirm your heat exchanger with the completed HEN diagram and worksheet as shown in the figures below.

Figure 1.39



Heat Exchanger	Cold Stream	Cold T in [C]	Tied	Cold Tout [C]	Tied	Hot Stream	Hot T in [C]	Tied	Hot T out [C]	Tied	Load [MW*]	Area [m2]
Exchanger 6	Desalter Feed	15.6	☑	121.0	☐	Light Naphtha	168.0	☑	107.9	☑	31.92	4918.4
Exchanger 10	Desalter Feed	15.6	☑	121.0	☐	Fuel Oil	167.6	☑	120.0	☑	7.980	1393.9
Air Cooler	Air	30.0	☐	35.0	☐	Light Naphtha	107.9	☑	71.0	☑	11.18	2854.2
CW1	Cooling Water	24.1	☐	25.0	☐	Fuel Oil	120.0	☑	90.0	☑	5.026	332.9
Exchanger 9	Pre-Flash Feed	120.0	☑	122.1	☐	Heavy Naphtha	235.0	☑	122.0	☑	0.8337	300.7
CW2	Cooling Water	24.0	☐	24.1	☐	Heavy Naphtha	122.0	☑	38.0	☑	0.5663	69.2
Exchanger 7	Pre-Flash Feed	122.1	☑	163.0	☐	Kerosene	268.0	☑	135.0	☑	8.702	2306.8
Exchanger 8	Pre-Flash Feed	122.1	☑	163.0	☐	Reflux	251.0	☑	169.3	☑	8.564	1538.4
CW3	Cooling Water	22.5	☐	24.0	☐	Reflux	169.3	☑	77.0	☑	8.436	481.0
CW4	Cooling Water	21.6	☐	22.5	☐	Kerosene	135.0	☑	38.0	☑	5.198	552.3
Exchanger 5	Pre-Flash Feed	163.0	☑	186.0	☑	Gas Oil	341.0	☑	210.0	☑	13.80	1639.4
BFW Heating 1	LP Steam Generation	124.0	☐	124.3	☐	Gas Oil	210.0	☑	172.0	☑	3.600	286.1
CW5	Cooling Water	20.0	☐	21.6	☐	Gas Oil	172.0	☑	65.0	☑	8.800	522.2
Exchanger 4	Pre-Flash Feed	186.0	☑	194.0	☑	Fuel Oil	243.0	☑	213.5	☑	5.800	1605.5
Exchanger 3	Crude Tower Feed	189.0	☑	236.8	☑	Fuel Oil	349.0	☑	243.0	☑	22.80	3398.0
BFW Heating 2	LP Steam Generation	124.3	☐	125.0	☐	Fuel Oil	213.5	☑	167.6	☑	8.194	670.3
Furnace 1	Crude Tower Feed	236.8	☑	265.0	☑	Fired Heat (1000)	502.4	☐	400.0	☐	14.00	1003.0
Furnace 2	Crude Tower Feed	265.0	☑	368.0	☑	Fired Heat (1000)	1000.0	☐	502.4	☐	68.00	2509.0

Grid Diagram | **Work Sheet** | Notes

When you examine all of the calculated values, you will notice that all values are very close to those indicated on the initial Process Flowsheet.

1.5 References

[1] Linnhoff, B., Townsend, D.W., Boland, D., Hewitt, G.F., Thomas, B.E.A., Guy, A.R., Marsland, R.H., A User Guide on Process Integration for the Efficient use of Energy, IChemE England, 1982.

2 Data Extraction from HYSYS

2.1 Introduction

One of the extremely useful features of HX-Net is its ability to extract information from HYSYS or Aspen Plus so that heat integration analysis can be performed on a pre-built simulation, without having to re-enter the information.

In this tutorial, you will examine one of the default HYSYS cases provided with HX-Net, and then extract the information into HX-Net.

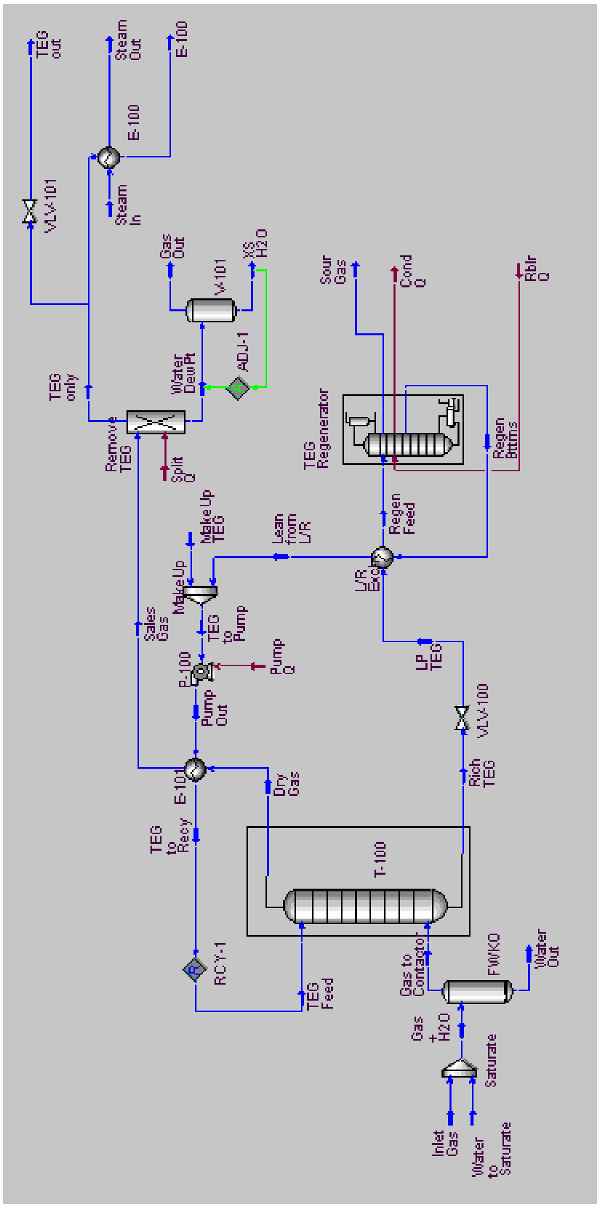
> **You must have HYSYS version 3.0 or higher installed on your computer in order to use this tutorial.**

This tutorial assumes that you have used HYSYS and understand how to navigate through HX-Net. It is also assumed that you have completed the HI Case or HI Project tutorial, and that you understand how to create streams and create a heat exchanger network using the Grid Diagram.

The HYSYS case that will be extracted is a modified version of the sample HYSYS case G-2.hsc. The file included with HX-Net is named **dataext.hsc** and can be found in the Samples\HYSYSTestCases directory.

The HYSYS flowsheet is shown in the figure below.

Figure 2.1



The natural gas industry commonly uses tri-ethylene glycol (TEG) for gas dehydration where low gas dew point temperatures are required, such as in the design of offshore platforms in the Arctic or North Sea regions or for other cryogenic processes.

The composition of the natural gas stream (Inlet Gas) has been provided on a water-free basis. To ensure water saturation, this stream is mixed with stream Water To Saturate. The water-saturated gas stream Gas + H2O is then fed to a scrubber to knock out the free water. This scrubbed stream (Gas To Contactor) is fed to the TEG Contactor, where it is contacted with a regenerated lean TEG stream (TEG Feed). Stream TEG Feed absorbs most of the water in the Gas To Contactor stream. The rich TEG stream from the absorber bottoms (Rich TEG) is heated to 220 °F by the hot lean TEG from the regenerator (Regen Bttms), and is fed to the stripper column for regeneration. The stripper column is a refluxed tower consisting of 3 stages plus a condenser. The regenerated TEG is cooled and returned to the top of the TEG absorber.

For the purposes of demonstrating HX-Net capabilities to extract HYSYS data, the TEG only stream is being heated at the same time that the pressure is greatly reduced. A multiple attachment has been used to display two options, a heat exchanger using steam or a valve.

A recycle operation is required to complete this simulation because the lean TEG is being returned to the contactor. An initial estimate of the lean TEG is required to run the contactor, and when the rest of the simulation has been built, the regenerator will calculate the new lean TEG. This stream is updated by the recycle operation.

2.2 Preparing for Data Extraction

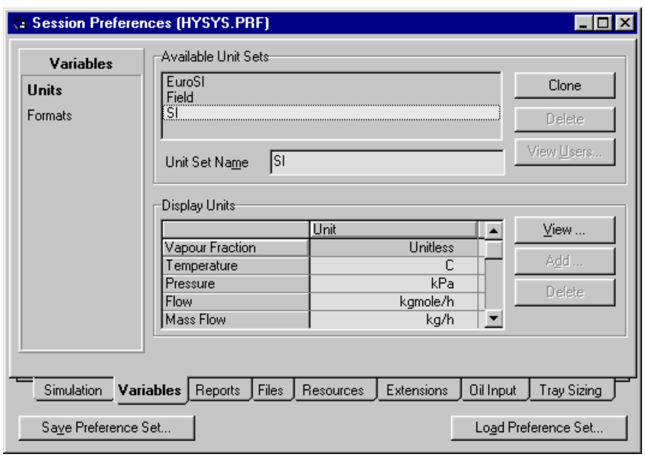
HYSYS or Aspen Plus data can be extracted into either HI Project or HI Case. Before performing a data extraction, it is always wise to read the Extraction Tips so that you can find as many possible problems in your simulation flowsheet before performing the first extraction.

2.2.1 Setting Unit Preferences

Before you begin, verify that the units currently selected in the preferences are the ones you want to use. HX-Net will perform the unit conversion calculations when it extracts the data from HYSYS, but it is easier to compare numbers if they use the same units.

1. Open HYSYS (version 3.0 or higher).
2. Open the case **dataext.hcs**, included in the \Samples\HYSYSTestCases directory of your HX-Net program.
3. In the HYSYS **Tools** menu, select **Preferences**. The Session Preferences view appears.
4. Click the **Variables** tab, then click the **Units** page. The **Unit Set Name** should be **SI**. If it isn't, select SI in the Available Unit Sets list.

Figure 2.2



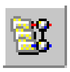
5. From the Windows **Start** menu, open HX-Net.

6. In HX-Net, repeat steps #3 and #4 to set the units to match the HYSYS case.
7. Save the modified case under a new name, such as **newDataEXT.hsc**. Do not save any changes over the original case.

Before you can examine the extraction tips, open the HI Case view or HI Project view.

2.2.2 Opening a HI Case or HI Project

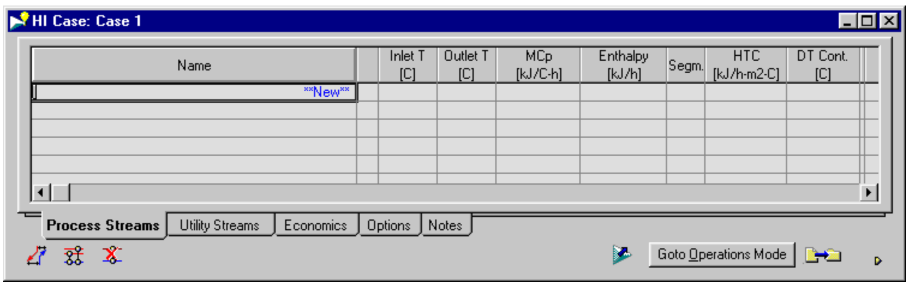
1. To access the HI Case view or HI Project view, do one of the following:
   - From the **Features** menu, select **HI Case** or **HI Project**.
   - From the **Managers** menu, select **Heat Integration Manager**. The Heat Integration Manager view appears. In the left column, select **HeatIntegrationCase** or **HeatIntegrationProject**, then click the **Add** button.
   - On the tool bar, click the **Heat Integration Manager** icon. The Heat Integration Manager view appears. In the left column, select **HeatIntegrationCase** or **HeatIntegrationProject**, then click the **Add** button.



Heat Integration Manager icon

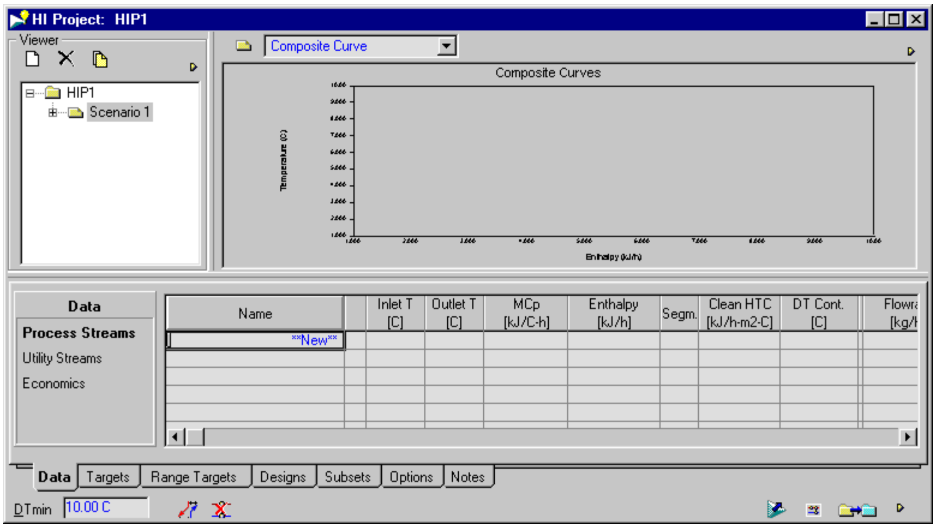
The HI Case view or HI Project view appears.

Figure 2.3



HI Case view

Figure 2.4



HI Project view

You are now ready to look at the extraction tips.

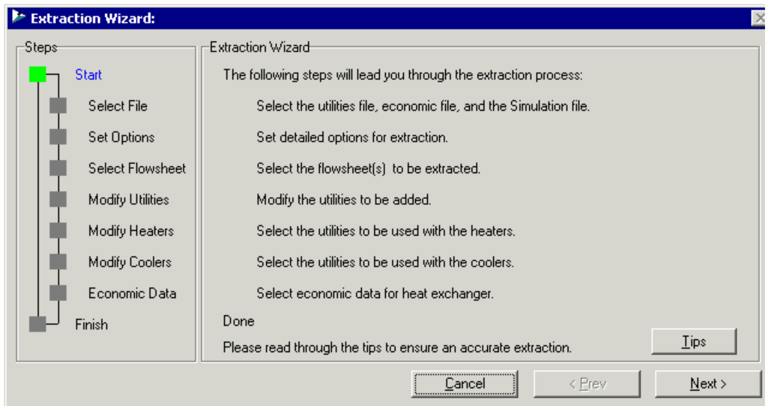
2.2.3 Examining the Extraction Tips

In the HI Case view, the Data Extraction icon appears only on the Process Streams tab. In the HI Project view, it appears on the Process Streams page of the Data tab.

Data Extraction icon

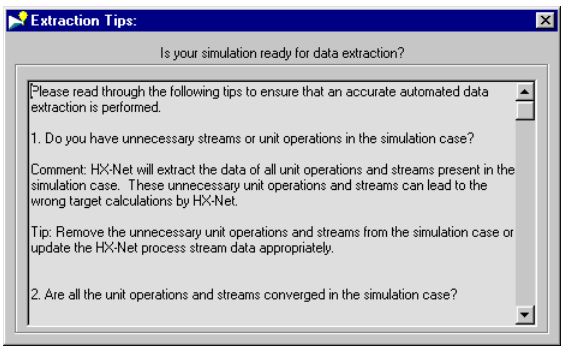
1. Click the **Data Extraction** icon. The Extraction Wizard appears.

Figure 2.5



2. Click the **Tips** button. The Extraction Tips view appears.

Figure 2.6



Read all tips carefully before continuing to the next section, as you will be manipulating the case based on these tips. It is good practice to read these tips before every extraction. This will allow you to find some if not all of the errors before you perform the data extraction.

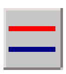
2.3 Editing the HYSYS Case

Before you extract information from HYSYS, you will use the tips you just reviewed to check the HYSYS case to be extracted to find possible problems. Although HX-Net will produce warnings about many of the issues in HYSYS, it will not produce messages about others that can result in incorrect targets, such as repeated names.

2.3.1 Checking Mode & Solved Status of Unit Operations

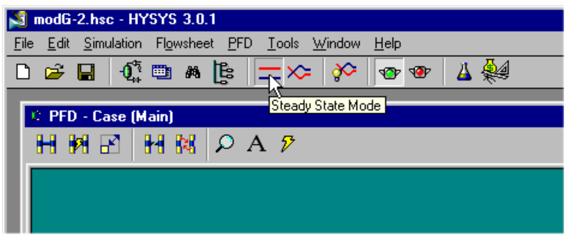
In order to extract a HYSYS case into HX-Net, the case must be in steady state mode (tip 3) and the entire flowsheet must be solved (tip 2). Since these conditions are very easy to check, and absolutely essential to the extraction, they will be checked first.

1. Ensure that the HYSYS case to be extracted is open and that you've set the unit preferences as described in steps #1 and #2 in **Section 2.2.1 - Setting Unit Preferences**.
2. On the HYSYS tool bar, ensure that the **Steady State Mode** icon is active, as shown in the figure below.



Steady State Mode icon

Figure 2.7

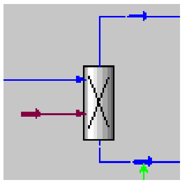


The sample case should be in steady state mode. If it is not, click the **Steady State Mode** icon, and ensure that the case converges.

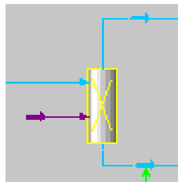
3. On the flowsheet, verify that all streams and unit operations have been solved by doing one or both of the following:

   - Examine all streams and unit operations. All material streams should appear dark blue, all energy streams should appear dark red, and all unit operations should be outlined in black.

Figure 2.8



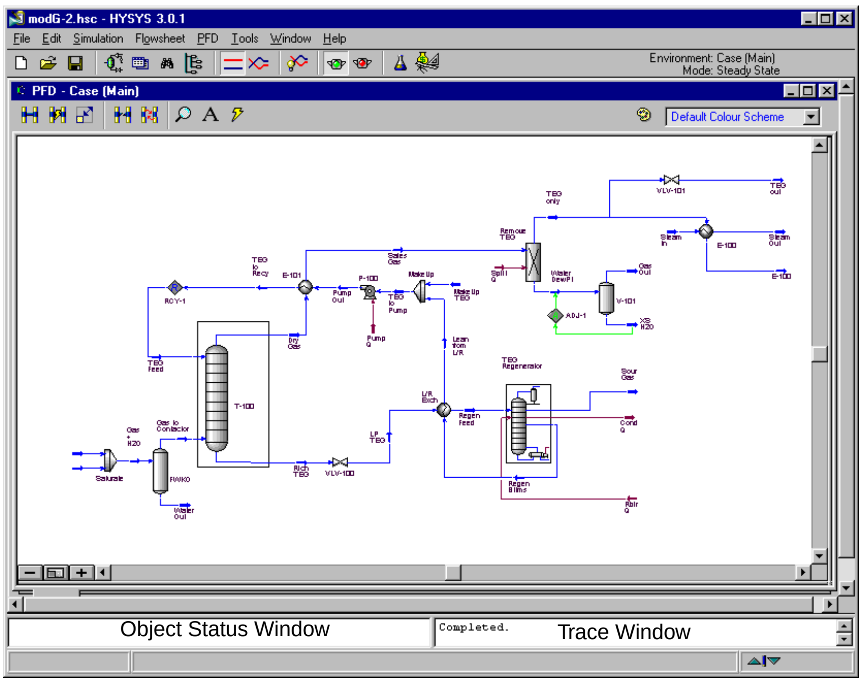
An example of a “solved” unit operation and attached streams



An example of an “unsolved” unit operation and attached streams

   - Check the Object Status Window at the bottom left corner of the window for error messages.

Figure 2.9



The sample case provided is solved. However, if the sample case is not solved, check the following:

- Check the Trace Window for the streams or operations that are missing information.
- If the Trace Window is empty, and the flowsheet is not solved, it could be because the solver is not active. Click the **Solver Active** icon.
- Check your HYSYS manuals for more information on how to solve flowsheets.
- Check that there are no extra unit operations or streams in the flowsheet (tip 1).

Solver Active icon

Remove the Valve and Extra Stream

In the sample case DataEXT. hsc, all of the operations and streams are directly related to the flowsheet (i.e., there are no streams and unit operations forming a second system). The stream TEG, however, only has a multiple attachment to a valve and a heat exchanger. Since multiple attachments pose problems for HX-Net, and the valve will not result in any changes to the network built in HX-Net, nor the targets calculated, remove the valve and extra stream.

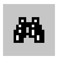
These items may already be removed from the sample case.

1. Disconnect stream **TEG only** from valve **VLV-101**.
2. Delete valve **VLV-101**and the material stream **TEG out**.

2.3.2 Checking Stream & Unit Operation Names

Checking for Duplicate Names

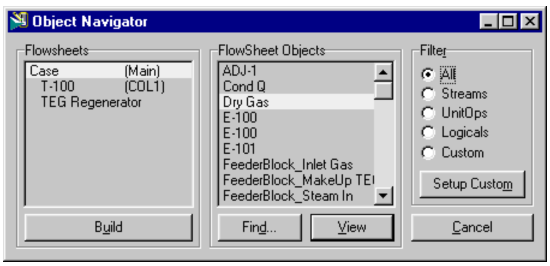
Multiple tips warn of extraction problems due to streams and unit operations having the same name, either within one flowsheet or across multiple flowsheets (tips 6, 8, 9,). The easiest way to check for this is by using the Object Navigator.



Object Navigator icon

1. Access the Object Navigator by doing one of the following:
   - On the toolbar, click the **Object Navigator** icon.
   - In the **Flowsheet** menu, select **Find Object**.
   - Press the **F3** hot key.
2. Search for unit operations and streams that have the same name. In the **Flowsheets** group, select the main flowsheet. In the **Filter** group, click the **All** radio button. This will display all unit operations and streams. The Object Navigator appears as shown in the figure below.

Figure 2.10



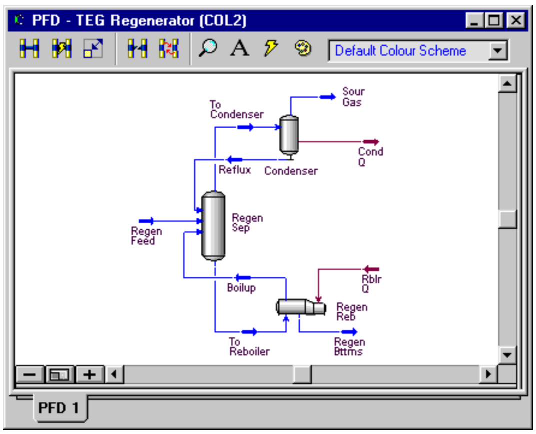
This may already be corrected in the sample case.

3. Scroll down the Flowsheet Objects list. There are two objects named E-100. If you look on the flowsheet, you will find that there is a heat exchanger and a stream with this name. Rename the stream to TEG out.
4. Using the Object Navigator, repeat step #2 to ensure that there are no other multiple names. In this case there are none. This takes care of tip 9.

Checking for Streams that Span Multiple Flowsheets

1. Use the Object Navigator to check for names that appear in more than one flowsheet.
   - All the inlet and outlet streams entering and exiting each column are subject to this rule.
   - Look for any stream that contains a heat exchanger in both the main flowsheet and the sub-flowsheet.
2. Examine the Regen Feed stream. In the main flowsheet, the Regen Feed stream exits a heat exchanger before entering the TEG Regenerator sub-flowsheet, which appears in the following figure. In this sub-flowsheet, however, there is no exchanger on the Regen Feed stream, so it will not pose a problem.

Figure 2.11



3. Open the TEG Regenerator sub-flowsheet. The streams Sour Gas and Regen Bttms exit the condenser and reboiler, respectively.
   - The Sour Gas stream does not have any exchangers on it on the main flowsheet so it will not pose any difficulties.
   - The Regen Bttms stream, however, enters a heat exchanger in the main flowsheet. HX-Net uses the stream entering the reboiler and returning to the column during extraction, so there will be no stream duplication in this case.
4. Return to the main flowsheet. The **Dry Gas** stream exiting the T-100 column is entering a heat exchanger.
5. Open the T-100 column sub-flowsheet. There is no reboiler or condenser on this column, as it is acting as an absorber, therefore, this will not create a stream duplication.

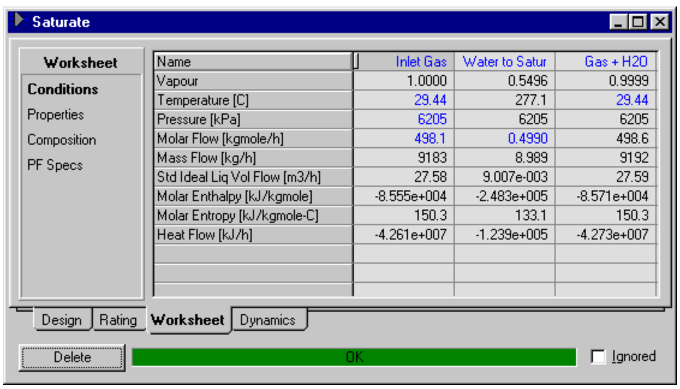
6. Use the Object Navigator to see if there are any internal streams existing within any sub-flowsheet that do not represent a real stream.
   - To do this, highlight each sub-flowsheet in the **Flowsheets** group of the Object Navigator, then select the **Stream** radio button in the **Filter** group.
   - This will display all material streams existing in the flowsheet. All internal streams will be extracted as real streams.
   - In this case, all of the internal streams do represent real streams in the process. Since the reboiler and condenser must be taken into account, no changes are required.
   - If there are streams that you do not want extracted, you will have to manually delete them after the data extraction.

2.3.3 Checking Mixers & Splitters

The last items to check before performing the data extraction in HYSYS are mixers and splitters. Check for non-isothermal mixers (tip 10). In this case, there are no multiple splitters or mixers in series, however, there are two mixers that should be checked to see if they are isothermal.

1. Open the property view for the **Saturate** mixer, and click on the **Worksheet** tab.

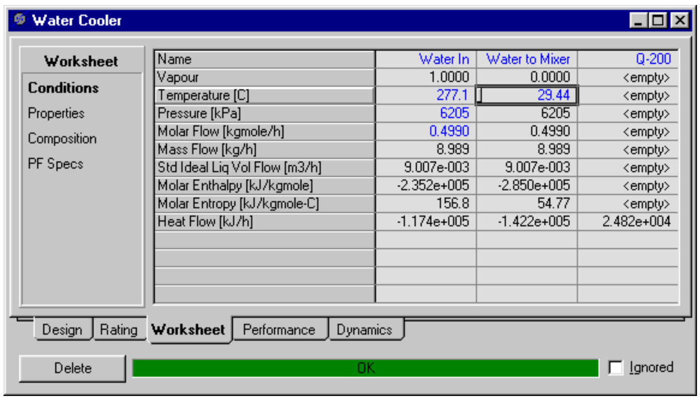
Figure 2.12



The temperatures of the two inlet streams, Inlet Gas and Water to Saturate, are not the same temperature, so the Water to Saturate stream should be cooled down to the temperature of the Inlet Gas and the mixer outlet stream Gas + H2O, before it enters the mixer.

2. Create a new material stream named Water In. Copy all of the information from the Water to Saturate stream by using the **Define from Other Stream** button.
3. Add a cooler named Water Cooler. Its inlet stream is the Water In stream you just created, and its outlet stream will be a new stream named Water to Mixer. Define an energy stream named Q-200.
4. On the **Design** tab of the cooler property view, click on the **Parameters** page, and define a pressure drop of 0 kPa.
5. Click on the **Worksheet** tab. In the Water to Mixer Temperature field, enter 29.44°C to make it the same as the mixer inlet stream temperature. The Water Cooler property view appears as shown in the figure below.

Figure 2.13



Name	Water In	Water to Mixer	Q-200
Vapour	1.0000	0.0000	<empty>
Temperature [C]	277.1	29.44	<empty>
Pressure [kPa]	6205	6205	<empty>
Molar Flow [kgmole/h]	0.4990	0.4990	<empty>
Mass Flow [kg/h]	8.989	8.989	<empty>
Std Ideal Liq Vol Flow [m3/h]	9.007e-003	9.007e-003	<empty>
Molar Enthalpy [kJ/kgmole]	-2.352e+005	-2.850e+005	<empty>
Molar Entropy [kJ/kgmole-C]	156.8	54.77	<empty>
Heat Flow [kJ/h]	-1.174e+005	-1.422e+005	2.482e+004

6. Close the Water Cooler property view.
7. Open the Saturate mixer property view.
8. Click the **Design** tab, then click the **Connections** page.

9. Click in the inlets stream cell **Water to Saturate**, and from the drop-down list, select the Water to Mixer stream. Also, define a new outlet stream named Mixer Outlet. This should be done because, even if the inlet temperatures are identical, the outlet temperature can be different than the original outlet stream due to phase changes.

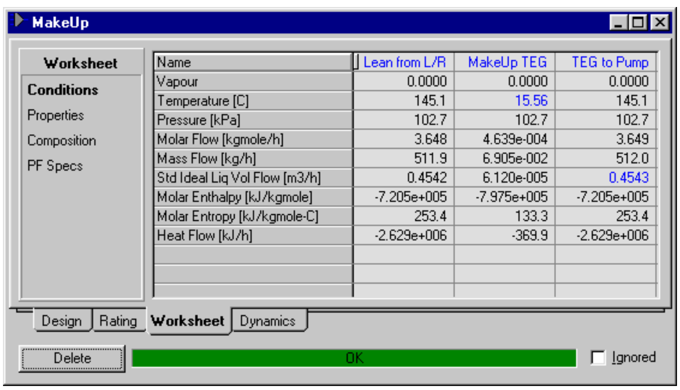
> **The first part of the flowsheet will now calculate. The rest of the flowsheet will not be calculated.**

10. As per (tip 1), delete the now unnecessary Water to Saturate stream.
11. Move the cursor over the Mixer Outlet stream. The fly-by for the Mixer Outlet stream indicates that the calculated temperature is 28.69°C. This is less than the original outlet temperature of the Gas + H2O stream, which is 29.44°C. Therefore, a heater is required between these two streams.
12. Add a heater with the name Gas + H2O Heater, and set the inlet stream as Mixer Outlet, outlet stream as Gas+H2O, and an energy stream as Q-201.
13. On the **Design** tab, click on the **Parameters** page, and define a pressure drop of 0 kPa.
14. The entire flowsheet should now recalculate.
15. Close all property views.

You will now check the second mixer to see if it is non-isothermal.

16. Open the property view for the MakeUp mixer, and click on the **Worksheet** tab.

Figure 2.14



In this case, the stream MakeUp TEG must be heated up to the mixer outlet temperature.

17. Using the same procedure as step #2, create a stream named Cool MakeUp TEG based on the information in the stream MakeUp TEG.
18. Add a heater named Heat MakeUp TEG. Set the inlet stream as Cool MakeUp TEG and define the new outlet stream as MakeUp TEG to Mixer. Define a new energy stream as Q-202.
19. On the **Design** tab, click on the **Parameters** page, then define a pressure drop of 0 kPa.
20. On the **Worksheet** tab, enter the MakeUp TEG to Mixer temperature as 145.07°C. This is the same as the other mixer inlet stream.
21. Close the Heat MakeUp TEG heater property view, and re-open the MakeUp mixer property view (if not already open).
22. On the **Design** tab, click on the **Connections** page. Replace the outlet stream with a new stream TEG from Mixer, and replace the inlet stream MakeUp TEG with the stream MakeUp TEG to Mixer.
23. Delete the now unnecessary stream MakeUp TEG.
24. Close all property views.
25. Move the cursor over the TEG from Mixer stream. The fly-by for the TEG from Mixer stream indicates that the temperature calculated is 145.1°C. This is the same as the two inlet temperatures. So, you do not need to heat or cool this stream.
26. Open the property view for the pump P-100. On the **Connections** page of the **Designs** tab, replace the inlet stream TEG to Pump with the stream TEG from Mixer. Delete the stream TEG to Pump. The entire flowsheet should solve at this point.
27. If you want, you can manipulate the PFD to make it appear neater.
28. Save the modified HYSYS case (renamed in step #7 of **Section 2.2.1 - Setting Unit Preferences**). Do not save over the original case.
29. Close HYSYS.

This is all of the work that will be performed on this particular case before performing the first extraction. It is important to remember that in other cases, you will also have to deal with LNG exchangers. It is also important to remember that the extraction cannot perform perfectly on the first attempt. After the extraction is complete, the warning section will indicate that more changes are required to our HYSYS flowsheet.

2.4 Performing the Data Extraction

2.4.1 Performing the Initial Data Extraction

1. If you have closed HX-Net, re-open it at this time.
2. Open a new **HI Project**. Data extraction can also be performed in HI Case, but for this tutorial you will use HI Project.

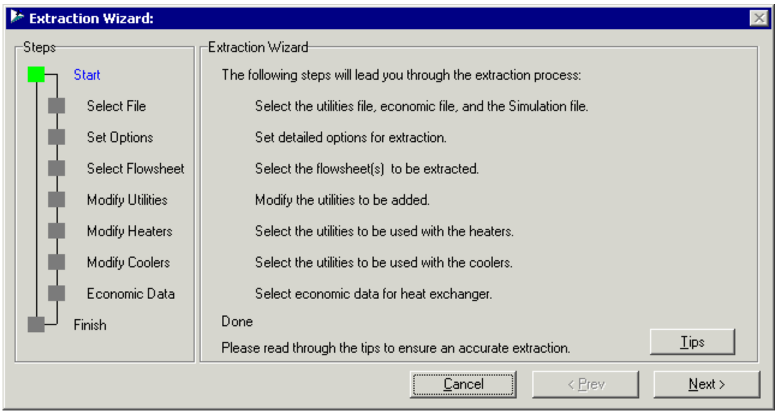
Data Extraction icon

The Extraction Wizard takes you through the following steps for the extraction process:

- Select File
- Set Options
- Select Flowsheet
- Modify Utilities
- Modify Heaters
- Modify Coolers
- Economic Data

3. Click the **Data Extraction** icon. The Extraction Wizard appears.

Figure 2.15



4. Click the **Next** button. The next page appears, where you can select the utilities file, economic file, and the simulation file.

Select File (Step 1 of 7)

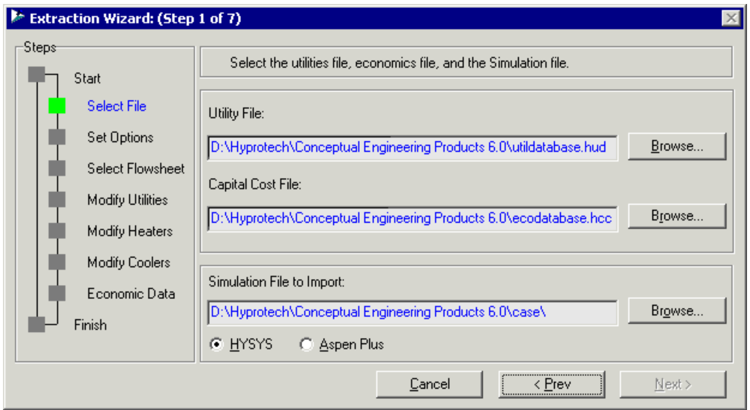
The data extraction process requires utility and economic information in order to perform the extraction and the costing target calculations.

HX-Net automatically selects the default utility and capital cost default files, but you can change the selection if required.

You can click the **Browse** button to select different files for the utility and capital cost data.

1. For the Utility File and Capital Cost File, accept the default selection.

Figure 2.16



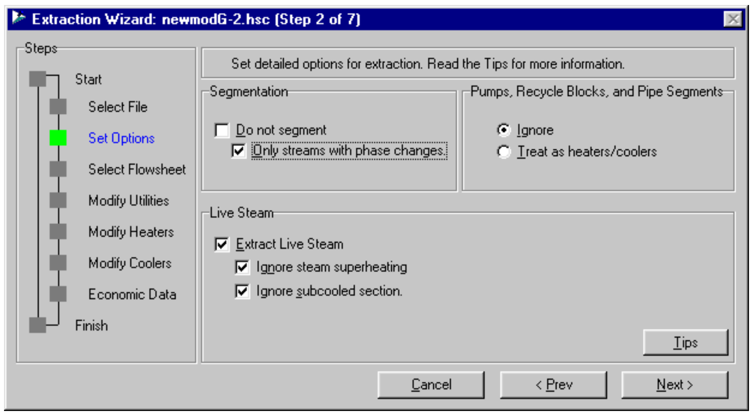
2. In the bottom group, select the **HYSYS** radio button.
3. Click the **Browse** button and locate the file you saved in step #28 in Section 2.3.3 - Checking Mixers & Splitters.
4. Click the **Next** button to set detailed options for extraction.

Set Options (Step 2 of 7)

You can select different options that will affect the way in which the data is extracted.

1. Ensure that the following options are set:
   - The **Only streams with phase changes** checkbox is checked.
   - All checkboxes in the **Live Steam** group are checked.
   - The **Ignore** radio button is selected in the **Pumps, Recycle Blocks, and Pipe Segments** group.

Figure 2.17



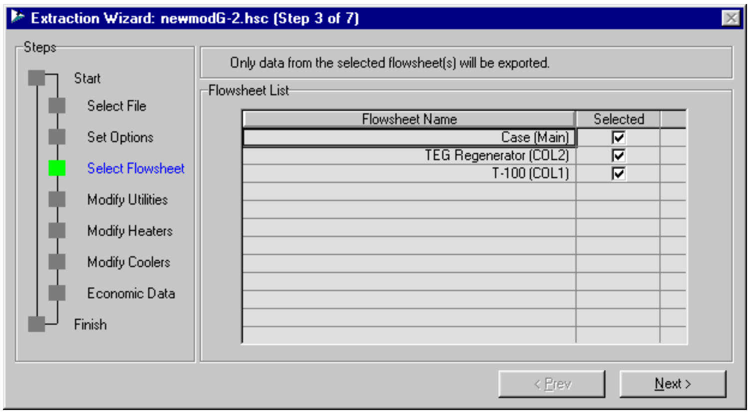
2. Click the **Next** button to select the flowsheet(s) to be extracted.

Select Flowsheet (Step 3 of 7)

The data from the selected flowsheet(s) is exported. When you click the **Next** button after setting the options, HX-Net starts HYSYS running and will extract the data.

1. Wait until the Extraction Wizard (Step 3 of 7) view appears as shown in the figure below.

Figure 2.18

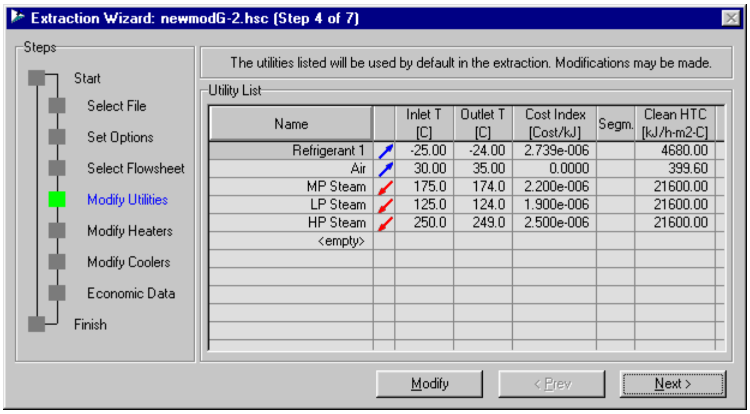


2. In this tutorial, you will be extracting data from all three different flowsheet. So make sure all the checkboxes under the **Selected** column are checked.
3. Click the **Next** button to see the utilities to be added.

Modify Utilities (Step 4 of 7)

The utilities listed on this page are used in the extraction.

Figure 2.19



1. If you want to modify the utilities to be added, click the **Modify** button.

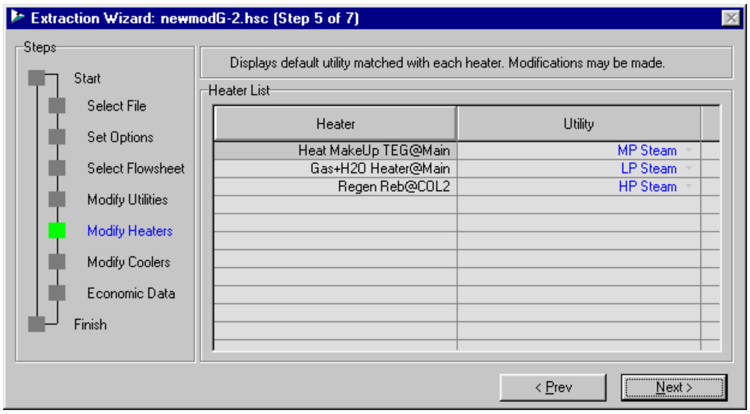
> **When you click the Modify button, the text changes to blue colour. The blue colour indicates you are allowed to change the utilities. The Modify button is replaced by the Lock button. When you click the Lock button, the text changes to black and it cannot be modified.**

2. You can also add more utilities by clicking the down arrow in the **<empty>** cell and selecting the new utility from the drop-down list.
3. In this tutorial, the default utilities selected by HX-Net are sufficient and do not require any modification.
4. Click the **Next** button to see the utilities to be used with the heaters.

Modify Heaters (Step 5 of 7)

On this page, you can modify the default utility matched with each heater.

Figure 2.20

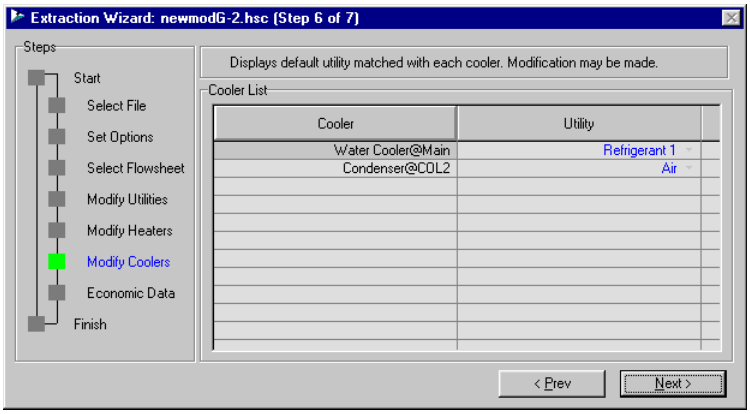


1. If you want to modify the default utility matched with each heater, click on the cell under the **Utility** column.
2. Open the drop-down list in the cell and select the utility you want.
3. In this tutorial, the default utilities selected for the heaters are sufficient and do not require any modification.
4. Click the **Next** button to see the utilities to be used with the coolers.

Modify Coolers (Step 6 of 7)

On this page, you can modify the default utility matched with each cooler.

Figure 2.21



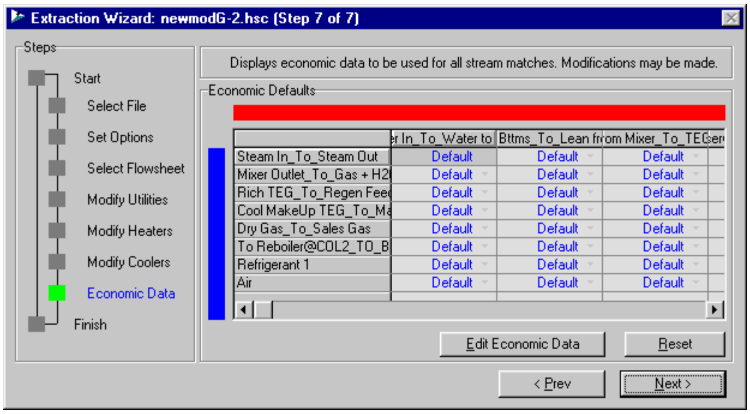
1. If you want to modify the default utility matched with each cooler, click on the cell under the **Utility** column.
2. Open the drop-down list in the cell and select the utility you want.
3. In this tutorial, the default utilities selected for the coolers are sufficient and do not require any modification.
4. Click the **Next** button to see the economic data for the heat exchangers.

Economic Data (Step 7 of 7)

The Reset button allows you to reset to the default values.

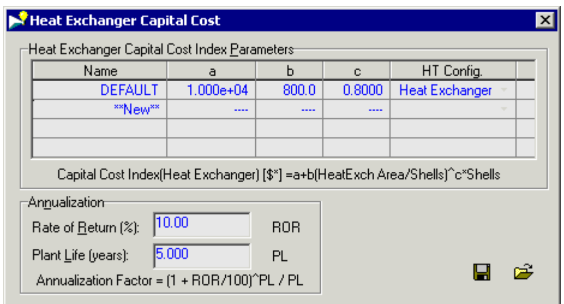
On this page, you can see and modify the type of economic data to be used for all stream matches in the heat exchanger network.

Figure 2.22



1. If you want to edit the economic data for the heat exchanger, click the **Edit Economic Data** button. The Heat Exchanger Capital Cost view appears.

Figure 2.23



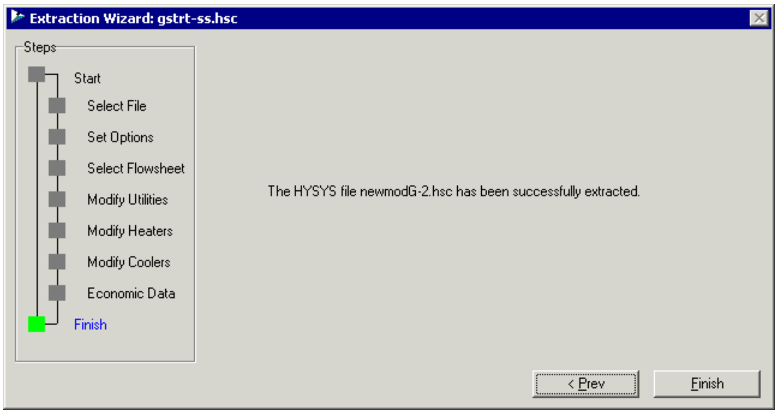
2. In this tutorial, the default economic data the heat exchangers are sufficient and do not require any modification.

Close icon

3. Click the **Close** icon to close the Heat Exchanger Capital Cost view, and return to the Extraction Wizard (Step 7 of 7).

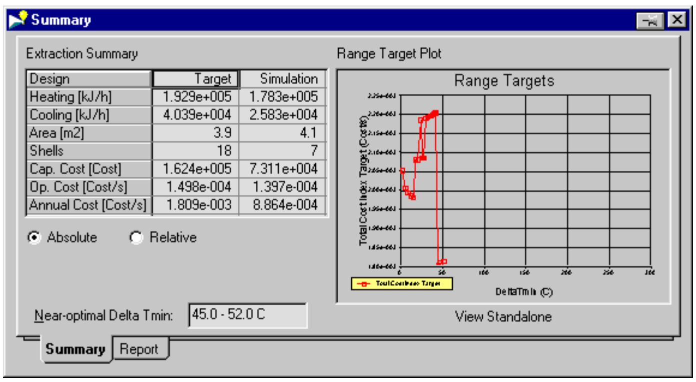
4. Click the **Next** button. A message appears indicating that the HYSYS file was extracted successfully.

Figure 2.24



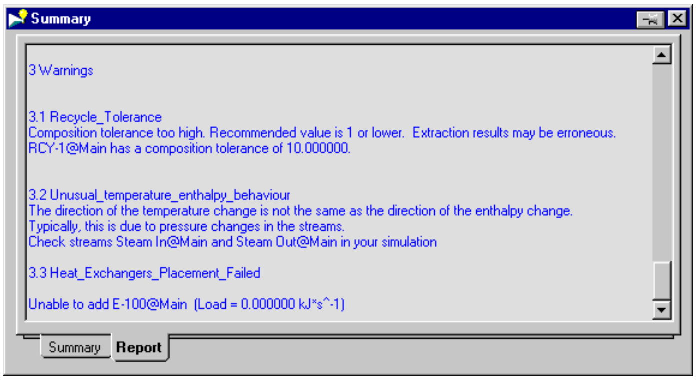
5. Click the **Finish** button, and the Summary view appears as shown in the figure below.

Figure 2.25



6. Click on the **Report** tab, and read the report carefully.
    - The first section of the report displays the new streams that have been created in HX-Net and the corresponding streams in the HYSYS case.
    - The second section displays the utility streams that have been selected from the list of default utilities in order to satisfy the heat load on the heaters and coolers.
    - The third section displays warnings of any potential problems that were found in the HYSYS case, and any heat exchangers that could not be placed properly. The warning section should have the warnings as shown in the figure below.

Figure 2.26



7. Close the Summary view.

You can review the Summary view at any time in HI Project by clicking the Data Extraction Report button on the Notes tab when you select the Scenario level in the Viewer group.

Since there are still some problems in the HYSYS case, you will return to it and fix the problems. The data will have to be extracted again after the new modifications to the HYSYS flowsheet, so there is not much sense in examining the data extracted at this point.

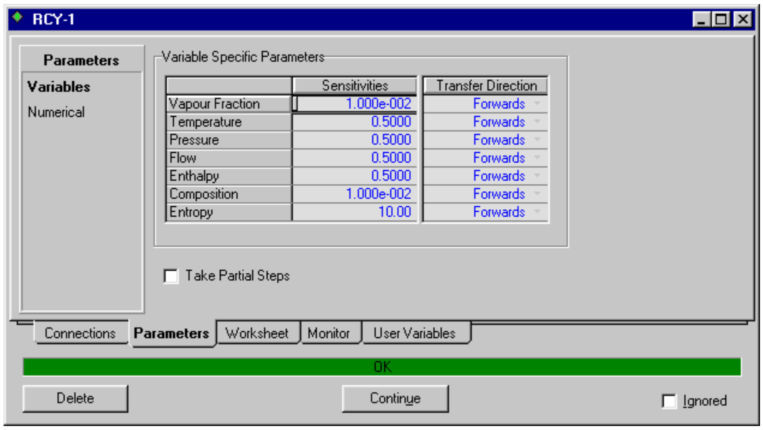
2.4.2 Fixing Warnings in the HYSYS Case

The HYSYS case should be open because HX-Net automatically opens it during the data extraction process.

1. The first warning is about the tolerances on the **Recycle** operation. In HYSYS, open the property view for this operation. Click the **Parameters** tab, then click the **Variables** page.
2. Reduce the Composition tolerance.
   - The Composition tolerance is 10, as indicated in the warning. The composition tolerance should always be 1.0 or less, however, altering the tolerances can result in an unsolved flowsheet.
   - Begin by reducing the tolerance to 1.0. The flowsheet should still solve.
   - Continue to reduce the tolerance by 0.1, ensuring that the flowsheet continues to solve. You should be able to reach a tolerance of 0.1.
3. Reduce other tolerance values.
   - Some of the other tolerances are quite high. Although this will not affect the data extraction process, it can improve the results within the HYSYS case.
   - Reduce the tolerances as indicated in the table below. The flowsheet should still solve as shown in **Figure 2.27**.

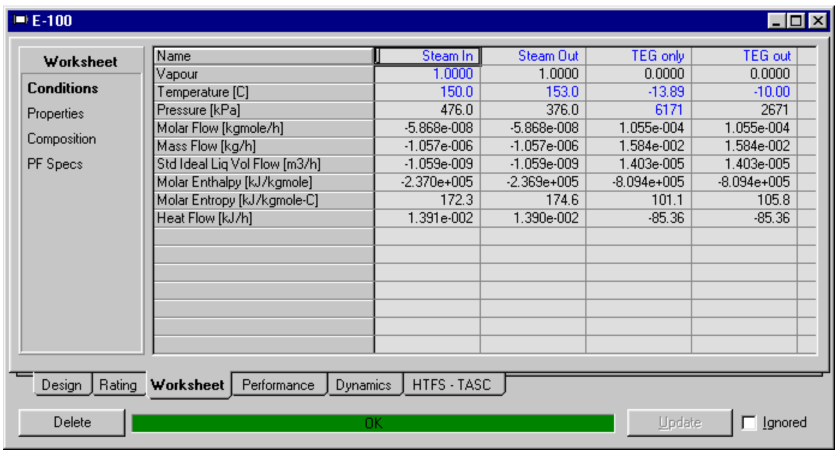
Variable	Tolerance
Vapour Fraction	0.01
Temperature	0.5
Pressure	0.5
Flow	0.5
Enthalpy	0.5
Composition	0.01

Figure 2.27



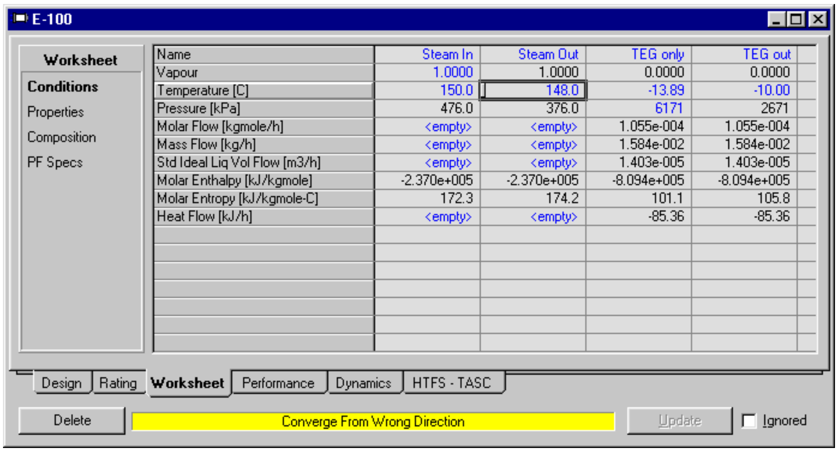
4. Close the Recycle property view.
   The second warning was about a temperature enthalpy reversal. The streams listed are around the heat exchanger E-100.
5. Open the property view for heat exchanger E-100.
6. Click the **Worksheet** tab. The Steam In temperature is lower than the Steam Out temperature. This does not make sense, since the steam is acting as the hot stream and should be cooling down in temperature.

Figure 2.28



7. To solve the problem, set the Steam Out temperature lower than the Steam In temperature. In the **Temperature** cell of the **Steam Out** column, enter 148°C.
   The Status bar in the E-100 property view turns yellow and indicates a Converge From Wrong Direction problem.

Figure 2.29



8. In the **Temperature** cell of the **TEG Out** column, enter -8°C. This should solve the converge from wrong direction problem.
9. The entire flowsheet should solve.
10. Close the E-100 property view.
11. Save the modified case and rename it **newDataEXT.hsc**.

Although this should now take care of the warnings listed in the Data Extraction report, there is one more potential problem.

Steam is used in the heat exchanger E-100. Even though this is a utility stream it will be extracted as a process stream. There are two options in this case. The first is to make E-100 a heater, which will automatically cause HX-Net to select a utility for it. The second is to manually replace the heat exchanger in HYSYS with a heater.

For now, leave the HYSYS case as it is and you will manipulate the data in HX-Net after the next data extraction.

You are now ready to extract the information from HYSYS again.

2.4.3 Performing the Final Data Extraction

You do not have to close the HYSYS case to perform the data extraction. Leave HYSYS open and return to HX-Net.

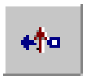
1. Create a new **HI Project**. Re-extracting the data into an existing project will generate inaccurate results.
2. Perform the steps in **Section 2.4.1 - Performing the Initial Data Extraction**. When the extraction Summary view appears again, read the report carefully. It should no longer display any warnings.

2.5 Adjusting the Extracted Data in HX-Net

Now that all of the HYSYS information has been extracted into HX-Net, you can examine the newly create streams heat exchanger network.

1. In HI Project, click on the **Scenario** level in the **Viewer** group.
2. Click the **Data** tab, then click the **Process Streams** page. Eleven streams have been extracted, however, one stream will be manually deleted as it is actually a utility stream, as noted after step #11 in **Section 2.4.2 - Fixing Warnings in the HYSYS Case**.
3. Before deleting the Steam stream, examine the Grid Diagram to see if there are any exchangers on this stream.
4. In the Viewer group of the main HI Project view, click on the **SimulationBaseCase** design.
5. On the Grid Diagram, locate the stream Steam_In_to_Steam_Out. There is one heat exchanger on this stream that is matched with the stream TEG_only_to_TEG_out. When the Steam stream is deleted, this stream will become unsatisfied.
6. In the Viewer Group, click the Scenario level and return to the **Process Streams** page on the **Data** tab.
7. Select the stream Steam_In_to_Steam_Out, then press the **DELETE** key on the keyboard.
8. Click the **Utility Streams** page. There are utilities that have been added during the extraction process, including three steam utilities, so a utility will not have to be added to take into account the stream that was deleted.

Open Palette View icon



Open Property Preset View icon

9. Return to the **SimulationBaseCase** design.
10. Click the **Open Palette View** icon in the lower right-hand corner of the Grid Diagram tab. The Design Tools palette appears.
11. Click the **Open Property Preset View** icon. The Property Presets view appears.
12. Select **Preset 6: Alphabetical**, which orders the streams alphabetically and shows the utility streams. Close the Property Presets view.
13. Add a heat exchanger between the TEG_only_to_TEG_out stream and the utility stream LP steam.
14. Open the heat exchanger property view.
15. Click the **Tied** checkbox for both of the cold process stream temperatures. The heat exchanger will solve. The network status bar turns green, and all streams and heat exchangers are completely satisfied and solved.

The data extraction process is now complete. All required adjustments to the HYSYS flowsheet were performed, and no warnings appeared in the extraction summary report. Changes were made to the process stream data in HX-Net to account for duplicate streams, and the heat exchanger network was completed by adding the final heat exchanger. You can now be confident that the targets calculated by HX-Net are accurate, and can continue on in the analysis of the existing network, or can perform changes to improve the network.

3 Automatic HEN Design in HI Project

3.1 Introduction

This tutorial serves two functions:

- Introduces the HX-Net Heat Integration (HI) Project environment.
- Demonstrates how to use the Recommend Designs feature to automatically generate Heat Exchange Network (HEN) designs.

If you are a new user to HX-Net, it is highly recommended that you complete the Crude Pre-Heat Train Network tutorial (refer to **Chapter 1 - Crude Pre-Heat Train Network**) before starting this tutorial.

It is assumed that you know how to add and complete heat exchangers on the Grid Diagram tab.

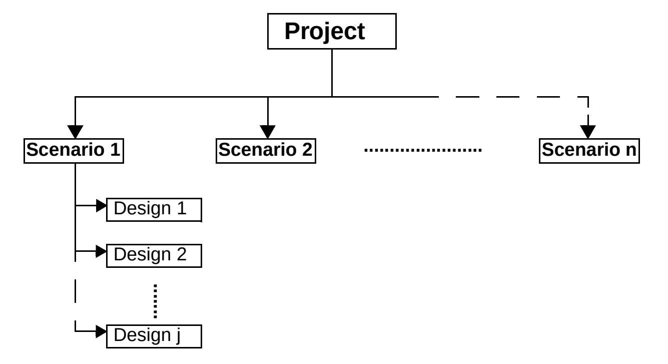
In this tutorial, you will create an HX-Net HI Project, enter stream and utility information, then use the HX-Net Recommend Designs feature to automatically generate heat exchanger network designs.

To demonstrate HX-Net's ability to optimize HEN designs, you will build a very simple network that will be far above the target values, then use the Recommend Designs feature to optimize the network design.

HX-Net provides you with a self-contained environment where you can create a HI Project with multiple Scenarios and Designs.

At the Project level, you define what you want to design. Within each Project, there can be numerous Scenarios and Designs, as shown in the figure below.

Figure 3.1



The Project level contains only the most general description of the problem set being examined.

The Scenario level contains the assumptions, conditions, and information required to generate a design. These conditions include process stream specifications, utility streams, and economic factors.

The Design level contains the generated HEN design solutions. A Scenario can contain multiple designs. The generated designs are determined by the conditions, assumptions and specifications defined at the Scenario level.

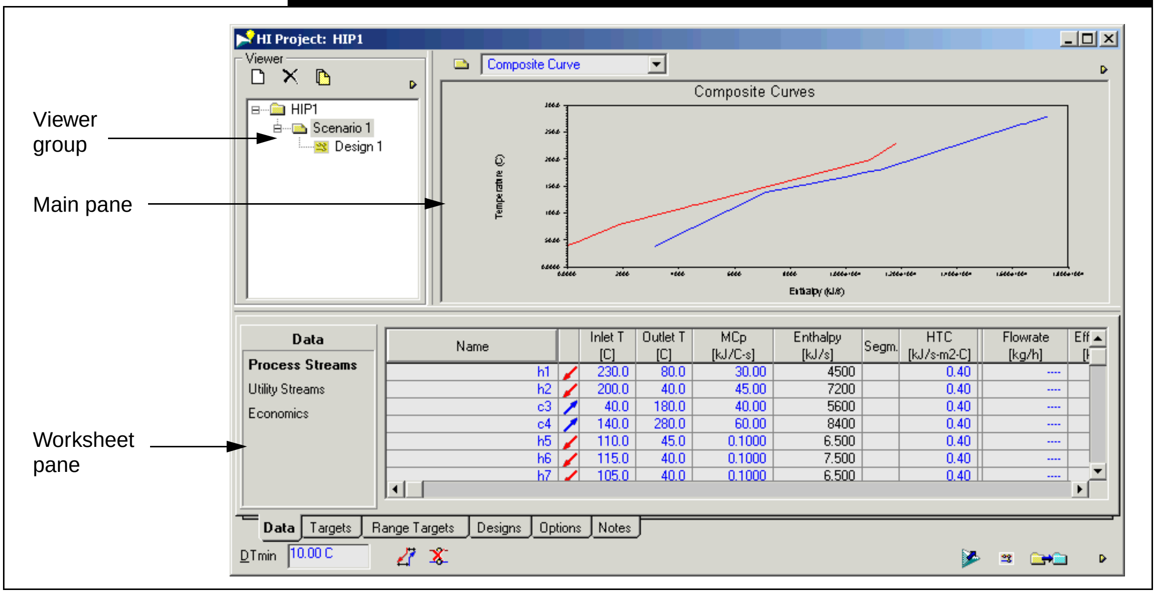
3.1.1 Navigating Through HI Project

The Heat Integration Project View

The Heat Integration Project view in HX-Net is divided into three sections: the Viewer group, the Main pane, and the Worksheet pane.

The three sections are displayed and labeled in the figure below.

Figure 3.2

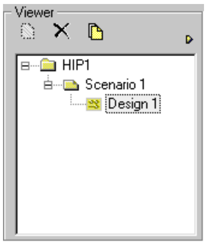


Viewer Group

The Viewer group is always visible in the Heat Integration (HI) Project view and contains the HX-Net tree browser, which is used to access, create and delete Scenario and Design levels within a HI Project.

To expand or compress the tree, click on the + or - beside the level you want to view.

Figure 3.3



When you create a new HI Project, HX-Net automatically creates a Scenario and Design level. You cannot delete the default Scenario and Design levels.

Main Pane

At the Scenario level, the Main pane displays a plot. You can select the type of plot to display from a drop-down list at the top of the view.

At the Design level, the Main view displays the Heat Exchanger Network (HEN) diagram.

Worksheet Pane

The Worksheet pane of the HI Project view displays the entered and calculated values, both at the Scenario and Design levels.

The following sections describes some of the commonly used tabs available at the Scenario and Design levels of the HI Project view.

Scenario Level

The following table lists and describes the tabs found at the Scenario level.

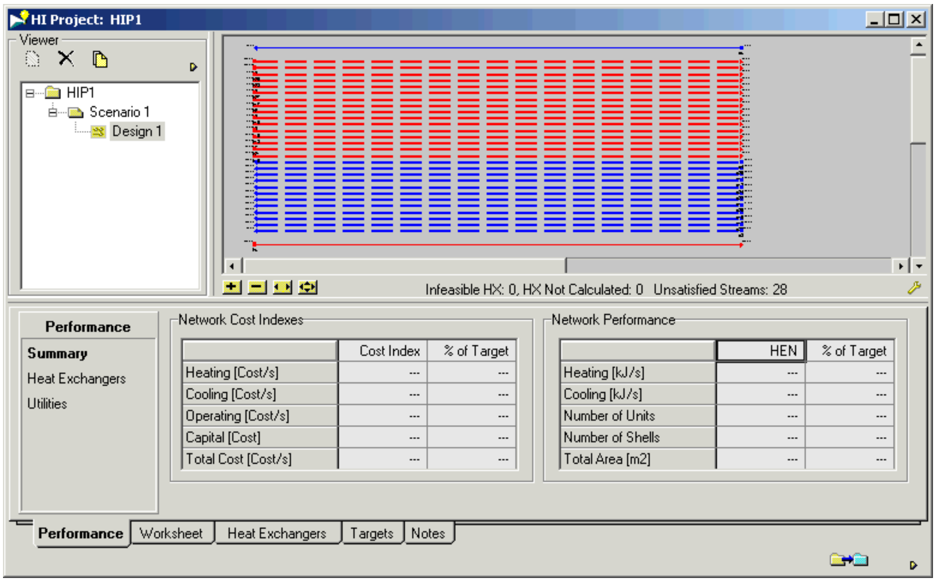
Tab	Description
Data tab	On this tab, you enter all process and utility stream data and set the cost parameters.
Targets tab	All three pages on this tab contain the target values calculated by HX-Net. These values represent the performance of an ideal heat exchanger network design for the entered stream and economic data.
Range Targets tab	The options on this tab are useful for determining the optimal minimum approach temperature, or DTmin, value. Click the **Calculate** button, and use the plot or the table to find the DTmin value for a minimum area or minimum cost value. For more information, refer to **Section 1.3.1 - Range Targeting**.
Designs tab	Use this tab to compare all designs within a Scenario. Here, you can display all designs or display only completed designs. You can also display the designs as a percentage related to the target values.
Options tab	Use this tab to manipulate the utility load allocation method and access the utility and HTC databases.
Notes tab	Use this tab to enter notes for the Scenario level.

Design Level

To view the Design level, expand the tree in the **Viewer** group. To do this, click the + beside the Scenario folder. Any designs contained within the Scenario appear in the tree.

To view a design, click the design name. The Main pane now displays a Grid Diagram instead of a plot.

Figure 3.4



The following table lists and describes the tabs found at the Design level.

Tab	Description
Performance tab	This tab displays the performance information for all heat exchangers and utilities in the design.
Worksheets tab	As in HI Case, this tab provides an alternative way to manipulate the heat exchangers on the Grid Diagram.
Heat Exchangers tab	This tab displays detailed information about each heat exchanger. When the Show All checkbox is checked, this tab shows all heat exchangers. When the checkbox is unchecked, only solved exchangers appear.
Targets tab	This tab shows all the same targets information available at the Scenario level.
Notes tab	This tab displays notes for a particular design. This tab also contains a Modification Log page, which automatically records and displays all modifications made to the Grid Diagram.

Now that you have an understanding of the setup and structure of the HI Project views, you are ready to begin the tutorial.

3.2 Creating a HI Project for Automatic Design Generation

To use HX-Net's Recommend Design feature, the HI Project must contain all required process stream data and utility streams that are sufficient to satisfy the energy demands of the process streams.

In the following sections you will set unit preferences, create the HI Project, and enter process and utility stream data.

3.2.1 Setting Unit Preferences

In this section, you will define a new unit set. For this tutorial, the temperatures are in Celsius, and the MCp is in kJ/C-s.

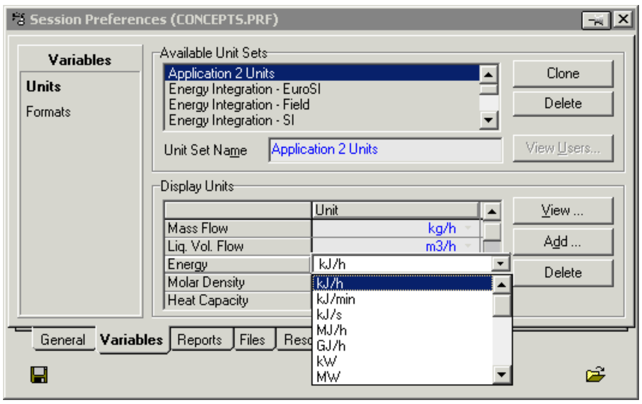
1. Start the HX-Net program, if it is not already open.
2. From the **Tools** menu, select **Preferences**. The Session Preferences view appears.
3. Click the **Variables** tab, then select the **Units** page.

The current set does not use the units required for this tutorial, so you will create a new set and modify the units.

4. In the Available Unit Sets list, select the unit set **SI**, then click the **Clone** button. This will create a cloned unit set named **New User.**
5. In the **Unit Set Name** field, rename the New User set to **Application 2 Units**.
6. In the **Display Units** group, scroll down the list to find the **Energy** units cell. The default unit is kJ/h.

7. In the **Energy** unit cell, click the down arrow. A drop-down list appears containing various unit options as shown in the figure below.

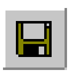
Figure 3.5



The unit kJ/s is equal to kW, so in some cases you can choose which units you want to display.

8. From the drop-down list, select **kJ/s.**
9. In the Display Units group, scroll to the **MCp** cell.
10. In the **MCp** cell, click the drop-down arrow and select **kJ/C-s.**
11. Scroll through the rest of the list and change the units for the following variables:
    - Ht Tran Coeff (kJ/s-m2-C)
    - Heat Flux (kJ/s-m2)
    - Fouling (C-m2/kW)
    - Enthalpy per Length (kJ/s-m)
    - Power (kJ/s)

    Although some of these variables may not be used, it is always a good idea to keep all of the units consistent.
12. Optional: At this point you can save the newly created preference set, which will allow you to use it for future cases.
    To save, click the **Save Preference File** icon. On the Save Preference File view, enter a file name and location, then click the **Save** button.



Save Preference File icon

Although you can overwrite the default preference set included with HX-Net, it is not recommended.

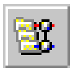
13. Click the **Close** icon to close the Session Preferences view.

3.2.2 Creating the HI Project

In this section you will create the Heat Integration (HI) Project.

To access the HI Project view, do one of the following:

- From the **Features** menu, select **HI Project**.
- Click the **Heat Integration Manager** icon, or, from the **Managers** menu, select **Heat Integration Manager**. The manager view appears. In the left list, select **HeatIntegrationProject**, then click the **Add** button.



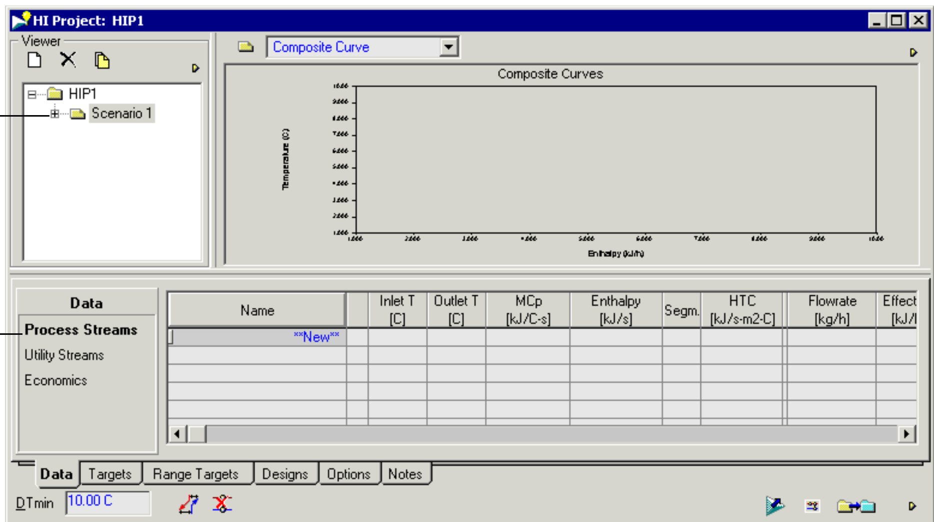
Heat Integration Manager icon

The HI Project view appears.

Figure 3.6

You should be at the Scenario level in the Viewer group.

In the Worksheet pane, you should be on the Process Streams page of the Data tab

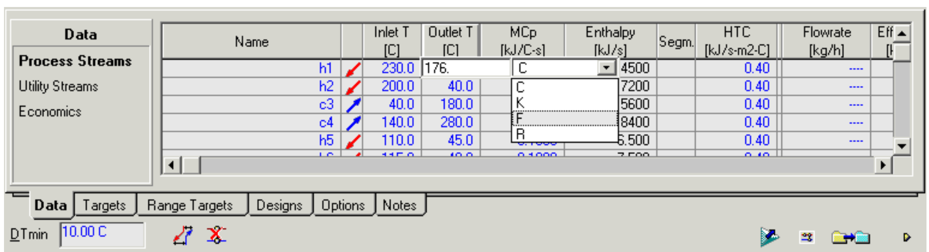


3.2.3 Entering Process Stream Data

In this section you will enter data for the all the process streams.

1. Ensure that you are in the Scenario view, **Data** tab, **Process Streams** page.
2. In the **Name** column, click on ****New****.
3. Type **h1**, then press the ENTER key. The cursor automatically moves to the **Inlet T** cell.
4. In the **Inlet T** cell enter **230°C**. The default units that appear in the unit drop-down list are already the desired unit, degrees Celsius, so they do not need to be changed.
5. In the **Outlet T** cell, enter **80°C**.
   If you know the temperature in a unit other than the default, type the known temperature in the cell, then select the appropriate units from the drop-down list, as shown in the figure below. HX-Net automatically converts the value to the default units. For example, if you enter 176°F, HX-Net converts this value to 80°C.

Figure 3.7



6. In the **MCp** cell, enter 30 kJ/°C-s.
7. In the **HTC** cell, enter 0.4 kJ/s-m2-C.

After you enter the inlet and outlet temperatures, HX-Net knows if the stream type is hot or cold. A red or blue arrow appears in the second column. A red arrow indicates a hot stream; a blue arrow indicates a cold stream.

Next you will add more streams to the HI Project.

Using the procedure you just learned, enter the data for following process streams. The stream information provided is from U. Shenoy (1995)[1].

Name	Inlet T (°C)	Outlet T (°C)	MCp (kJ/°C-s)	HTC (kJ/s-m2-C)
h2	200	40	45	0.4
c3	40	180	40	0.4
c4	140	280	60	0.4
h5	110	45	0.1	0.4
h6	115	40	0.1	0.4
h7	105	40	0.1	0.4
h8	110	42	0.1	0.4
h9	117	48	0.1	0.4
h10	103	50	0.1	0.4
c11	170	270	0.1	0.4
c12	175	265	0.1	0.4
c13	180	275	0.1	0.4
c14	168	277	0.1	0.4
c15	181	267	0.1	0.4
h16	110	45	0.1	0.4
h17	115	40	0.1	0.4
h18	105	40	0.1	0.4
h19	110	42	0.1	0.4
h20	117	48	0.1	0.4
h21	103	50	0.1	0.4
c22	170	270	0.1	0.4
c23	175	265	0.1	0.4
c24	180	275	0.1	0.4
c25	168	277	0.1	0.4
c26	181	267	0.1	0.4
h27	115	42	0.1	0.4
h28	117	43	0.1	0.4

8. Verify that the information you just entered on the **Process Streams** page matches the figures in the table above.

3.2.4 Entering Utility Stream Data

In this section, you will specify all the required heating and cooling utilities for the HEN design.

1. On the **Data** tab, select the **Utility Streams** page.

> **The hot and cold status bars at the bottom of the tab display "insufficient", which means there are not enough cold and hot utilities to satisfy the process streams.**

Click the drop-down arrow in the **Name** column to view a list of default utilities available within HX-Net.

Usually, you would use these values, but for this tutorial you will define the utilities manually.

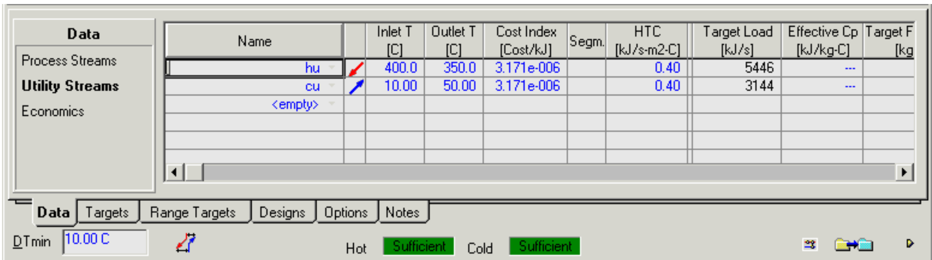
2. First you will define the hot utility. In the **Name** column, click in the **<empty>** cell. Type **hu**, then press ENTER.
3. Click in the **Inlet T** cell and enter **400°C**.
4. Click in the **Outlet T** cell and enter **350°C**. This is the minimum information required for a utility.

> **The hot utility is now sufficient, which means that the hot utility entered has enough energy to heat all of the cold process streams.**

5. Click in the **HTC** cell and enter **0.4**.
6. Now you will define the cold utility. In the **Name** column, click in the **<empty>** cell and type **cu**.
7. In the **Inlet T** cell, enter **10°C**.
8. In the **Outlet T** cell, enter **50°C**.
9. In the **HTC** cell, enter **0.4**.
   Since the cost information for the utility is unknown, use the default value displayed. HX-Net requires cost information for each utility to perform the cost target calculations.

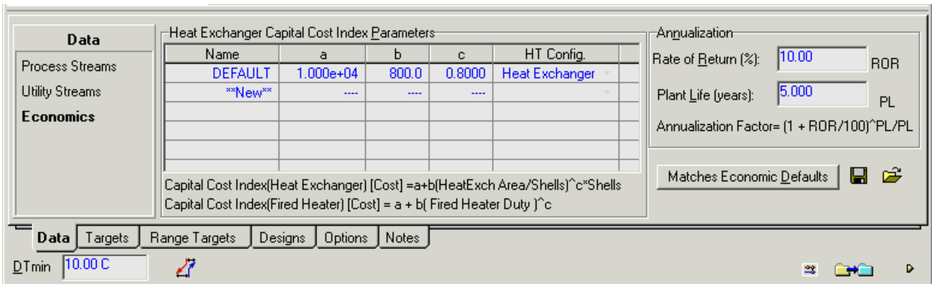
10. Verify that the information on the **Utility Streams** page appears similar to the figure below.

Figure 3.8



11. On the **Data** tab, select the **Economics** page.

Figure 3.9



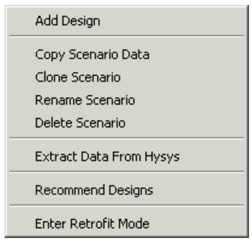
HX-Net supplies a default set of economic parameters. At least one set of economic data must be available for the calculation of the capital cost targets and network capital costs. For this tutorial, leave the default values as they are.

3.3 Generating HEN Designs

In this section you will use HX-Net's Recommend Designs feature to automatically generate HEN designs. HX-Net lets you control how many designs are generated. You can then compare the designs and make any modifications required.

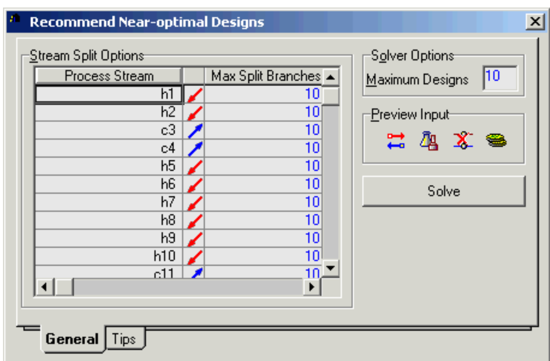
1. In the **Viewer** group, click on the Scenario level.
2. Right-click the mouse button on the selected Scenario. The following menu appears.

Figure 3.10



3. From this menu, select **Recommend Designs**. The Recommend Designs view appears as shown in the figure below. This view allows you to control certain aspects of the automatic design feature.

Figure 3.11



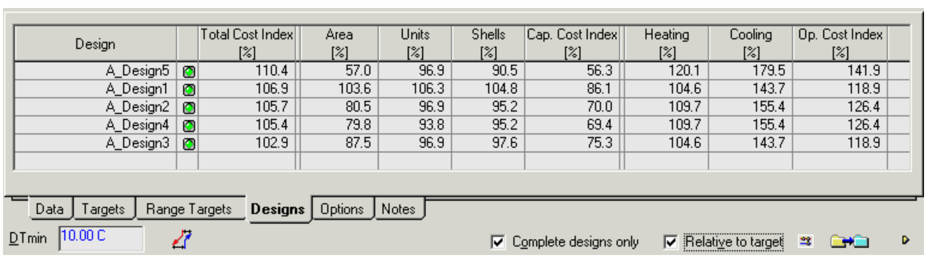
HX-Net is capable of solving for more than five designs, however, the time required for the calculations increases with the number of designs. For learning purposes, five will be sufficient. If you have time and want to calculate more, change the maximum design value to the necessary number.

4. On the **General** tab, in the **Stream Split Options** group, you can set the maximum number of branch splits. Accept the current default values.
5. In the **Preview Input** group, you can preview any of the input values for the process streams, utility streams, economics or forbidden matches. For this tutorial, leave all current default values as they are.
6. In the **Solver Options** group, change the maximum number of designs to **5**.
7. Click the **Solve** button. HX-Net will begin automatically creating heat exchanger networks. Depending on the speed of your system, this could take up to ten minutes to complete.

All generated designs will have a name that starts with "A_", indicating that these have been automatically generated by HX-Net.

8. At the Scenario level, click the **Designs** tab, then check the **Relative to target** checkbox. This will show all the key variables as a percentage of the calculated target value. The view should appear similar to the figure below. HX-Net sorts the results by Total Cost Index.

Figure 3.12



Design	Total Cost Index [%]	Area [%]	Units [%]	Shells [%]	Cap. Cost Index [%]	Heating [%]	Cooling [%]	Op. Cost Index [%]
A_Design5	110.4	57.0	96.9	90.5	56.3	120.1	179.5	141.9
A_Design1	106.9	103.6	106.3	104.8	86.1	104.6	143.7	118.9
A_Design2	105.7	80.5	96.9	95.2	70.0	109.7	155.4	126.4
A_Design4	105.4	79.8	93.8	95.2	69.4	109.7	155.4	126.4
A_Design3	102.9	87.5	96.9	97.6	75.3	104.6	143.7	118.9

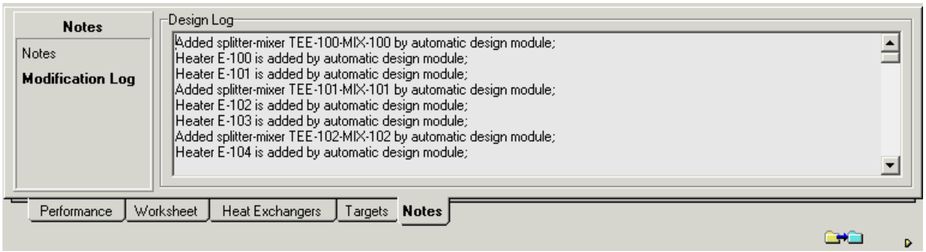
From the figure above, some of the designs recommended have total cost indexes higher than the target values, but area values less than the target values. This minimal area has been made possible by exceeding the utility energy targets. In most cases this will be true; to minimize area you must increase utility consumption, and vice-versa. From the previous figure, it appears as though A_Design3 has the smallest total cost index, and A_Design5 has the smallest total area.

9. In the Viewer group, click the design level **A_Design3.**

> **Depending on your settings, you may have slightly different results in the list of recommended designs. For the step above, click on the design that has the smallest total cost index.**

10. On the **Notes** tab, select the **Modification Log** page. This page displays all the actions performed by HX-Net during the creation of this network design.

Figure 3.13



All of the designs generated by HX-Net are optimal for the given network structure, however, if you had a design that was not already minimized for area or cost, you can optimize the design by using the retrofit options described in **Chapter 4 - Heat Exchanger Network Retrofit**.

3.4 References

[1] Shenoy, U.V., Heat Exchanger Network Synthesis: Process Optimization by Energy and Resource Analysis, Gulf Publishing Company, Houston, USA, 1995.

4 Heat Exchanger Network Retrofit

4.1 Introduction

If you are a new user to HX-Net, it is highly recommended that you complete the HI Case tutorial (refer to **Chapter 1 - Crude Pre-Heat Train Network**) and HI Project tutorial (refer to **Chapter 3 - Automatic HEN Design in HI Project**) before starting this tutorial.

It is assumed that you know how to add and complete heat exchangers on the Grid Diagram.

In this tutorial, you will use the Automatic Retrofit feature of HX-Net. You will start by creating a heat exchanger network (HEN) in the HI Project environment. Then, you will enter the Automatic Retrofit environment to retrofit the HEN. During the retrofit, you will use four methods:

- modifying utility exchangers
- resequencing heat exchangers
- repiping heat exchangers
- adding heat exchangers

HEN retrofit focuses on modifying an existing heat exchanger network to improve energy efficiency. In the past, HEN retrofits using Pinch technology required an expert user, and the Mathematical programming method reduced the interaction between the design engineer and HX-Net. HX-Net performs the HEN Retrofit algorithm one step at a time so the engineer may still control the decision making process.

The design engineer can apply constraints during the design process that will affect the retrofit calculations. Within the retrofit environment, the design engineer is required to choose one type of modification at a time. The design engineer also has to assess the operational safety and practicality of the optimal designs generated by HX-Net.

4.2 Creating a HI Project for Retrofit

The following sections describe how to create a HEN design within the HI Project environment.

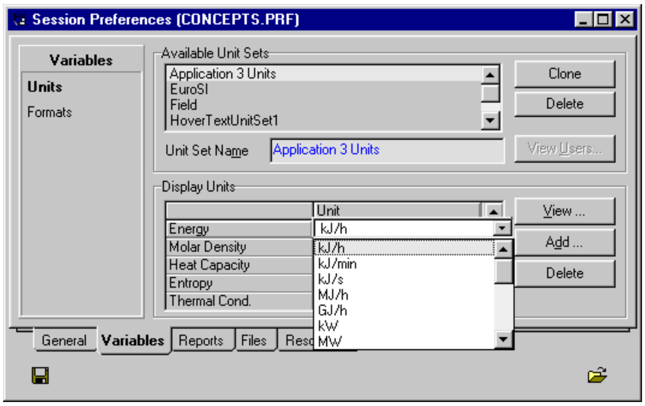
4.2.1 Setting Unit Preferences

Before you begin, verify that the units currently selected in the session preferences are the ones you want to use. For this tutorial, the temperatures are in degrees Celsius, and the MCp is in kJ/°C-s.

1. Open HX-Net if it is not already open.
2. From the **Tools** menu, select **Preferences**. The Session Preferences view appears.
3. Click the **Variables** tab, then select the **Units** page.
4. In the Available Unit Sets group, select the unit set **SI**.
   The default energy units appear in the units of kJ/h, so, you will need to define a new unit set.
5. With the unit set **SI** selected, click the **Clone** button.
   This will create a cloned unit set named New User.
6. In the **Unit Set Name** field, rename the New User set to **Application 3 Units**.
7. In the Display Units group, scroll down the list to find the **Energy** units cell. The default unit is kJ/h.

8. In the **Energy** units cell, click the down arrow. A drop-down list appears containing various unit options as shown in the figure below.

Figure 4.1

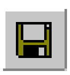


9. From the drop-down list, select **kJ/s**.
10. In the Display Units group, scroll to the **MCp** cell.
11. In the **MCp** cell, click the drop down list and select **kJ/C-s**.

The unit kJ/s is equal to kW, so in some cases you can choose which units you want to see displayed.

12. Scroll through the rest of the list and change the units for the following variables:
    - Ht Tran Coeff **(kJ/s-m2-C)**
    - Heat Flux (**kJ/s-m2**)
    - Fouling (**C-m2/kW**)
    - Enthalpy per Length (**kJ/s-m**)
    - Power (**kJ/s**)

Although some of these variables might not be used, it is always a good idea to keep all of the units consistent.



Save Preference Set icon

Although you can overwrite the default preference set included with HX-Net, it is not recommended.

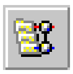
13. Optional: At this point you can save the newly created preference set, which will allow you to use it in future cases.
    To save, click the **Save Preference Set** icon. On the Save Preference File view, enter a file name and location, then click the **Save** button.

4.2.2 Creating the HI Project

Now you will create the Heat Integration (HI) Project in HX-Net.

To access the HI Project view, do one of the following:

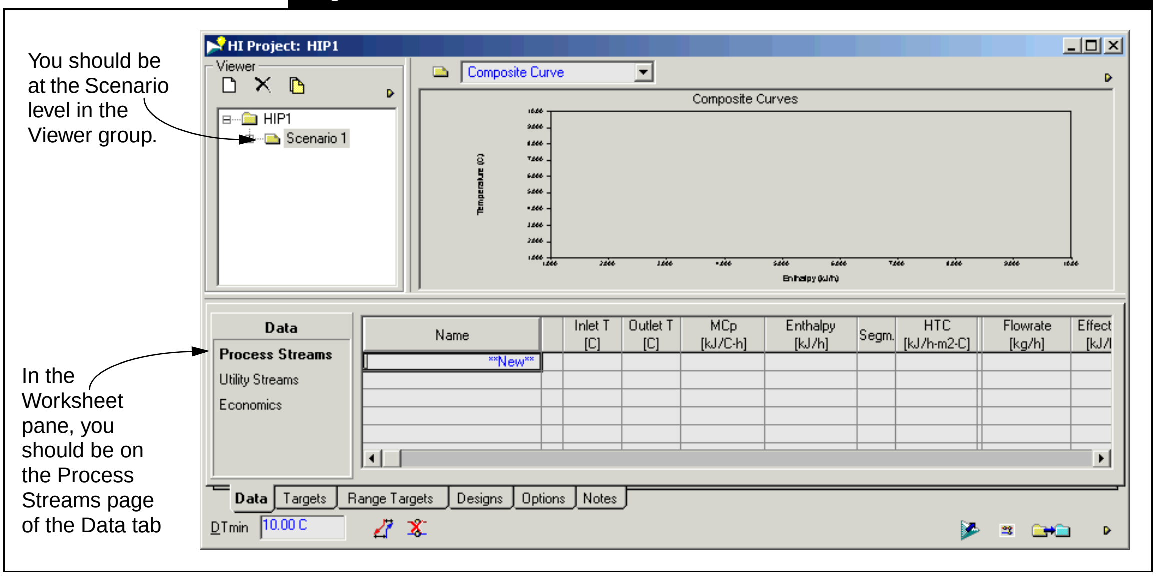
- From the **Features** menu, select **HI Project**.
- Click the **Heat Integration Manager** icon, or, from the **Managers** menu, select **Heat Integration Manager.** The manager view appears. In the left list, select **HeatIntegrationProject**, then click the **Add** button.



Heat Integration Manager icon

The HI Project view appears.

Figure 4.2

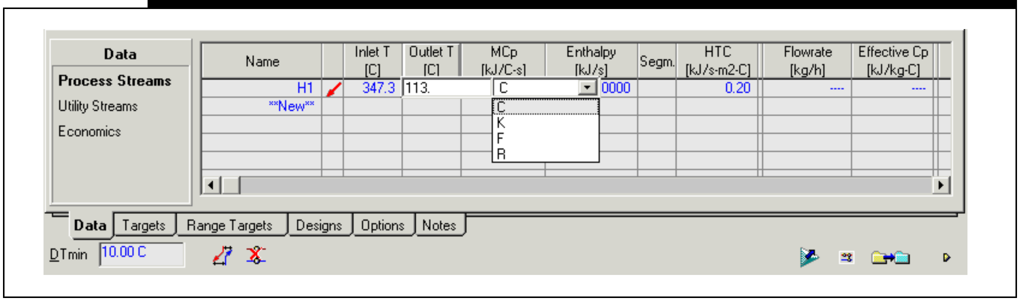


4.2.3 Entering the Process Stream Data

In this section, you will enter data for the process streams on the Process Stream page on the Data tab.

1. In the **Name** column, click on **New**.
2. Type H1, then press the **ENTER** key. The cursor automatically moves to the **Inlet T** cell.
3. In the **Inlet T** cell, enter 347.3°C. The default units that appear in the unit drop-down list are already in degrees Celsius, so they do not need to be changed.
4. In the **Outlet T** cell, enter 45°C.
   If you know the temperature in a unit other than the default, type the known temperature in the cell, then select the appropriate units from the drop-down list, as shown in the figure below. HX-Net automatically converts the value to the default units. For example, if you enter 113°F, HX-Net converts this value to 45°C.

Figure 4.3



5. In the **MCp** cell, enter 180.1 kJ/°C-s.

After you enter the inlet and outlet temperatures, HX-Net knows if the stream type is hot or cold. A red or blue arrow appears in the second column. A red arrow indicates a hot stream; a blue arrow indicates a cold stream.

Next, you will segment this stream and add other streams to the HI Project.

Segmenting Process Streams

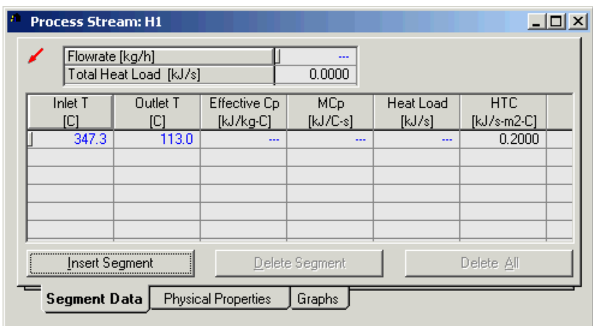
The H1 stream requires an enthalpy or heat capacity value to be complete. All other information is optional. When you enter this information, HX-Net generates a default heat transfer coefficient value for the HTC cell. For this tutorial, you will use the default coefficient values generated by HX-Net.

Double-click in the HTC column to open the HTC Default Values view, which contains a list of default heat transfer coefficients for various materials. Accept the default value or select a new default value.

In this section you will add streams and segment some of the streams. Stream segmentation is extremely useful for streams that change phase or have non-linear variations in enthalpy as the temperature changes.

1. In the H1 stream row, double-click on any cell (except HTC) to open the Process Stream view.

Figure 4.4



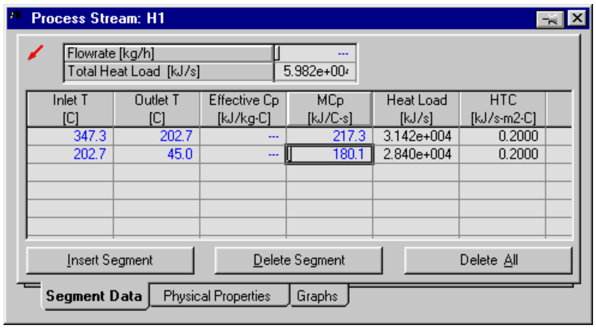
2. Click once in the **Outlet T** cell containing the value 45.0. This is the target outlet temperature.

The blank segment row always appears above the row containing the cursor.

3. Click the **Insert Segment** button. A blank row appears above the target outlet temperature.
4. The outlet temperature of the first segment is 202.7°C.
   Click in the empty **Outlet T** cell and enter **202.7.**
   HX-Net automatically inserts the inlet temperature for the following segment.

5. The MCp for the first segment is 217.3 kJ/°C-s.
   Click in the empty **MCp** cell and enter **217.3.**
   The process stream is complete as shown in the figure below.

Figure 4.5




Close icon

6. Click the **Close** icon to return to the **Data** tab of the HI Project view.
7. Now that you know how to successfully enter process stream information and create segmented streams, enter the following stream information.

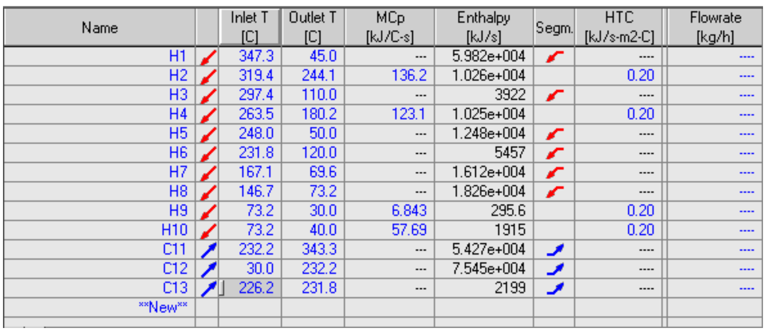
> **Enter the stream name, first Inlet T value and the target (last) Outlet T value on the Process Streams tab before accessing the Process Stream view to enter the segment information.**
>
> **Enter only the Outlet T values and the MCp values; the Inlet T values are calculated for you.**

Stream Name	Inlet T (°C)	Outlet T (°C)	MCp (kJ/°C-s)
H2	319.4	244.1	136.2
H3	297.4	203.2	22.08
	203.2	110.0	19.76
H4	263.5	180.2	123.1
H5	248.0	143.7	67.41
	143.7	50.0	58.11
H6	231.8	176.0	51.14
	176.0	120.0	46.49
H7	167.1	116.1	172.0
	116.1	69.6	158.1

Stream Name	Inlet T (°C)	Outlet T (°C)	MCp (kJ/°C-s)
H8	146.7	133.3	233.6
	133.3	120.0	202.2
	120.0	99.9	169.7
	99.9	73.2	338.2
H9	73.2	30.0	6.843
H10	73.2	40.0	57.69
C11	232.2	274.3	471.9
	274.3	343.3	498.6
C12	30.0	108.1	333.6
	108.1	211.3	381.2
	211.3	232.2	481.2
C13	226.2	228.7	352.2
	228.7	231.8	425.4

8. Verify that the information on the **Process Streams** page appears similar to the following figure.

Figure 4.6



Name		Inlet T [C]	Outlet T [C]	MCp [kJ/C-s]	Enthalpy [kJ/s]	Segm.	HTC [kJ/s-m2-C]	Flowrate [kg/h]
H1		347.3	45.0	---	5.982e+004		----	----
H2		319.4	244.1	136.2	1.026e+004		0.20	----
H3		297.4	110.0	---	3922		----	----
H4		263.5	180.2	123.1	1.025e+004		0.20	----
H5		248.0	50.0	---	1.248e+004		----	----
H6		231.8	120.0	---	5457		----	----
H7		167.1	69.6	---	1.612e+004		----	----
H8		146.7	73.2	---	1.826e+004		----	----
H9		73.2	30.0	6.843	295.6		0.20	----
H10		73.2	40.0	57.69	1915		0.20	----
C11		232.2	343.3	---	5.427e+004		----	----
C12		30.0	232.2	---	7.545e+004		----	----
C13		226.2	231.8	---	2199		----	----
New								

4.2.4 Entering Utility Stream Data

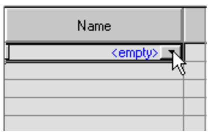
In this section, you will specify all the required heating and cooling utilities for the HEN design.

1. Click on the **Data** tab, then click on the **Utility Streams** page.

> **The hot and cold status bars at the bottom of the tab displays “insufficient”, which means that there are not enough cold and hot utilities to satisfy the process streams.**

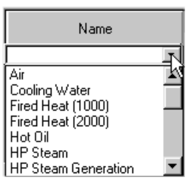
2. In the **Name** Column, click on **<empty>**. A drop-down arrow becomes active.

Figure 4.7



3. Click the arrow, and a drop-down list appears containing all of the default utilities available within HX-Net.

Figure 4.8



4. Select the following default utilities from the list:

- Cooling Water
- Fired Heat (1000) - If a warning box appears, click **OK**.
- HP Steam
- LP Steam Generation
- MP Steam
- MP Steam Generation

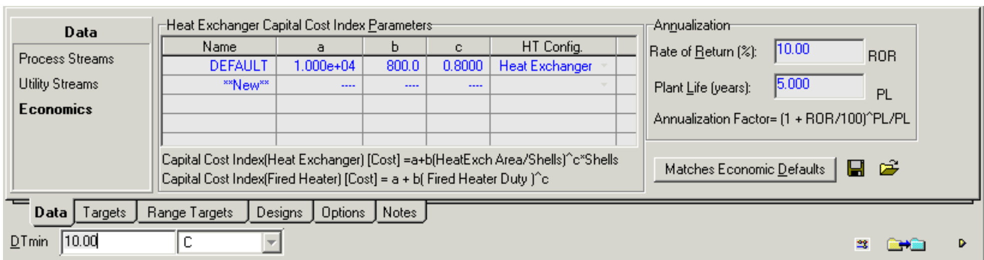
You can change or define economics parameters on the Economics page of the Data tab. Refer to step #6.

5. Verify that the information on the **Utility Streams** page appears similar to the figure below.

Figure 4.9

6. On the **Data** tab, click on the **Economics** page.

Figure 4.10



HX-Net supplies a default set of economic parameters for a typical heat exchanger. Here, you can change or add another type of installation cost and area-related cost law coefficient if required. At least one set of economic data must be available for the calculation of the capital cost targets and network capital costs.

For this tutorial, you will use the default values.

4.2.5 Building the Heat Exchanger Network

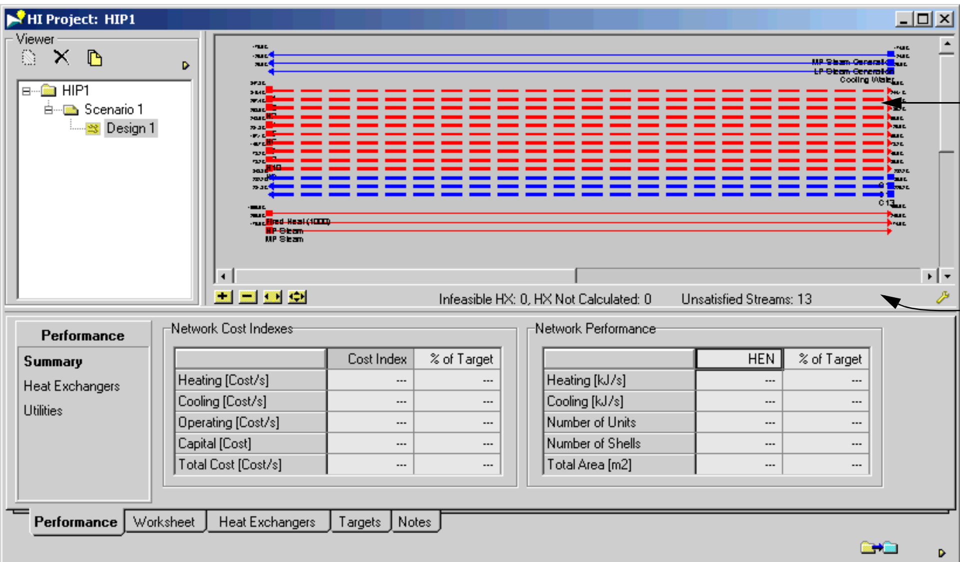
In this tutorial, you will add heat exchangers only to the heat exchanger network. Splitters are not required.

Accessing the Design Level

To build the HEN diagram, you must enter the Design level of the HI Project view.

1. In the **Viewer** group, click on the + beside **Scenario 1** to expand the tree.
2. Click on the design named **Design 1**. The view appears as shown in the figure below.

Figure 4.11

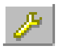


The Main pane displays the Grid Diagram instead of the plots.

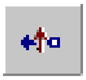
The status bar is not green. There are 13 unsatisfied streams.

Setting the Grid Diagram View Options

To modify the appearance of the heat exchanger network design in the Grid Diagram:



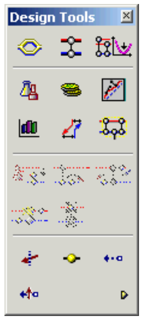
Open Palette View icon



Open Property Preset View icon

1. Open the Property Presets view by doing one of the following:
   - Click the **Open Palette View** icon. The Design Tools palette appears.Click the **Open Property Preset View** icon.

Figure 4.12

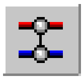


   - Right-click on the Grid Diagram, then select **Properties** from the Object Inspect menu.
2. The Property Presets view appears.
   Select **Preset 4: (Temperature)**, then click the **Edit** button.
   The Property Preset: Preset 4: (Temperature) view appears.
3. Click the **Annotations** tab.
4. In the Heat Exchangers group, click the **Middle** drop-down list.
5. From the drop-down list, select **Name**.
   The stream name will now appear in the Grid Diagram.
6. Close both the Property Presets and Property Preset: Preset 4: (Temperature) views to return to the HI Project view.

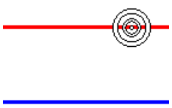
Adding Heaters

In this section, you will add heat exchangers to the network design.

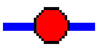
The Design Tools palette must be visible before you can add heat exchangers.



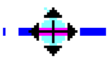
Add Heat Exchanger icon



Bull's eye icon



Red dot icon

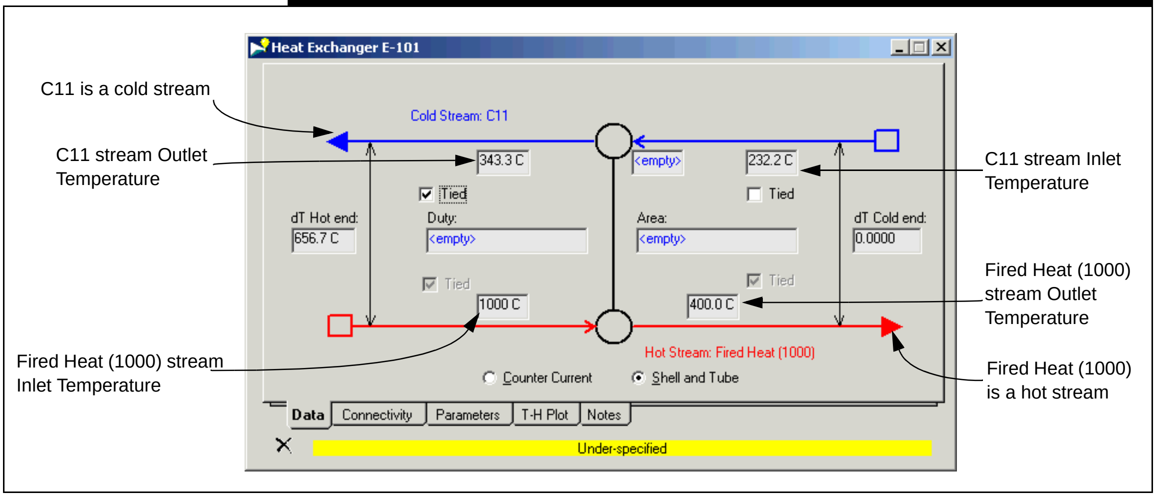


Light blue dot icon (under four arrows)

1. Press **F4** to open/access the Design Tools palette.
2. In the Design Tools palette, right-click and hold on the **Add Heat Exchanger** icon.
3. Drag the cursor over the C11 stream until the **Bull's eye** icon appears.
4. Release the mouse button. The heat exchanger appears as a solid red dot.
5. To attach the heat exchanger to the Fired Heat (1000) stream, click and hold on the red dot, then drag the cursor to the Fired Heat stream. A light blue dot will appear underneath the cursor as you drag it to the new stream.
6. Release the mouse button. The heat exchanger appears. Since this is a heater, the heat exchanger is red.
7. Double-click either end of the heat exchanger (the red dots) to open the Heat Exchanger property view.
8. Click the **Notes** tab.
9. In the **Name** field, enter HU1.
10. Click the **Data** tab.

11. On the **Data** tab, Click the **Tied** checkbox for the C11 cold stream outlet temperature. The heat exchanger property view appears as shown in the figure below.

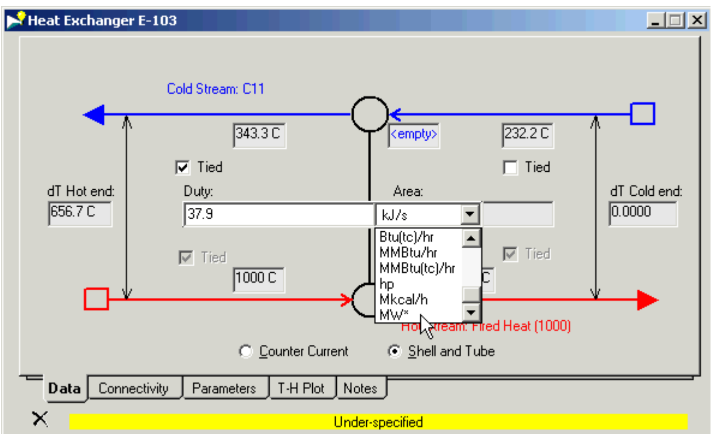
Figure 4.13



Since the C11 stream is being heated to the known outlet temperature, you can “tie” the cold stream outlet temperature value to the outlet temperature value previously entered on the **Process Streams** tab.

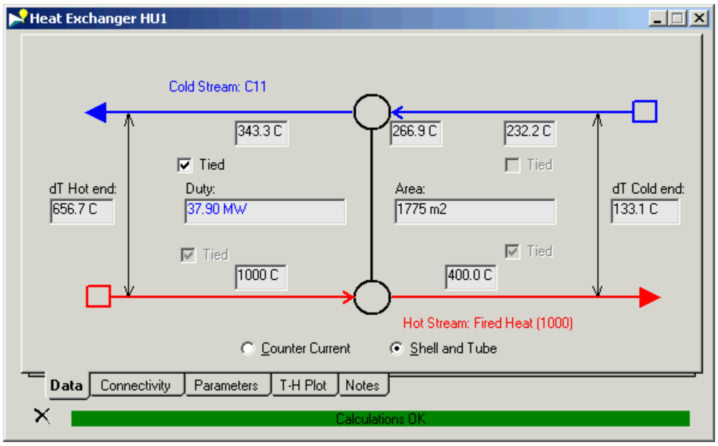
12. In the **Duty** field, enter 37.9, then select MW from the units list.

Figure 4.14



The heat exchanger solves, and the view appears as shown in the figure below.

Figure 4.15



13. Use the procedure you just learned and the data in the table below to add the other heaters.

When placing heat exchangers on the stream, remember that the hot streams flow from left to right, while the cold streams flow from right to left.

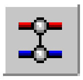
Name	Streams	Location of Heat Exchanger	Cold Stream (°C)		Duty (MW)
			Inlet T	Outlet T	
HU2	C12 & HP Steam	Place on C12 stream		Tied	27.8
HU3	C13 & HP Steam	Place on C13 stream	Tied	Tied	

“Tied” indicates that you must check the Tied checkbox as indicated. A blank cell in the table above indicates that HX-Net will calculate the value.

Adding Heat Exchangers

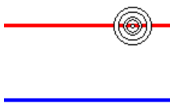
In this section, you will add heat exchangers to the Grid Diagram.

1. Press **F4** to open/access the Design Tools palette.
2. In the Design Tools palette, right-click and hold on the **Add Heat Exchanger** icon.



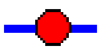
Add Heat Exchanger icon

3. Drag the cursor over the H6 stream until the **Bull's eye** icon appears.



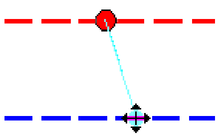
Bull's eye icon

4. Release the mouse button. The heat exchanger appears as a solid red dot.



Red dot icon

5. To attach the heat exchanger to the C12 stream, click and hold on the red dot, then drag the cursor to the C12 stream. A light blue dot will appear underneath the cursor as you drag it to the new stream.

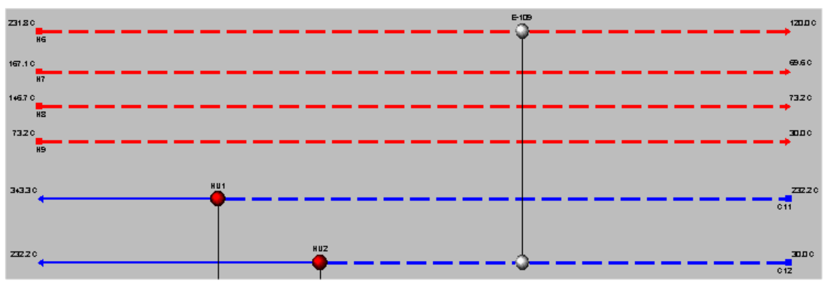


Light blue dot underneath four arrow cursor

Remember that cold streams flow from right to left. Place the new heat exchanger upstream from (to the right of) HU2 on the C12 stream.

6. Release the mouse button. The heat exchanger appears.

Figure 4.16



7. Double-click either end of the heat exchanger (the gray dots) to open the Heat Exchanger property view.
8. Click the **Notes** tab.
9. In the **Name** field, enter E1.
10. Click the **Data** tab.
11. Click the **Tied** checkbox for the inlet hot stream temperature.

You know from the stream information you entered on the Process Streams tab that the inlet hot stream temperature for E1 is the same as the initial stream temperature for H6.

12. Click the **Tied** checkbox for the outlet cold stream temperature.
13. In the **Duty** field, enter 0.7 MW. The heat exchanger solves.

If you make an error and need to delete a heat exchanger, right-click either end of the exchanger and select Delete from the Object Inspect menu.

14. Use the procedure you just learned and the data in the table below to add the rest of the heat exchangers and coolers to the Grid Diagram.

When placing heat exchangers on the stream, remember that the hot streams flow from left to right, while the cold streams flow from right to left. This is important when placing heat exchangers ‘before’ or ‘after’ other exchangers in the design.

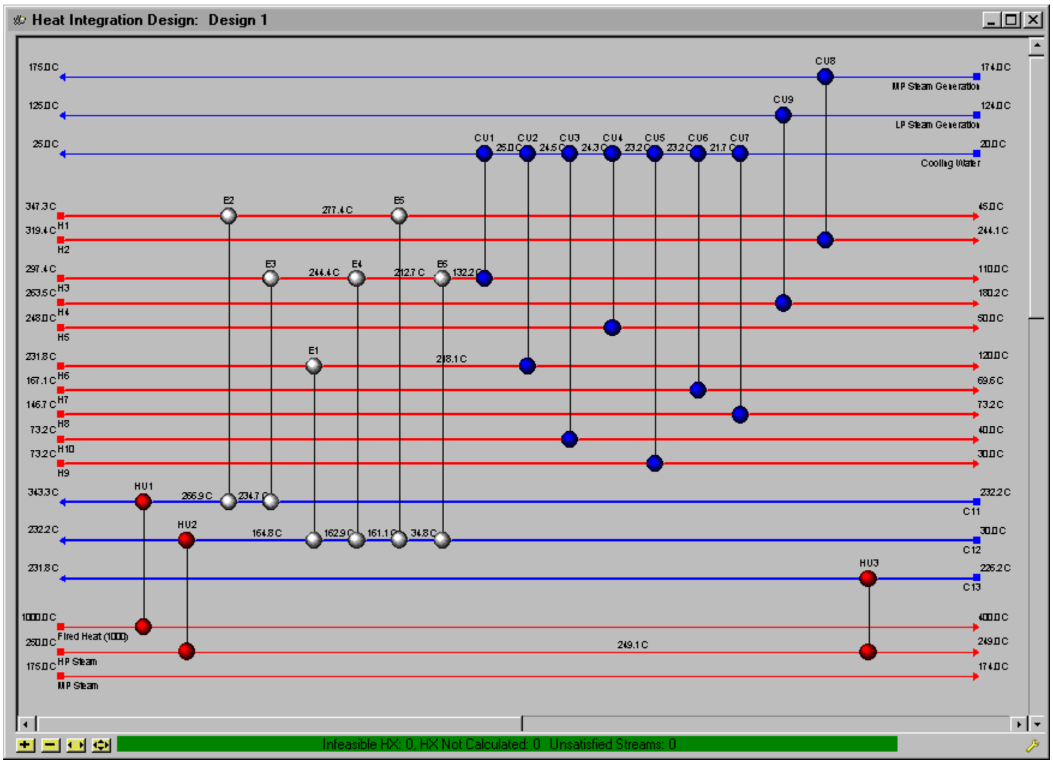
“Tied” indicates that you must check the stream Tied checkbox. A blank cell indicates that HX-Net will solve the value.

Name	Streams	Location of Heat Exchanger	Hot Stream (°C)		Cold Stream (°C)		Load/Duty (MW)
			Inlet T	Outlet T	Inlet T	Outlet T	
E2	H1 & C11	Place on C11 stream, before HU1	Tied			Tied	15.2
E3	H3 & C11	Place on C11 stream, before E2	Tied		Tied	Tied	
E4	H3 & C12	Place on H3 stream, after E3 Place on C12 stream, before E1	Tied			Tied	0.7
E5	H1 & C12	Place on H1 stream, after E2 Place on C12 stream, before E4	Tied	Tied		Tied	
E6	H3 & C12	Place on H3 stream, after E4 Place on C12 stream, before E5	Tied		Tied	Tied	
CU1	H3 & Cooling Water	Place on H3 stream, after E6	Tied	Tied			
CU2	H6 & Cooling Water	Place on H6 stream, after E1	Tied	Tied			
CU3	H10 & Cooling Water	Place on H10 stream	Tied	Tied			
CU4	H5 & Cooling Water	Place on H5 stream	Tied	Tied			
CU5	H9 & Cooling Water	Place on H9 stream	Tied	Tied			
CU6	H7 & Cooling Water	Place on H7 stream	Tied	Tied			

Name	Streams	Location of Heat Exchanger	Hot Stream (°C)		Cold Stream (°C)		Load/Duty (MW)
			Inlet T	Outlet T	Inlet T	Outlet T	
CU7	H8 & Cooling Water	Place on H8 stream	Tied	Tied			
CU8	H2 & MP Steam Generation	Place on H2 stream	Tied	Tied			
CU9	H4 & LP Steam Generation	Place on H4 stream	Tied	Tied			

After entering the information in the table above, the Grid Diagram should appear as shown in the figure below. There might be some variation in the heater placement.

Figure 4.17

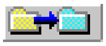


4.3 Performing the Retrofit

4.3.1 Entering the Retrofit Environment

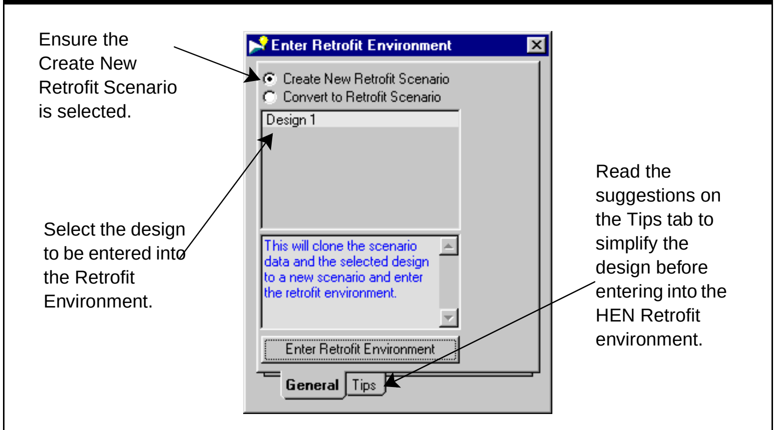
In this section, you will use HX-Net's retrofit tools to generate optimal HEN designs.

1. In the **Viewer** group, select **Scenario 1**.
2. Click the **Enter Retrofit Mode** icon located at the bottom right corner of the view.
3. The Enter Retrofit Environment view appears, as shown in the figure below.



Enter Retrofit Mode icon

Figure 4.18



While you are in the Retrofit Mode you cannot make any changes to the design and stream information.

The Tips tab contains the following information:

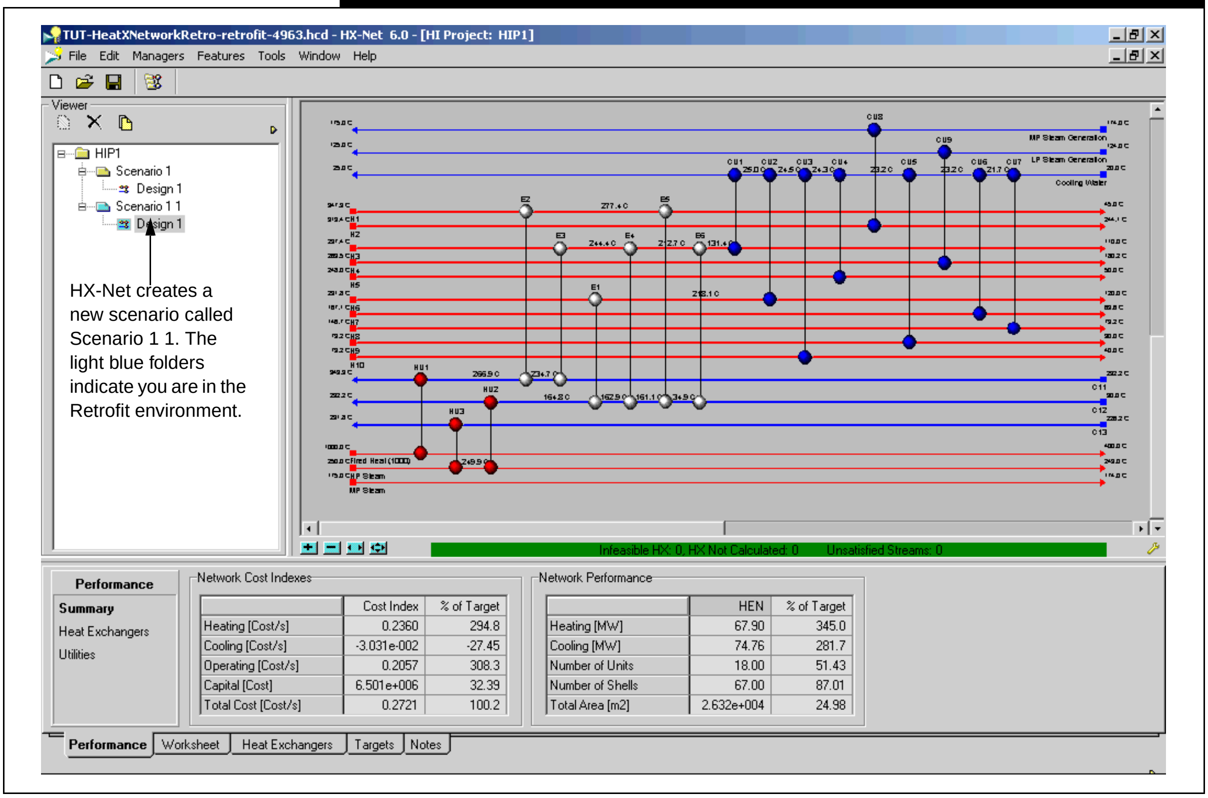
- Reduce the scope of the problem by minimizing the number of streams and heat exchangers in the heat exchanger network. Remove the exchanger(s) at either end of the streams that you don't want to modify and update the corresponding inlet or outlet stream temperatures. This can simplify the network and increase the efficiency of the model.
- Keep stream segmentation to a minimum as they increase the computational power required to solve the problem.

- Combine adjacent heat exchangers between two process streams into one heat exchange when possible. This has no effect in the final outcome but makes the solver work more efficiently.
- Remove all energy streams. Energy streams are important to establish the targets of a process but are not necessary to perform a retrofit study. By removing them, the problem becomes easier to solve.

> **Verify that the heat exchanger network represented in HX-Net matches the setup that exists in the plant before entering the retrofit environment. This will ensure that accurate and meaningful designs are generated when retrofit is performed.**

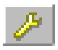
4. Click the **Enter Retrofit Environment** button.
   The following view appears (**Performance** tab, **Summary** page).

Figure 4.19

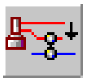


4.3.2 HEN Retrofit - Resequencing Heat Exchangers

In this section you will generate a retrofit design by resequencing the heat exchangers.



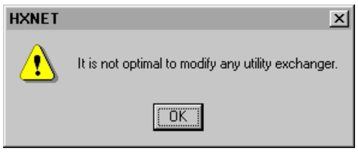
Open Palette View icon



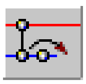
Modify utility heat exchanger icon

1. In the Viewer group, select **Design 1** under Scenario 1 1 (blue folder).
2. Click the **Open Palette View** icon. The Design Tools palette appears.
3. On the Design Tools palette, click the **Modify utility heat exchanger** icon.
   In this example, it is not optimal to modify any of the utility heat exchangers. The following view appears.

Figure 4.20

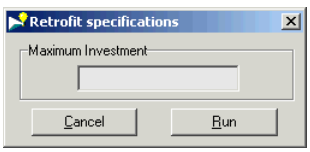


4. Click the **OK** button to close the view. The view for Scenario 1 1 appears.
5. In the Viewer group, select **Design 1** under Scenario 1 1 (blue folder) again.
6. Click the **Open Palette View** icon. The Design Tools palette appears.
7. On the Design Tools palette, click the **Move one end of a Heat Exchanger** icon. The Retrofit Specifications view appears.



Move one end of a Heat Exchanger icon

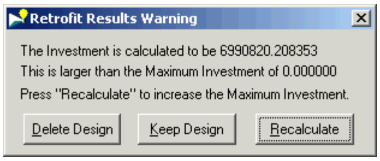
Figure 4.21



8. In the **Maximum Investment** field, enter a capital cost investment value. Entering a value here helps ensure that HX-Net will generate retrofit solutions. Leave the field blank if you want to see if HX-Net can generate a solution requiring no capital investment.

9. Click the **Run** button. HX-Net begins retrofit calculations.
   The following view will appear if HX-Net calculates a cost above what you entered in the previous step.

Figure 4.22



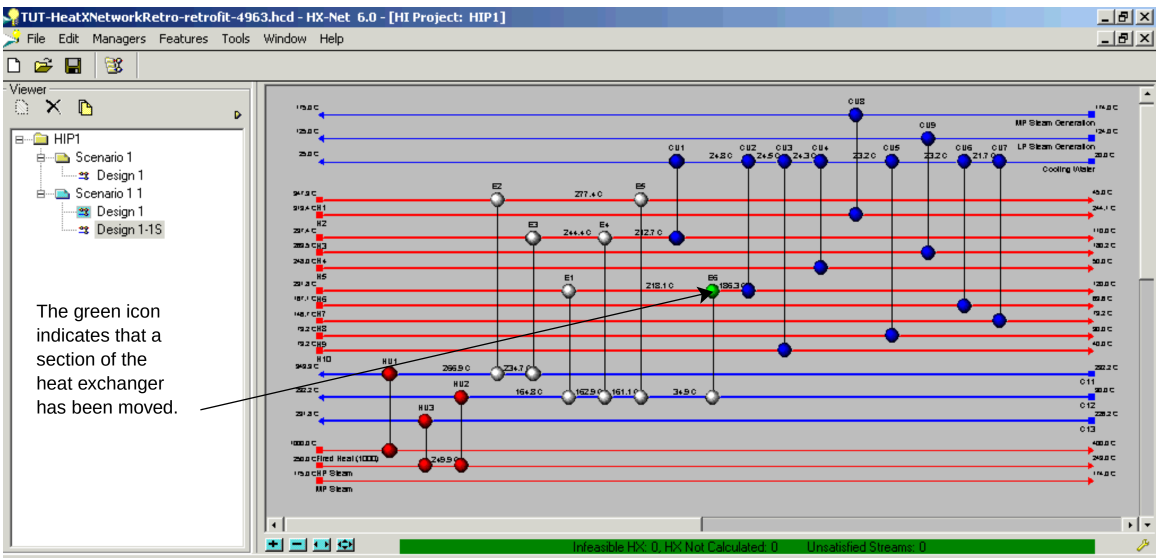
To display the exchanger names in the new design, refer to the previous **Setting the Grid Diagram View Options** section.

In this view, you can do the following:

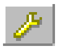
- Click **Delete Design** to exit the calculation and delete the design.
- Click **Keep Design** to finish the calculation and keep the design.
- Click **Recalculate** and enter a new Maximum Investment value.

10. For this tutorial, click **Keep Design**.
    A new design called **Design 1-1S** appears in the Viewer group.
11. In the Viewer group, select **Design 1-1S**. Verify that the Grid Diagram appears similar to the view below.

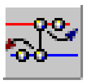
Figure 4.23



4.3.3 HEN Retrofit - Repiping Heat Exchangers



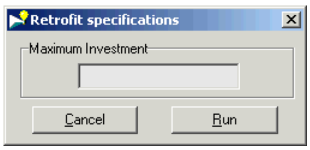
Open Palette View icon



Move both ends of a Heat Exchanger icon

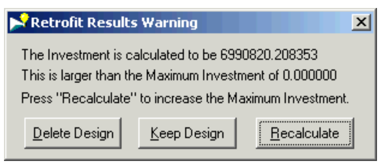
1. In the Viewer group, select **Design 1** under **Scenario 1 1** (blue folder).
2. Click the **Open Palette View** icon. The Design Tools palette appears.
3. On the Design Tools palette, click the **Move both ends of a Heat Exchanger** icon. The Retrofit Specifications view appears.

Figure 4.24



4. In the **Maximum Investment** field, enter a capital cost investment value. Entering a value here helps ensure that HX-Net will generate retrofit solutions. Leave the field blank if you want to see if HX-Net can generate a solution requiring no capital investment.
5. Click the **Run** button. HX-Net begins retrofit calculations.
   The following view will appear if HX-Net calculates a cost above what you entered in the previous step.

Figure 4.25



In this view, you can do the following:

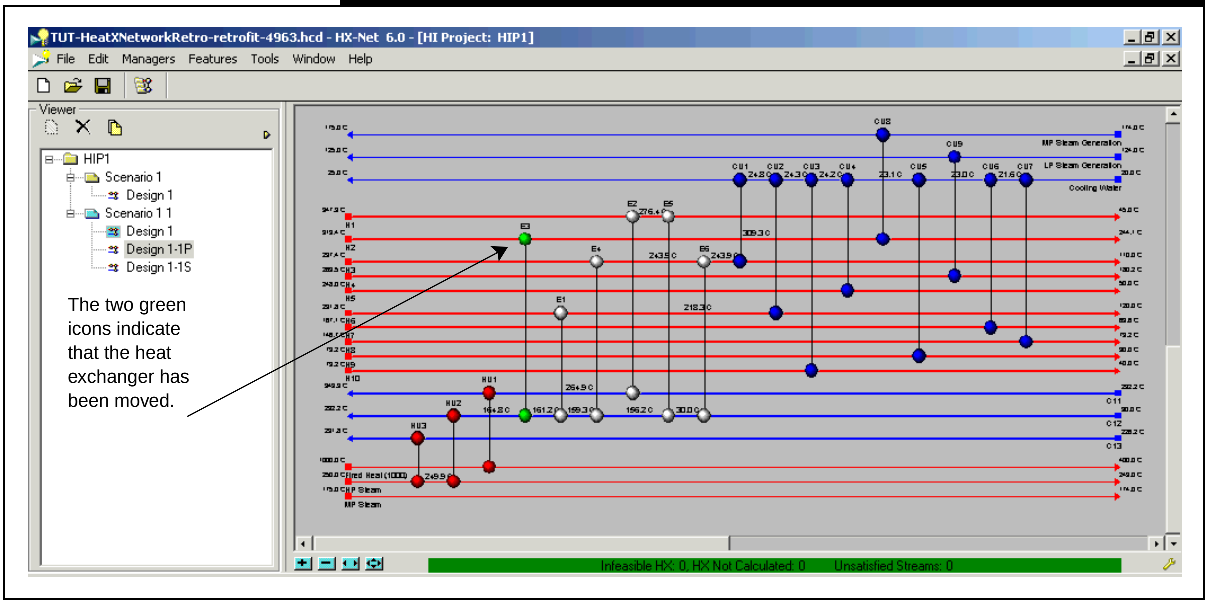
- Click **Delete Design** to exit the calculation and delete the design.
- Click **Keep Design** to finish the calculation and keep the design.
- Click **Recalculate** and enter a new Maximum Investment value.

6. For this tutorial, click **Keep Design**.
   A new design called **Design 1-1P** appears in the Viewer group.

To display the exchanger names in the new design, refer to the previous **Setting the Grid Diagram View Options** section.

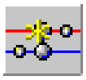
7. In the Viewer group, select **Design 1-1P**. Verify that the Grid Diagram appears similar to the view below.

Figure 4.26



4.3.4 HEN Retrofit - Adding Heat Exchangers

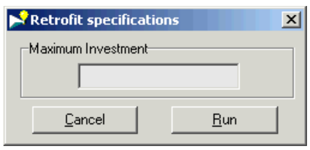
Open Palette View icon



Add Heat Exchanger icon

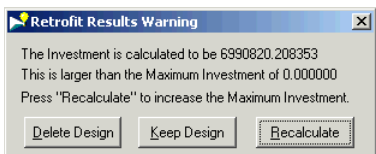
1. In the Viewer group, select **Design 1** under **Scenario 1 1**.
2. Click the **Open Palette View** icon. The Design Tools palette appears.
3. On the Design Tools palette, click the **Add a Heat Exchanger** icon. The Retrofit Specifications view appears.

Figure 4.27



4. In the **Maximum Investment** field, enter a capital cost investment value. Entering a value here helps ensure that HX-Net will generate retrofit solutions. Leave the field blank if you want to see if HX-Net can generate a solution requiring no capital investment.
5. Click the **Run** button. HX-Net begins retrofit calculations. The following view will appear if HX-Net calculates a cost above what you entered in the previous step.

Figure 4.28



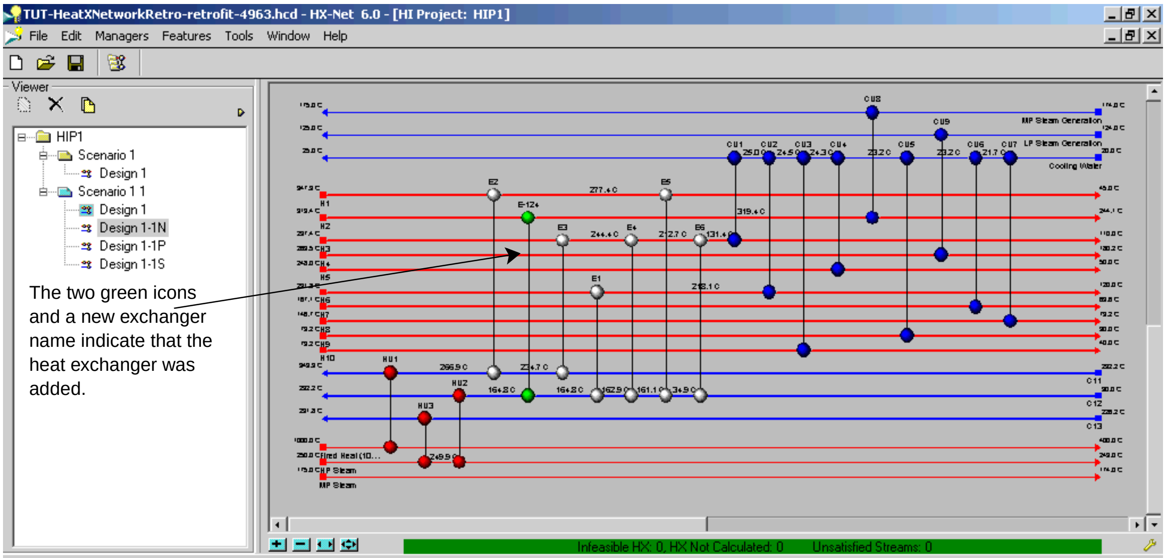
In this view, you can do the following:

- Click **Delete Design** to exit the calculation and delete the design.
- Click **Keep Design** to finish the calculation and keep the design.
- Click **Recalculate** and enter a new Maximum Investment value.

6. For this tutorial, click **Keep Design**. A new design called **Design 1-1N** appears in the Viewer group.
7. In the Viewer group, select **Design 1-1N**. Verify that the Grid Diagram appears similar to the view below.

To display the exchanger names in the new design, refer to the previous **Setting the Grid Diagram View Options** section.

Figure 4.29



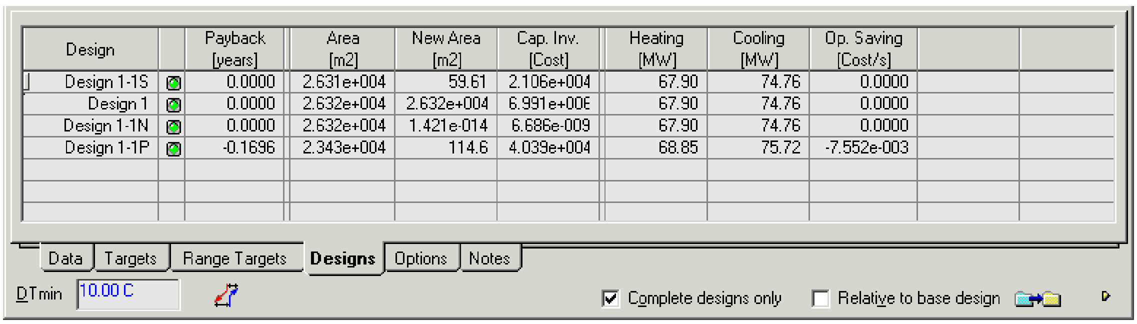
The two green icons and a new exchanger name indicate that the heat exchanger was added.

4.4 Comparing Designs

Now that HX-Net has generated three possible design improvements, you can compare each design and decide which one best suits the project requirements.

1. In the Viewer group, select **Scenario 1 1**.
2. Click the **Designs** tab. The following worksheet appears. The values displayed may be different than what appears below.

Figure 4.30



Design		Payback [years]	Area [m2]	New Area [m2]	Cap. Inv. [Cost]	Heating [MW]	Cooling [MW]	Op. Saving [Cost/s]
Design 1-1S		0.0000	2.631e+004	59.61	2.106e+004	67.90	74.76	0.0000
Design 1		0.0000	2.632e+004	2.632e+004	6.991e+006	67.90	74.76	0.0000
Design 1-1N		0.0000	2.632e+004	1.421e-014	6.686e-009	67.90	74.76	0.0000
Design 1-1P		-0.1696	2.343e+004	114.6	4.039e+004	68.85	75.72	-7.552e-003

The Designs worksheet displays data on the original and all the retrofit generated designs. For each retrofit generated design you can compare the following:

- the payback of the generated design
- new area required
- capital investment required
- energy consumption reduction
- operation costs reduction

Remember that all estimated cost values are based on the HX-Net default economic parameters. You can change the economic parameters on the **Economics** page of the **Data** tab. All modification changes are compared to the base case design.

3. Click the checkbox beside **Relative to base design** to view the above data with percent values relative to the original HEN design.

Figure 4.31

Design		Payback [years]	Area [%]	New Area [%]	Cap. Inv. [%]	Heating [%]	Cooling [%]	Op. saving [%]
Design 1-1S		0.0000	99.95	0.23	100.00	100.00	100.00	100.00
Design 1		0.0000	100.00	100.00	100.00	100.00	100.00	100.00
Design 1-1N		0.0000	100.00	0.00	100.00	100.00	100.00	100.00
Design 1-1P		-0.1696	89.02	0.44	88.32	101.41	101.28	101.34

Data | Targets | Range Targets | **Designs** | Options | Notes

DTmin 10.00 C ☑ Complete designs only ☑ Relative to base design

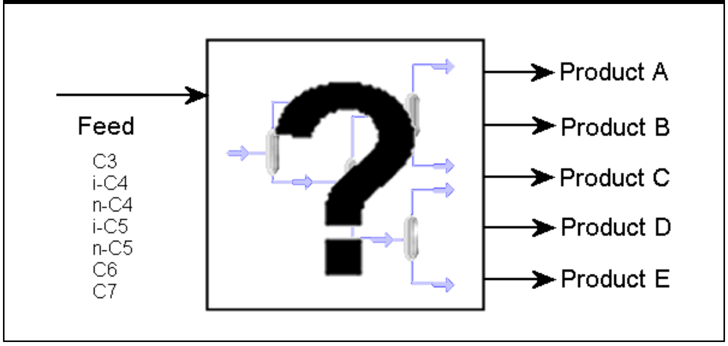
5 Column Sequencing for Multi-Components

5.1 Introduction

This tutorial serves as an introduction to the DISTIL Column Sequencing operation.

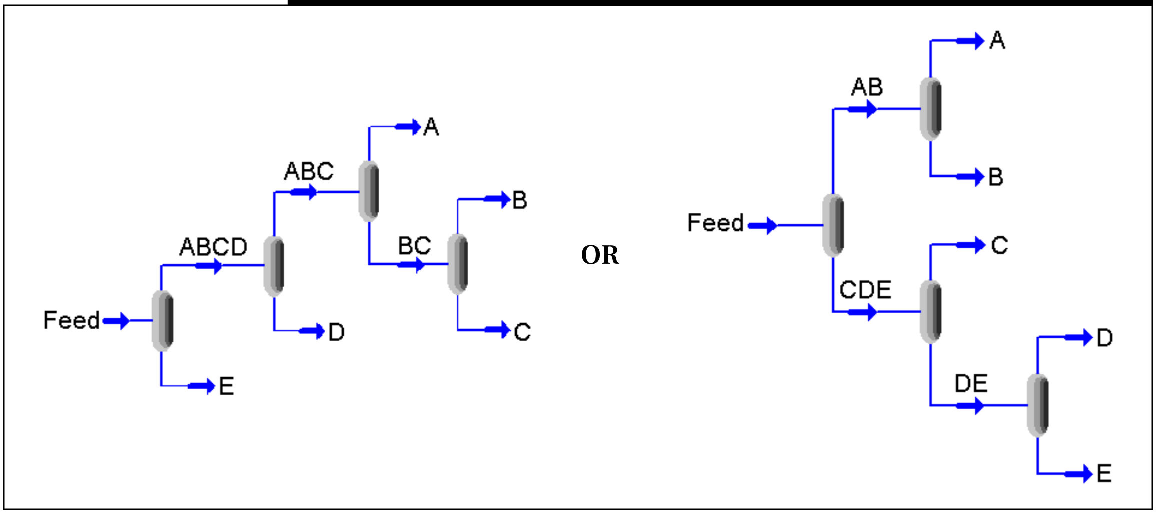
In this tutorial, you will use DISTIL to help you decide the best column sequence design to separate a feed stream into five product streams.

Figure 5.1



For a five product separation system, there are as many as 14 different column sequence designs. The figure below displays two possible sequences:

Figure 5.2



The objective of this tutorial is to illustrate how you can use the column sequencing feature for screening and scoping different design options.

For more information about *task*, refer to **Section 5.4.4 - Representation** from the **Reference Guide**.

The performance of all tasks is estimated using the Shortcut models and then the rigorous MILP model is employed to identify a few promising column sequences.

5.1.1 About the Scenarios

The following four different scenarios will be considered to see how the performance of the optimum design changes as the search domain is expanded.

- The first scenario looks at the classical column sequencing problem where all the tasks (i.e., all potential columns of sequence) operate at a fixed pressure (P = 5 bar). This scenario will illustrate how different sequences of splits affect the performance of the separation system.
- The second scenario expands the search domain by adjusting the operating pressure of all tasks to use cooling water as the cold utility and then performing column sequence screening. This scenario will illustrate that the optimum column sequence strongly depends on the operating pressure of the columns within the sequence.
- The third scenario further widens the search domain by allocating instances of tasks that perform identical separation but operate at different pressures. This scenario allows you to solve sequencing and pressure optimization problems simultaneously. The number of design options increases exponentially, but the rigorous MILP framework allows you to identify promising options without any difficulty.
- The fourth scenario imposes some constraints on the search space domain to utilize the available assets more effectively and find the most promising sequence.

For more information about instance, refer to **Section 5.4.5 - Task Classification** from the **Reference Guide**.

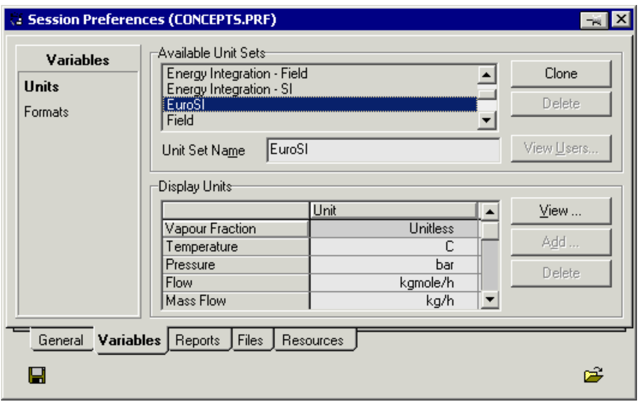
The first three scenarios assume a grass-roots situation, while the fourth scenario addresses retrofitting the real industrial plant.

5.2 Setting Unit Preferences

Before you begin setting up the simulation, verify that the units currently selected in the DISTIL preferences are the ones you want to use. For this example, the temperature is in Celsius, the pressure is in bars, and the cost per time is in $/year.

1. Open DISTIL if it is not already open.
2. From the **Tools** menu, select **Preferences**. The Session Preferences view appears.
3. Click the **Variables** tab, then select the **Units** page.

Figure 5.3



4. Select **Separation - EuroSI** from the Available Unit Sets group.

   If this unit set is not available, clone the **Euro-SI** unit set, name the new unit set Separation - EuroSI, and set the Cost Index per Time unit to **$/year**.

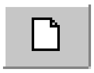
5. Scroll through the table in the Display Units group to confirm that the selected unit set contains the units you want to work with.

Blue text indicates that you can modify the value associated with the selected variable.

5.3 Creating the Fluid Package

To use the Column Sequencing operation, you have to create a fluid package containing the property package and components required to simulate the feed stream entering the separation system.

Fluid Package Manager icon



Add Fluid Package icon

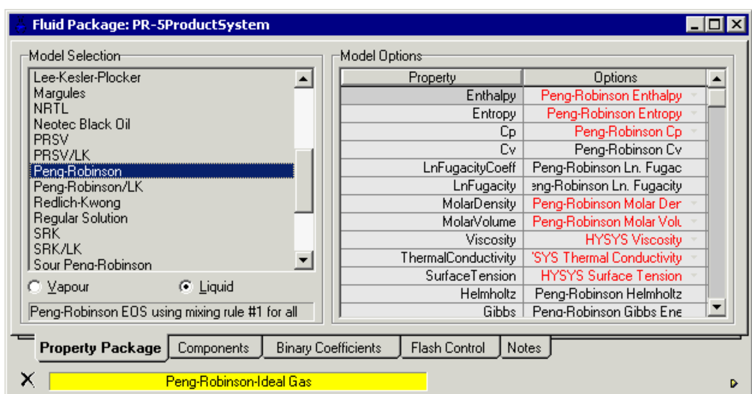
1. Open the Fluid Package Manager view by clicking the **Fluid Package Manager** icon.
2. In the Fluid Package Manager view, click the **Add Fluid Package** icon.
   The Fluid Package view will appear.

5.3.1 Selecting a Property Package

1. In the Fluid Package view, click the **Property Package** tab.
2. In the Model Selection group, select the **Vapour** radio button, then select **Ideal Gas** from the Model Selection list.
3. In the Model Selection group, select the **Liquid** radio button, then select **Peng-Robinson** from the Model Selection list.
4. Click the **Notes** tab. In the **Name** field, type **PR-5ProductSystem**.

The status bar at the bottom of the Fluid Package view should display Peng-Robinson-Ideal Gas, as shown in the figure below:

Figure 5.4



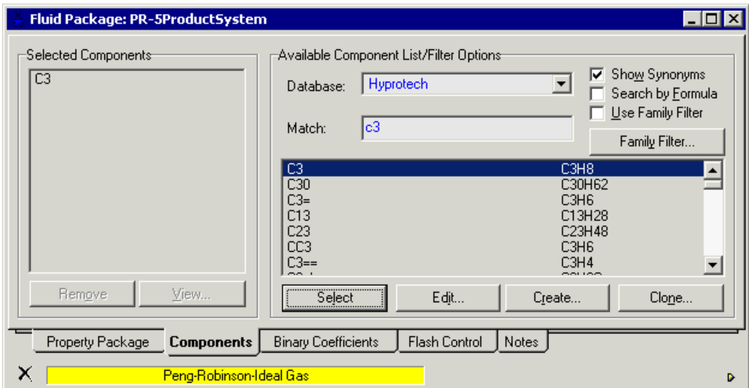
5.3.2 Selecting the Components

In this section you will add components to the property package.

1. Click the **Components** tab.
2. In the Available Component List/Filter Options group, enter **c3** in the **Match** field.
3. The list under the Match field will be sorted to display all components that have **c3** in the name.
4. Select **C3** from the list, then click the **Select** button. The component propane will be added to the Selected Components list.

You can also press **ENTER** to add the selected component from the list into the Selected Components group.

Figure 5.5

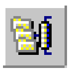


5. Repeat steps #2 to #4 to add the following components:
   - i-C4
   - n-C4
   - i-C5
   - n-C5
   - C6
   - C7
6. Click the **Close** icon to close the Fluid Package view.
7. Save the case as **5ProductDistillation**. DISTIL will save the file with ***.hcd** extension.

Close icon

5.4 Installing Column Sequencing

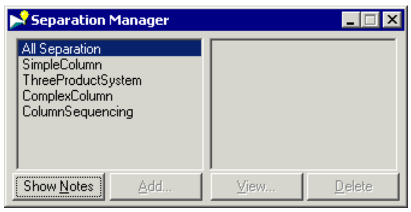
Now that you have specified the fluid package for the simulation, you are ready to use the Column Sequencing (CS) operation.



Separation Technology Manager icon

1. Click the **Separation Technology Manager** icon to open the Separation Manager view.

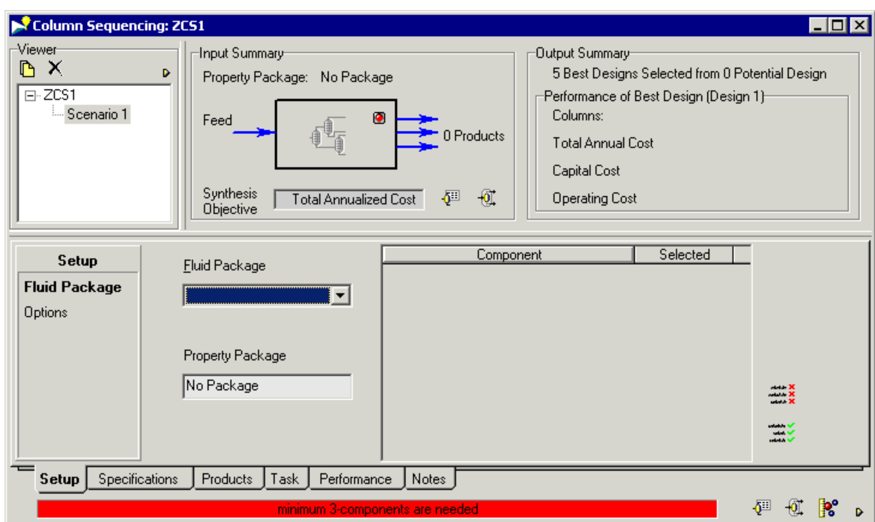
Figure 5.6



2. In the left list of the Separation Manager view, select **ColumnSequencing**.
3. Click the **Add** button, and Column Sequencing (CS) property view will appear.
   DISTIL automatically names the CS operation **CS1**.

The name of the CS operation will appear in the right list of the Separation Manager view.

Figure 5.7



The CS view is a project view. Refer to Section 2.3.5 - Project View in the User Guide for more information about project views.

The status bar will also display an error message if there are any problems during calculations.

The status bar at the bottom of the CS view indicates the status of the operation. If you have not specified the required information, the status bar will display a request for the missing information.

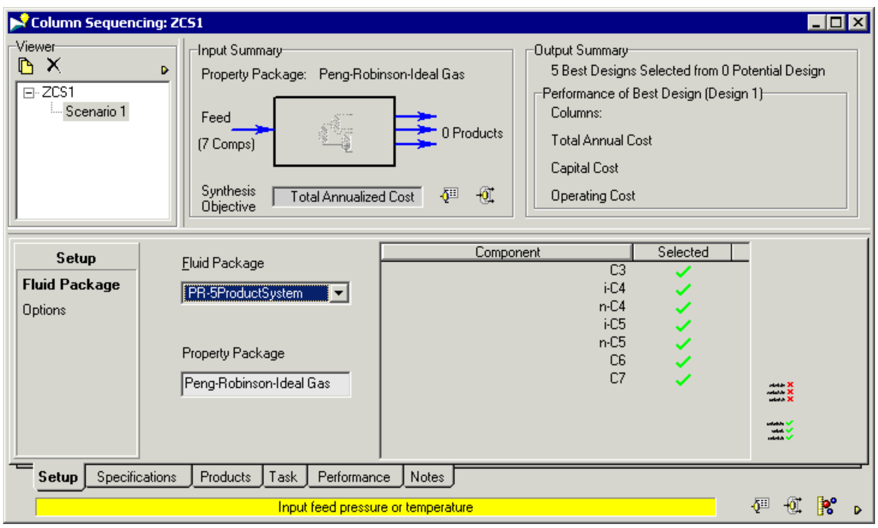
When the CS operation is first added, DISTIL automatically creates a blank scenario called Scenario 1, as shown in the Viewer pane. The Scenario 1 level is where you specify the conditions and properties of the feed and product streams, and the characteristics of the scenario.

5.4.1 Specifying the Feed Stream Conditions

In this section you will specify the feed stream:

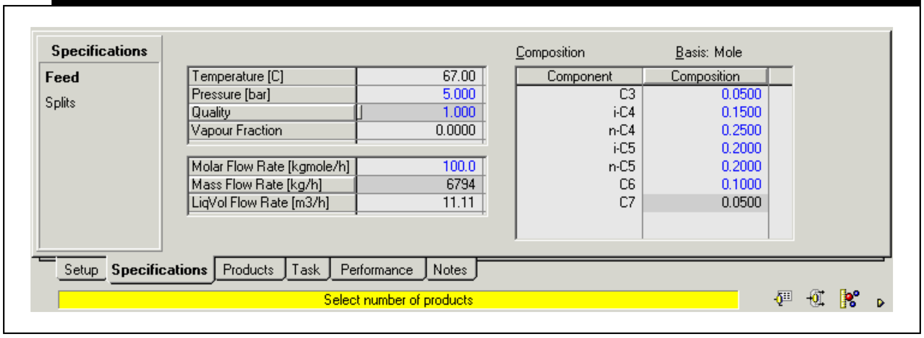
1. In the Worksheet pane, click the **Setup** tab, then select the **Fluid Package** page.
2. On the **Fluid Package** page, use the **Fluid Package** drop-down list to select the fluid package for the simulation.

Figure 5.8



3. Click the **Specifications** tab, then select the **Feed** page.
4. In the **Pressure** field, enter **5 bar**.
5. In the **Molar Flow Rate** field, enter **100 kgmole/hr**.
6. In the Composition table, specify the composition of the feed stream as shown in the figure below:

Figure 5.9



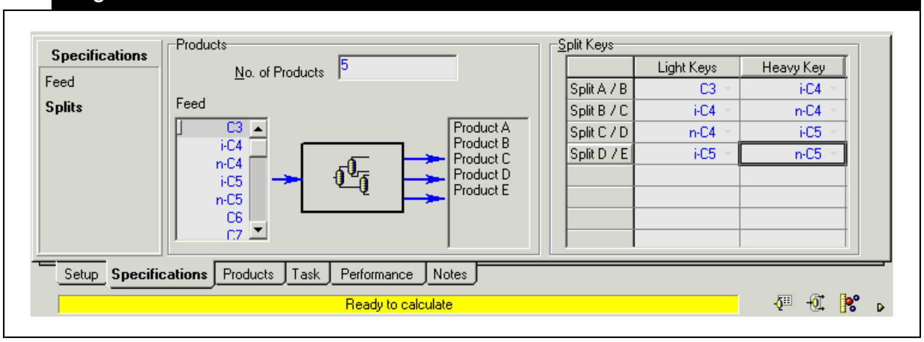
5.4.2 Specify Product Streams Conditions

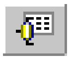
In this section, you will specify the product streams:

1. Click the **Splits** page.
2. In the **No of Product** field, type **5** and press **ENTER**.
3. Select the light and heavy key components for each product stream as shown in the figure below:

Refer to **Section 5.3.3 - Specification of Three Product System** in the **Reference Guide** for more information about light and heavy keys.

Figure 5.10

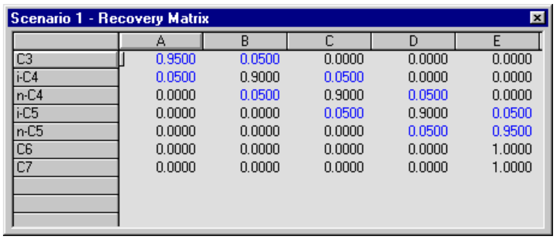




Recovery Matrix icon

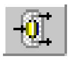
4. Click the **Recovery Matrix** icon to view the percentage split of the components in each product stream. Close the view when you are finished

Figure 5.11



Scenario 1 - Recovery Matrix

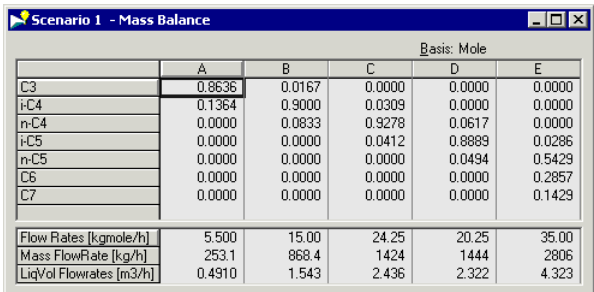
	A	B	C	D	E
C3	0.9500	0.0500	0.0000	0.0000	0.0000
i-C4	0.0500	0.9000	0.0500	0.0000	0.0000
n-C4	0.0000	0.0500	0.9000	0.0500	0.0000
i-C5	0.0000	0.0000	0.0500	0.9000	0.0500
n-C5	0.0000	0.0000	0.0000	0.0500	0.9500
C6	0.0000	0.0000	0.0000	0.0000	1.0000
C7	0.0000	0.0000	0.0000	0.0000	1.0000



Overall Mass Balance icon

5. Click the **Overall Mass Balance** icon to see the flow rate of each components and the product streams. Close the view when you are finished.

Figure 5.12

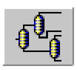


Scenario 1 - Mass Balance

Basis: Mole

	A	B	C	D	E
C3	0.8636	0.0167	0.0000	0.0000	0.0000
i-C4	0.1364	0.9000	0.0309	0.0000	0.0000
n-C4	0.0000	0.0833	0.9278	0.0617	0.0000
i-C5	0.0000	0.0000	0.0412	0.8889	0.0286
n-C5	0.0000	0.0000	0.0000	0.0494	0.5429
C6	0.0000	0.0000	0.0000	0.0000	0.2857
C7	0.0000	0.0000	0.0000	0.0000	0.1429
Flow Rates [kgmole/h]	5.500	15.00	24.25	20.25	35.00
Mass FlowRate [kg/h]	253.1	868.4	1424	1444	2806
LiqVol Flowrates [m3/h]	0.4910	1.543	2.436	2.322	4.323

Now that all the required information for the feed and product streams have been specified, the status bar at the bottom of the CS view indicates that the operation is ready to generate the feasible column sequencing designs.



Press to generate feasible column sequences icon

The **Press to generate feasible column sequences** icon will become available. This icon activates the Column Sequencing calculations and feasible column sequences/designs are generated based on the calculation results. The generated feasible designs, however, will have no relation to the scenarios/limitations in your plant.

5.5 Generating Grass Root Scenarios

As mentioned in **Section 5.1 - Introduction**, this tutorial will consider four scenarios. The first three are user-constructed, and the fourth retrofits an existing design.

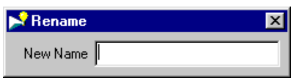
5.5.1 Columns with Constant Pressure

The first scenario considers the effect of a constant operating pressure of five bars for all the columns in the possible/feasible column sequence designs.

You already set a constant pressure for the columns when you specified the pressure of the feed stream (i.e., 5 bars).

1. Rename the Scenario by selecting **Scenario 1** in the Viewer group, and right-clicking to open the Object Inspect menu.
2. Select the **Rename Scenario** command from the Object Inspect menu. The Rename view appears.

Figure 5.13



3. Enter **Constant Pressure** in the **New Name** field, then press **ENTER**.
4. In the Worksheet pane, click the **Setup** tab, then select the **Options** page.

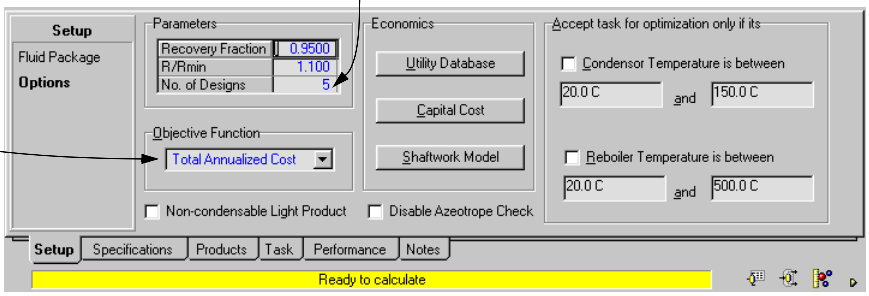
The Options page allows you to specify/change the conditions/limitations and assumptions used to generate the feasible designs.

The defaul settings in the **Options** page are correct for this scenario.

Figure 5.14

The objective function for designing the column sequences is the Total Annualized Cost (TAC).

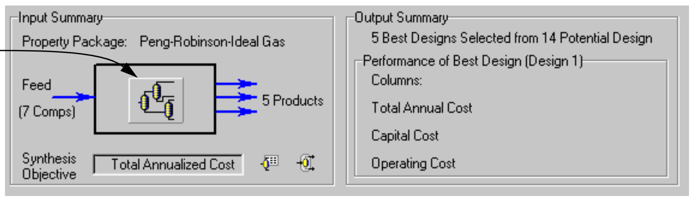
Limits the number of designs listed/kept at the top five best designs.



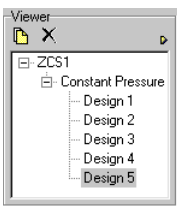
5. In the Main pane, click the **Press to generate feasible column sequences** icon to begin generating feasible designs.

Figure 5.15

Press to generate feasible column sequences icon



Generated Feasible Designs



Viewer group

The Column Sequencing (CS) operation will generate possible feasible designs and list the feasible designs in the Viewer group.

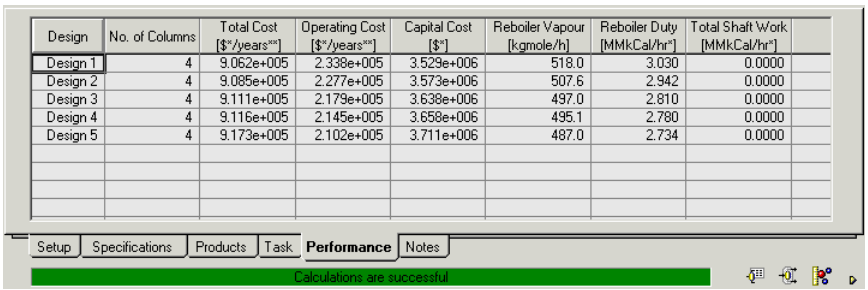
The Viewer group will contain maximum five possible designs under the Constant Pressure scenario, as specified in the Options page. The designs are also arranged in ascending order of the TAC (i.e., the design costing the least amount of TAC is first).

Viewing Summary Information

To obtain summary information of every design in the scenario, do the following:

1. Select the **Constant Pressure** scenario in the Viewer group.
2. In the Worksheet pane, click the **Performance** tab.

Figure 5.16



Design	No. of Columns	Total Cost [$*/years**]	Operating Cost [$*/years**]	Capital Cost [$*]	Reboiler Vapour [kgmole/h]	Reboiler Duty [MMkCal/hr*]	Total Shaft Work [MMkCal/hr*]
Design 1	4	9.062e+005	2.338e+005	3.529e+006	518.0	3.030	0.0000
Design 2	4	9.085e+005	2.277e+005	3.573e+006	507.6	2.942	0.0000
Design 3	4	9.111e+005	2.179e+005	3.638e+006	497.0	2.810	0.0000
Design 4	4	9.116e+005	2.145e+005	3.658e+006	495.1	2.780	0.0000
Design 5	4	9.173e+005	2.102e+005	3.711e+006	487.0	2.734	0.0000

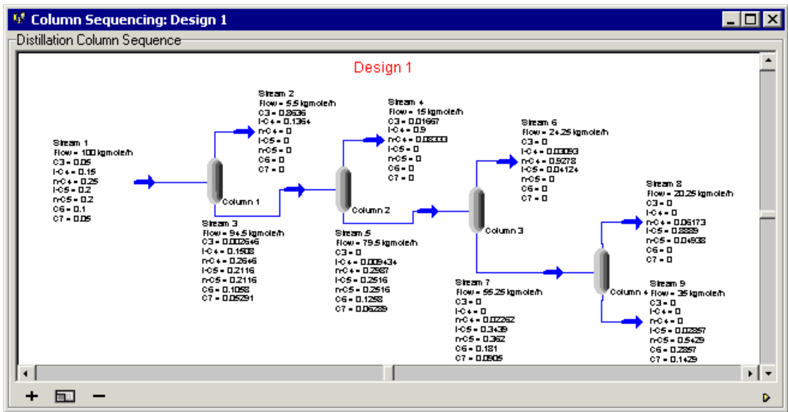
The Performance tab contains a table that lists the following information on each design: name, number of columns, total annualized cost, operating cost/year, capital cost, reboiler vapour flow rate, reboiler duty, and total shaft work.

Viewing Detailed Design Information

To view detailed information for the optimum design, do the following:

1. Select the **Constant Pressure** design in the Viewer group. The Main pane will display the Process Flow Diagram of the selected design.

Figure 5.17



2. In the Worksheet pane, click the **Streams** tab. This tab contains the information for all the streams in the selected column sequence design.

Figure 5.18

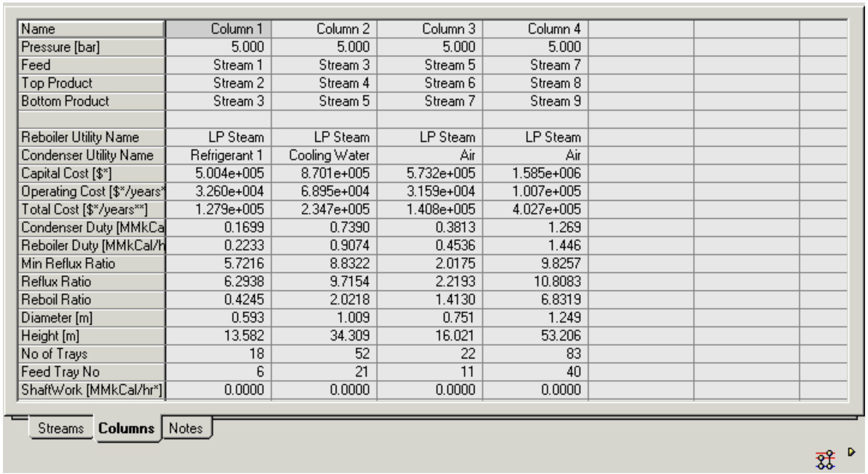
Streams
Products
Intermediates
Feed
All

Name	Stream 2	Stream 4	Stream 6	Stream 8	Stream 9
Vapour Fraction	0.0000	0.0000	0.0000	0.0000	0.0000
Pressure [bar]	5.000	5.000	5.000	5.000	5.000
Temperature [C]	9.010	42.76	55.83	87.40	114.0
Flow Rate [Molar	5.500	15.00	24.25	20.25	35.00
Meet Specs	✓	✓	✓	✓	✓
C3	0.8636	0.0167	0.0000	0.0000	0.0000
i-C4	0.1364	0.9000	0.0309	0.0000	0.0000
n-C4	0.0000	0.0833	0.9278	0.0617	0.0000
i-C5	0.0000	0.0000	0.0412	0.8889	0.0286
n-C5	0.0000	0.0000	0.0000	0.0494	0.5429
C6	0.0000	0.0000	0.0000	0.0000	0.2857
C7	0.0000	0.0000	0.0000	0.0000	0.1429

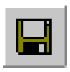
Streams | Columns | Notes

3. In the Worksheet pane, click the **Columns** tab. This tab contains information for all the columns in the selected column sequence design.

Figure 5.19



Name	Column 1	Column 2	Column 3	Column 4
Pressure [bar]	5.000	5.000	5.000	5.000
Feed	Stream 1	Stream 3	Stream 5	Stream 7
Top Product	Stream 2	Stream 4	Stream 6	Stream 8
Bottom Product	Stream 3	Stream 5	Stream 7	Stream 9
Reboiler Utility Name	LP Steam	LP Steam	LP Steam	LP Steam
Condenser Utility Name	Refrigerant 1	Cooling Water	Air	Air
Capital Cost [$*]	5.004e+005	8.701e+005	5.732e+005	1.585e+006
Operating Cost [$*/years*	3.260e+004	6.895e+004	3.159e+004	1.007e+005
Total Cost [$*/years**]	1.279e+005	2.347e+005	1.408e+005	4.027e+005
Condenser Duty [MMkCa	0.1699	0.7390	0.3813	1.269
Reboiler Duty [MMkCal/h	0.2233	0.9074	0.4536	1.446
Min Reflux Ratio	5.7216	8.8322	2.0175	9.8257
Reflux Ratio	6.2938	9.7154	2.2193	10.8083
Reboil Ratio	0.4245	2.0218	1.4130	6.8319
Diameter [m]	0.593	1.009	0.751	1.249
Height [m]	13.582	34.309	16.021	53.206
No of Trays	18	52	22	83
Feed Tray No	6	21	11	40
ShaftWork [MMkCal/hr*]	0.0000	0.0000	0.0000	0.0000



Save Case icon

4. Click the **Save Case** icon to save the case.

From the column sequence in Figure 5.17, the optimum sequence performs a *A/BCDE* split first and moves the difficult split (*D/E*) until the end of the column sequence.

5.5.2 Columns with Constant Top Temperature

The second scenario considers a constant top temperature of 35°C for all the columns in the column sequence designs. By limiting the top temperature for all columns to a specific value, the generated design will use only one type of utility to cool/condense the vapours in the column. For this scenario, the cooling utility for the condensers in all the column is limited to cooling water.

Cloning the Constant Pressure Scenario

Rather than re-specify all the information regarding the feed and product stream, you can save time by cloning the Constant Pressure scenario.

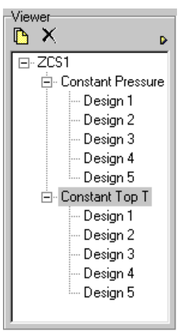
1. Select **Constant Pressure** in the Viewer group.
2. Do one of the following:
   - Right-click on the **Constant Pressure** scenario to open the Object Inspect menu and select the **Clone Scenario** command.
   - Click the **Clone Scenario/Design** icon.

Clone Scenario/Design icon

The Clone view will appear.

Figure 5.20

3. In the **New Name** field, enter **Constant Top T**, then press **ENTER**.
4. The clone operation will clone the **Constant Pressure** scenario and all designs within the scenario.



Viewer group

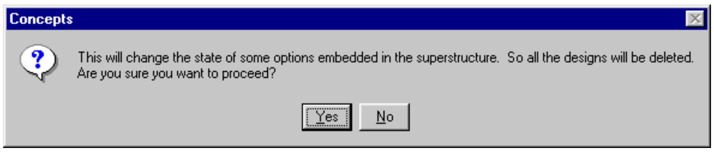
Limiting Designs to Constant Top Temperature

Now that you have a clone of the Constant Pressure scenario, you can lock/fix the top temperature value for all the columns and re-generate/calculate possible column sequence designs.

1. Select the **Constant Top T** in the Viewer group.
2. Click the **Task** tab, then select the **Classification** page.
3. In the **Number of Instances** field, enter **1** to indicate only one variable will be specified/modified.
4. A warning view will appear.

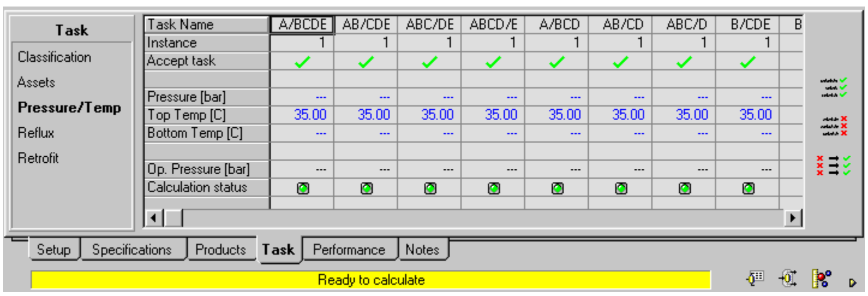
This view informs you that since the conditions of the selected scenario have changed, any designs in the selected scenario will be deleted.

Figure 5.21

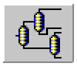


5. Click the **Yes** button to confirm that you want to delete the old designs. The warning view will automatically close.
6. In the Task classification based on group, select the **Instance Conditions** radio button.
7. In the Instances group, enter **35°C** in the cell under the **Top Temperature** column.
8. Click the **Populate Task Operating Conditions** button to place 35°C in all the possible column splits.
9. Click the **Pressure/Temp** page and confirm that any columns generated will have the top temperature value of **35°C**.

Figure 5.22



The other assumptions/conditions like Total Annualized Cost (TAC) for the objective function remains the same, so the Constant Top T scenario is ready to generate possible column sequences based on the new conditions/limitations.



Press to generate feasible column sequences icon

10. Click the **Press to generate feasible column sequences** icon in the Main pane to begin calculation for the new designs.

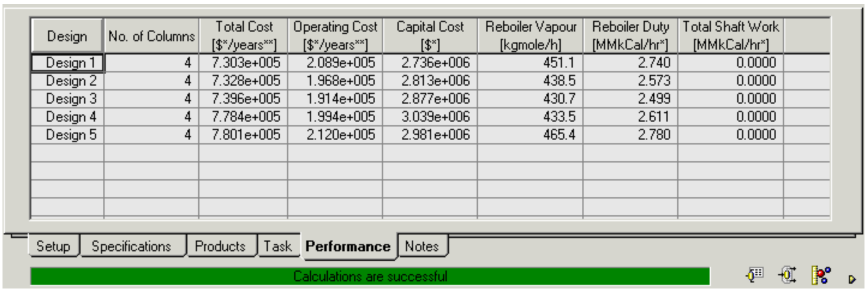
Generated Feasible Designs

The designs are once again sorted in ascending order of the objective function value, with the optimum design as the very first design.

The new designs based on the limitation that all columns had to use cooling water in the condenser are listed under the Constant Top T scenario.

At the Scenario level, the Performance tab displays summary information of every design in the scenario.

Figure 5.23

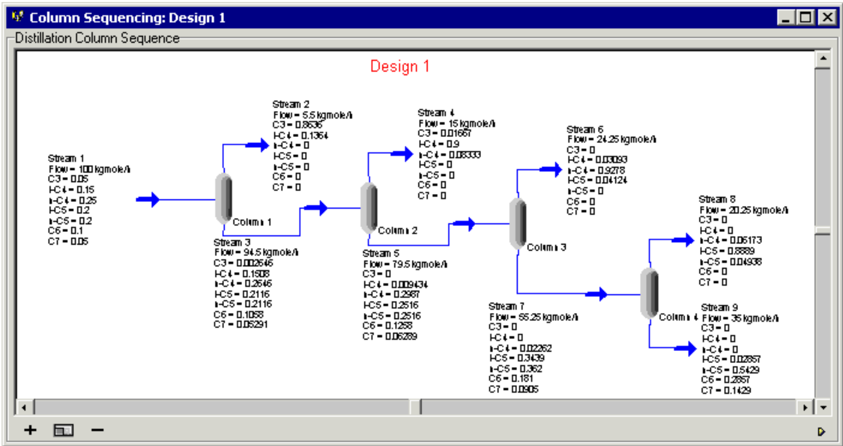


Design	No. of Columns	Total Cost [$*/years**]	Operating Cost [$*/years**]	Capital Cost [$*]	Reboiler Vapour [kgmole/h]	Reboiler Duty [MMkCal/hr*]	Total Shaft Work [MMkCal/hr*]
Design 1	4	7.303e+005	2.089e+005	2.736e+006	451.1	2.740	0.0000
Design 2	4	7.328e+005	1.968e+005	2.813e+006	438.5	2.573	0.0000
Design 3	4	7.396e+005	1.914e+005	2.877e+006	430.7	2.499	0.0000
Design 4	4	7.784e+005	1.994e+005	3.039e+006	433.5	2.611	0.0000
Design 5	4	7.801e+005	2.120e+005	2.981e+006	465.4	2.780	0.0000

To view detailed information about the designs, select the design name in the Viewer group, then click the Streams tab and Columns tab to view the information.

The Main pane displays the PFD for the selected design.

Figure 5.24



The **Streams** tab, in the Worksheet pane, contains the information regarding all the streams in the selected column sequence design.

Figure 5.25

Streams
Products
Intermediates
Feed
All

Name	Stream 2	Stream 4	Stream 6	Stream 8	Stream 9
Vapour Fraction	0.0000	0.0000	0.0000	0.0000	0.0000
Pressure [bar]	9.153	4.130	2.963	1.324	1.324
Temperature [C]	34.85	34.85	34.85	34.85	57.73
Flow Rate [Molar	5.500	15.00	24.25	20.25	35.00
Meet Specs	✓	✓	✓	✓	✓
C3	0.8636	0.0167	0.0000	0.0000	0.0000
i-C4	0.1364	0.9000	0.0309	0.0000	0.0000
n-C4	0.0000	0.0833	0.9278	0.0617	0.0000
i-C5	0.0000	0.0000	0.0412	0.8889	0.0286
n-C5	0.0000	0.0000	0.0000	0.0494	0.5429
C6	0.0000	0.0000	0.0000	0.0000	0.2857
C7	0.0000	0.0000	0.0000	0.0000	0.1429

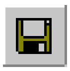
Streams Columns Notes

The **Columns** tab, in the Worksheet pane, contains information for all the columns in the selected column sequence design.

Figure 5.26

Name	Column 1	Column 2	Column 3	Column 4
Pressure [bar]	9.153	4.130	2.963	1.324
Feed	Stream 1	Stream 3	Stream 5	Stream 7
Top Product	Stream 2	Stream 4	Stream 6	Stream 8
Bottom Product	Stream 3	Stream 5	Stream 7	Stream 9
Reboiler Utility Name	LP Steam	LP Steam	LP Steam	LP Steam
Condenser Utility Name	Cooling Water	Cooling Water	Cooling Water	Cooling Water
Capital Cost [$*]	4.562e+005	8.396e+005	5.043e+005	9.366e+005
Operating Cost [$*/years*	1.851e+004	6.711e+004	3.339e+004	8.989e+004
Total Cost [$*/years**]	1.054e+005	2.271e+005	1.295e+005	2.684e+005
Condenser Duty [MMkCa	0.1848	0.7226	0.3695	1.049
Reboiler Duty [MMkCal/h	0.2452	0.8829	0.4382	1.173
Min Reflux Ratio	6.8469	8.4345	1.7968	7.0762
Reflux Ratio	7.5316	9.2780	1.9765	7.7838
Reboil Ratio	0.4965	1.9392	1.3064	5.0821
Diameter [m]	0.619	1.008	0.762	1.286
Height [m]	15.411	32.480	14.802	36.747
No of Trays	21	49	20	56
Feed Tray No	7	20	10	27
ShaftWork [MMkCal/hr*]	0.0000	0.0000	0.0000	0.0000

Streams | Columns | Notes



Save Case icon

11. Click the **Save Case** icon to save the case.

From the column sequence in **Figure 5.24**, the optimum sequence performs a *A/BCDE* split first and keeps the difficult split (*D/E*) until the end of the column sequence.

5.5.3 Optimum Top Temperature

The third scenario considers three possible top temperature values (20°C, 30°C, and 40°C) and allows DISTIL to select the optimum temperature value (one of the three temperatures) for each column in the design.

Cloning Constant Top T Scenario

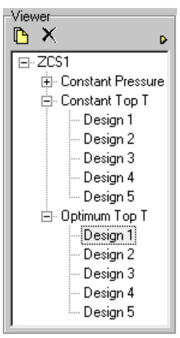
The third scenario will be cloned from the Constant Top T scenario.

1. Select **Constant Top T** in the Viewer group.
2. Do one of the following:
   - Click the **Clone Scenario/Design** icon.
   - Right-click the **Constant Top T** scenario name to open the Object Inspect menu, then select the **Clone Scenario** command.
3. The Clone view will appear.

Clone Scenario/Design icon

Figure 5.27

4. In the Name field, enter **Optimum Top T**, then press **ENTER**.
5. The clone operation will clone **Constant Top T** and the designs within the scenario.



Viewer group

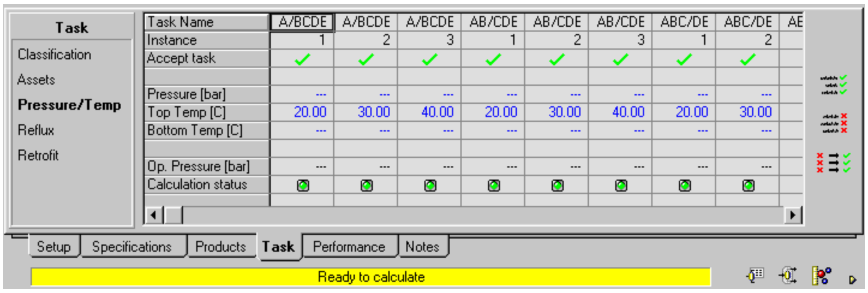
Finding the Optimum Top Temperature

Now that you have cloned the Constant Top T scenario, you can specify the three possible top temperature values for the columns and allow DISTIL to calculate and generate new column sequence designs with optimum top temperatures.

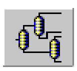
1. Select **Optimum Top T** in the Viewer group.
2. Click the **Task** tab, then select the **Classification** page.
3. In the **Number of Instances** field, enter **3** to indicate that three variables will be specified/modified.

4. A warning view will appear.
   The warning view informs you that since the conditions of the selected scenario have changed, any designs in the selected scenario will be deleted.
5. Click the **Yes** button to confirm that you want to delete the old designs. The warning view will automatically close.
6. In the Task classification based on group, select the **Instance Conditions** radio button.
7. In the Instances table, enter **20°C**, **30°C**, and **40°C** in the cells under the **Top Temperature** column.
8. In the Instances group, click the **Populate Task Operating Conditions** button to place the three temperatures in all the possible column splits.
9. Click the **Pressure/Temp** page and confirm that any columns generated will have one of the three specified top temperature values.

Figure 5.28



The other assumptions/conditions like Total Annualized Cost (TAC) for the objective function remains the same, so the Optimum Top T scenario is ready to generate possible column sequences based on the new conditions/limitations.



Press to generate feasible column sequences icon

10. Click the **Press to generate feasible column sequences** icon in the Main pane to begin calculation for the new designs.

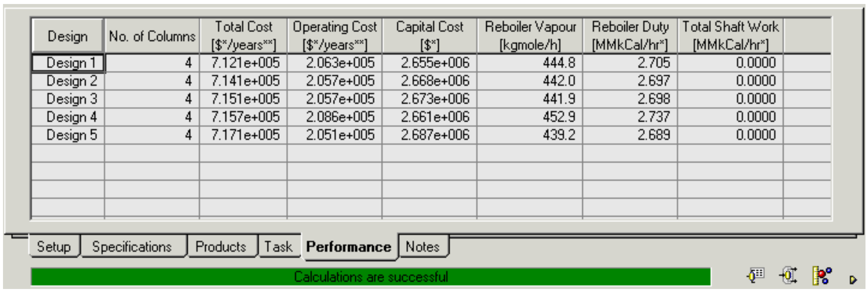
Generated Feasible Designs

The designs are sorted in ascending order of the objective function value, with the optimum design as the very first design.

The new designs based on the limitation that all columns had to use cooling water in the condenser are listed under the Constant Top T scenario.

The Performance tab at the Scenario level contains summary information for every design.

Figure 5.29

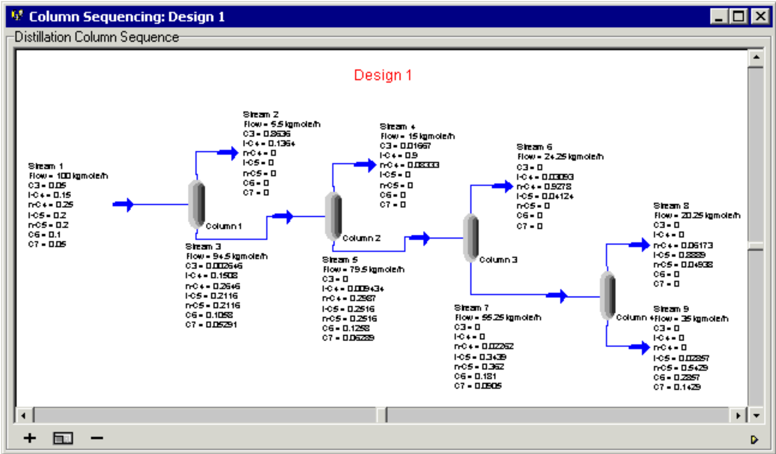


Design	No. of Columns	Total Cost [$*/years**]	Operating Cost [$*/years**]	Capital Cost [$*]	Reboiler Vapour [kgmole/h]	Reboiler Duty [MMkCal/hr*]	Total Shaft Work [MMkCal/hr*]
Design 1	4	7.121e+005	2.063e+005	2.655e+006	444.8	2.705	0.0000
Design 2	4	7.141e+005	2.057e+005	2.668e+006	442.0	2.697	0.0000
Design 3	4	7.151e+005	2.057e+005	2.673e+006	441.9	2.698	0.0000
Design 4	4	7.157e+005	2.086e+005	2.661e+006	452.9	2.737	0.0000
Design 5	4	7.171e+005	2.051e+005	2.687e+006	439.2	2.689	0.0000

To view detail information about the designs, select the design name in the Viewer group, then click the Streams tab and Columns tab to view the information.

The Main pane will display the PFD of the selected design.

Figure 5.30



The **Streams** tab, in the Worksheet pane, contains the information for all the streams in the selected column sequence design.

Figure 5.31

Streams: **Products** | Intermediates | Feed | All

Name	Stream 2	Stream 4	Stream 6	Stream 8	Stream 9
Vapour Fraction	0.0000	0.0000	0.0000	0.0000	0.0000
Pressure [bar]	10.13	3.635	3.385	1.131	1.131
Temperature [C]	39.85	29.85	39.85	29.85	52.38
Flow Rate [Molar	5.500	15.00	24.25	20.25	35.00
Meet Specs	✓	✓	✓	✓	✓
C3	0.8636	0.0167	0.0000	0.0000	0.0000
i-C4	0.1364	0.9000	0.0309	0.0000	0.0000
n-C4	0.0000	0.0833	0.9278	0.0617	0.0000
i-C5	0.0000	0.0000	0.0412	0.8889	0.0286
n-C5	0.0000	0.0000	0.0000	0.0494	0.5429
C6	0.0000	0.0000	0.0000	0.0000	0.2857
C7	0.0000	0.0000	0.0000	0.0000	0.1429

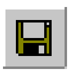
Tabs: **Streams** | Columns | Notes

The **Columns** tab, in the Worksheet pane, contains the information for all the columns in the selected column sequence design.

Figure 5.32

Name	Column 1	Column 2	Column 3	Column 4
Pressure [bar]	10.13	3.635	3.385	1.131
Feed	Stream 1	Stream 3	Stream 5	Stream 7
Top Product	Stream 2	Stream 4	Stream 6	Stream 8
Bottom Product	Stream 3	Stream 5	Stream 7	Stream 9
Reboiler Utility Name	LP Steam	LP Steam	LP Steam	LP Steam
Condenser Utility Name	Cooling Water	Cooling Water	Cooling Water	Cooling Water
Capital Cost [$*]	4.399e+005	8.287e+005	4.879e+005	8.999e+005
Operating Cost [$*/years*	1.878e+004	6.592e+004	3.370e+004	8.788e+004
Total Cost [$*/years**]	1.026e+005	2.238e+005	1.267e+005	2.594e+005
Condenser Duty [MMkCa	0.1873	0.7119	0.3726	1.027
Reboiler Duty [MMkCal/h	0.2488	0.8670	0.4422	1.147
Min Reflux Ratio	7.0864	8.1902	1.8482	6.8446
Reflux Ratio	7.7950	9.0092	2.0331	7.5290
Reboil Ratio	0.5119	1.8885	1.3313	4.9346
Diameter [m]	0.627	1.009	0.758	1.300
Height [m]	15.411	31.261	14.802	35.528
No of Trays	21	47	20	54
Feed Tray No	7	19	10	26
ShaftWork [MMkCal/hr*]	0.0000	0.0000	0.0000	0.0000

Streams | Columns | Notes



Save Case icon

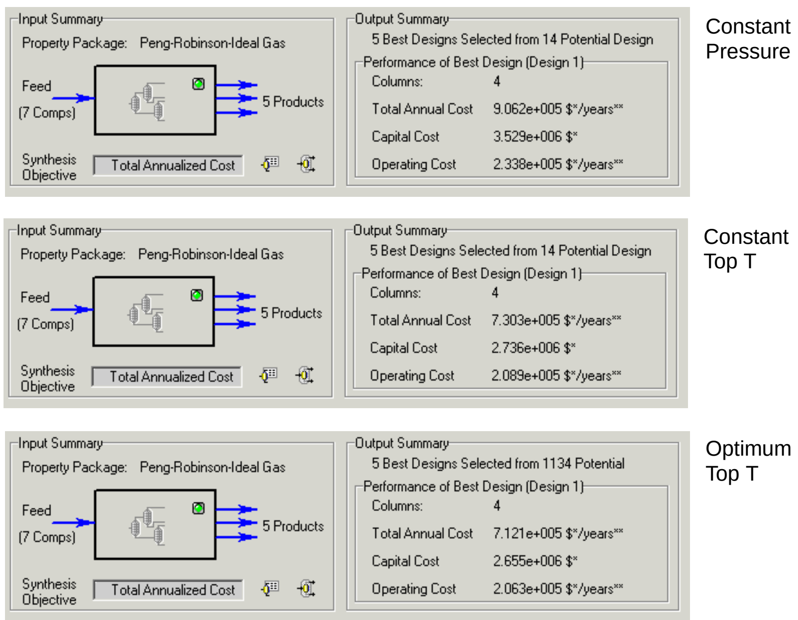
11. Click the **Save Case** icon to save the case.

5.6 Comparing Grass Root Designs

Before examining the fourth scenario, retrofitted designs, you will compare the best design from each previous scenario and select the column sequence design that best suits your plant.

1. Select the **Constant Pressure** scenario name in the Viewer group. In the Main pane, the Output Summary group displays the summary cost of the best design in the scenario (first design).
2. Click the scenario name for the other two scenarios you created. The figure below displays the Main pane for all three scenarios.

Figure 5.33



The following table lists and compares the total annualized cost (TAC) of the best design from each of the three scenarios:

Scenario	Rank	TAC ($/year)	Relative (%)
Constant Pressure	3	$9.062*10^5$	100
Constant Top T	2	$7.303*10^5$	80.6
Optimum Top T	1	$7.121*10^5$	78.6

5.7 Generating a Retrofit Scenario

The last scenario we will examine is retrofitting an existing separation system. As mentioned in **Section 5.1 - Introduction**, the existing separation system contains four columns with the following number of trays: **20**, **65**, **30**, and **75**.

5.7.1 Cloning an Existing Scenario

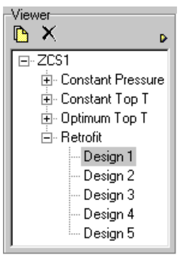
The forth scenario will be cloned from the Constant Top T scenario.

1. Select **Constant Top T** in the Viewer group.
2. Do one of the following:

Clone Scenario/Design icon

- Click the **Clone Scenario/Design** icon.
- Right-click the **Constant Top T** scenario name to open the Object Inspect menu, then select the **Clone Scenario** command.

3. The Clone view will appear.
4. In the New Name field, enter **Retrofit**, then press **ENTER**.
5. The clone operation will clone **Constant Top T** and the designs within the scenario.



Viewer group

5.7.2 Remove Temperature Constraints

Now that you have a clone of the Constant Top T scenario, you can specify the number of columns and trays allowed in the retrofit designs. Before specifying the limited number of trays, you must remove the temperature constraints on the top column temperature.

1. Select **Retrofit** in the Viewer group.
2. In the Worksheet pane, click the **Task** tab, then select the **Classification** page.
3. In the **Number of Instances** field, enter **1** to clear the value in the Instances group.
4. A warning view will appear.
5. Click the **Yes** button to confirm that you want to delete the old designs. The warning view will automatically close.

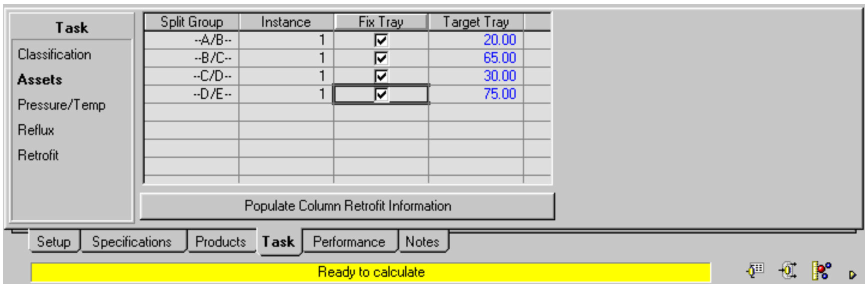
The warning view informs you that since the conditions of the selected scenario have changed, any designs in the selected scenario will be deleted.

5.7.3 Specify the Existing Number of Trays

The objective function of the retrofit scenario is operating cost, not total annualized cost (TAC). TAC takes into account both operating and capital cost, but since you do not have capital expense (the hardware is already there) you want to focus on reducing the operating cost.

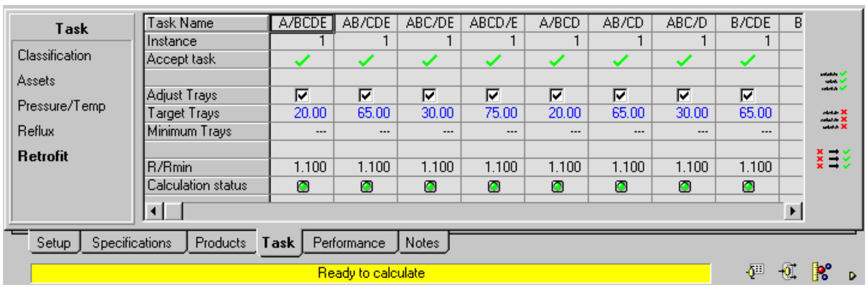
1. Click the **Setup** tab, then select the **Options** page.
2. In the Objective Function group, select **Operating Cost** from the drop-down list.
3. Click the **Task** tab, then select the **Assets** page.
4. Enter the information shown in the picture below, and check all the checkboxes under the **Fix Tray** column.

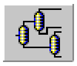
Figure 5.34



5. Click the **Populate Column Retrofit Information** button.
6. Select the **Retrofit** page and confirm that all possible splits contains only one of the four possible tray numbers.

Figure 5.35





Press to generate feasible column sequences icon

7. Click the **Press to generate feasible column sequences** icon in the Main pane to begin calculation for the new designs.

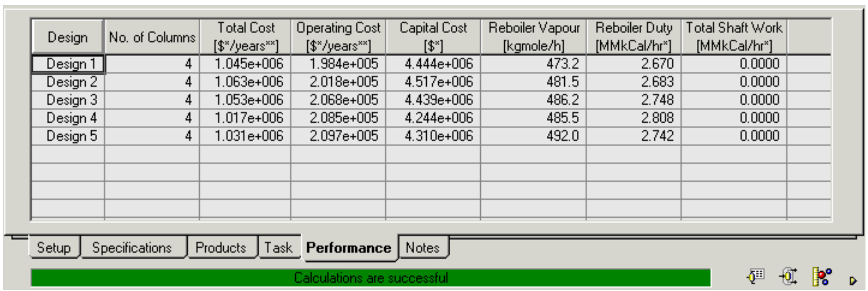
5.7.4 Generated Feasible Designs

The designs are once again sorted in ascending order of the *objective function* value, with the optimum design as the very first design.

The new designs based on the limitation that columns in the column sequence design are from existing columns are listed under the Retrofit scenario.

The Performance tab displays summary information for every design in the scenario.

Figure 5.36

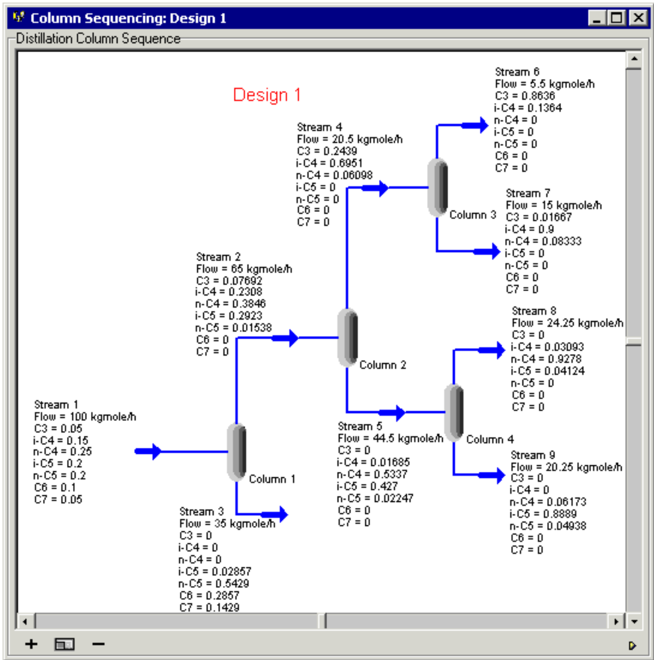


Design	No. of Columns	Total Cost [$*/years**]	Operating Cost [$*/years**]	Capital Cost [$*]	Reboiler Vapour [kgmole/h]	Reboiler Duty [MMkCal/hr*]	Total Shaft Work [MMkCal/hr*]
Design 1	4	1.045e+006	1.984e+005	4.444e+006	473.2	2.670	0.0000
Design 2	4	1.063e+006	2.018e+005	4.517e+006	481.5	2.683	0.0000
Design 3	4	1.053e+006	2.068e+005	4.439e+006	486.2	2.748	0.0000
Design 4	4	1.017e+006	2.085e+005	4.244e+006	485.5	2.808	0.0000
Design 5	4	1.031e+006	2.097e+005	4.310e+006	492.0	2.742	0.0000

To view detail information about the designs, select the design name in the Viewer group, then click the Streams tab and Column tab.

The Main pane will display the PFD of the selected design.

Figure 5.37



The **Streams** tab, in the Worksheet pane, contains the information for all the streams in the selected column sequence design.

Figure 5.38

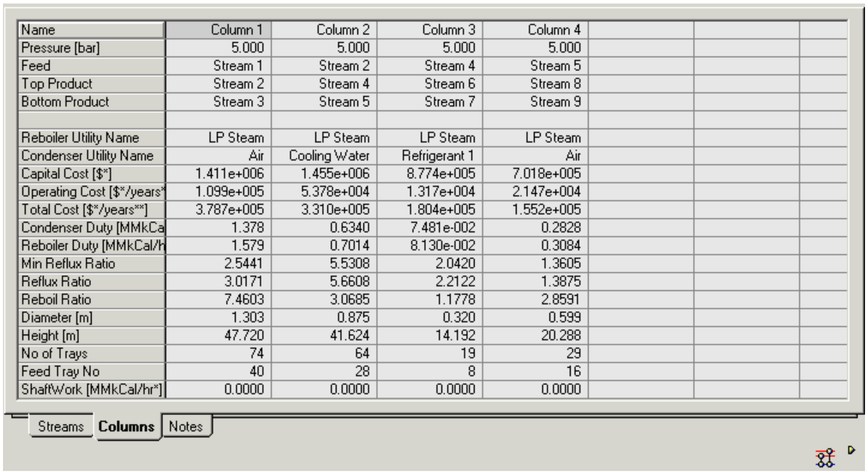
Streams: **Products** | Intermediates | Feed | All

Name	Stream 3	Stream 6	Stream 7	Stream 8	Stream 9
Vapour Fraction	0.0000	0.0000	0.0000	0.0000	0.0000
Pressure [bar]	5.000	5.000	5.000	5.000	5.000
Temperature [C]	114.0	9.010	42.76	55.83	87.40
Flow Rate [Molar	35.00	5.500	15.00	24.25	20.25
Meet Specs	✓	✓	✓	✓	✓
C3	0.0000	0.8636	0.0167	0.0000	0.0000
i-C4	0.0000	0.1364	0.9000	0.0309	0.0000
n-C4	0.0000	0.0000	0.0833	0.9278	0.0617
i-C5	0.0286	0.0000	0.0000	0.0412	0.8889
n-C5	0.5429	0.0000	0.0000	0.0000	0.0494
C6	0.2857	0.0000	0.0000	0.0000	0.0000
C7	0.1429	0.0000	0.0000	0.0000	0.0000

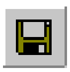
Tabs: **Streams** | Columns | Notes

The **Columns** tab, in the Worksheet pane, contains the information for all the columns in the selected column sequence design.

Figure 5.39



Name	Column 1	Column 2	Column 3	Column 4
Pressure [bar]	5.000	5.000	5.000	5.000
Feed	Stream 1	Stream 2	Stream 4	Stream 5
Top Product	Stream 2	Stream 4	Stream 6	Stream 8
Bottom Product	Stream 3	Stream 5	Stream 7	Stream 9
Reboiler Utility Name	LP Steam	LP Steam	LP Steam	LP Steam
Condenser Utility Name	Air	Cooling Water	Refrigerant 1	Air
Capital Cost [$*]	1.411e+006	1.455e+006	8.774e+005	7.018e+005
Operating Cost [$*/years*	1.099e+005	5.378e+004	1.317e+004	2.147e+004
Total Cost [$*/years**]	3.787e+005	3.310e+005	1.804e+005	1.552e+005
Condenser Duty [MMkCa	1.378	0.6340	7.481e-002	0.2828
Reboiler Duty [MMkCal/h	1.579	0.7014	8.130e-002	0.3084
Min Reflux Ratio	2.5441	5.5308	2.0420	1.3605
Reflux Ratio	3.0171	5.6608	2.2122	1.3875
Reboil Ratio	7.4603	3.0685	1.1778	2.8591
Diameter [m]	1.303	0.875	0.320	0.599
Height [m]	47.720	41.624	14.192	20.288
No of Trays	74	64	19	29
Feed Tray No	40	28	8	16
ShaftWork [MMkCal/hr*]	0.0000	0.0000	0.0000	0.0000



Save Case icon

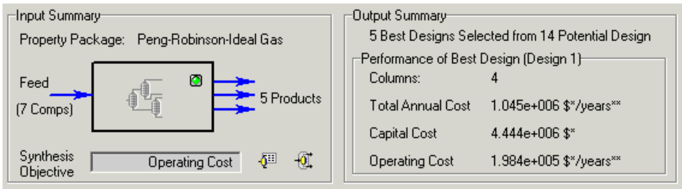
8. Click the **Save Case** icon to save the case.

5.8 Comparing Grass Root and Retrofit Designs

Now that you have information for the best retrofit design, you can compare the grass root scenarios with the retrofit scenario.

1. Select the **Retrofit** scenario in the Viewer group.
   In the Main pane, the Output Summary group displays the summary cost of the best design in the scenario (first design).
   The figure below displays the Main pane for the selected scenario.

Figure 5.40



2. Click the scenario name for the grass root scenarios you created.

The following table lists and compares the operating cost and capital expense of the best design from each of the four scenarios.

Scenario	Rank	Operating Cost ($/year)	Relative (%)	Capital Cost ($)
Constant Pressure	4	$2.338*10^5$	100	$3.529*10^6$
Constant Top T	3	$2.089*10^5$	89.3	$2.736*10^6$
Optimum Top T	2	$2.063*10^5$	88.2	$2.655*10^6$
Retrofit	1	$1.984*10^5$	84.9	0

The Retrofit scenario ranked 1st in the operating cost and first for the capital cost. The other three scenarios require a capital investment, while the Retrofit scenario requires little or no capital investment.

6 Benzene Removal

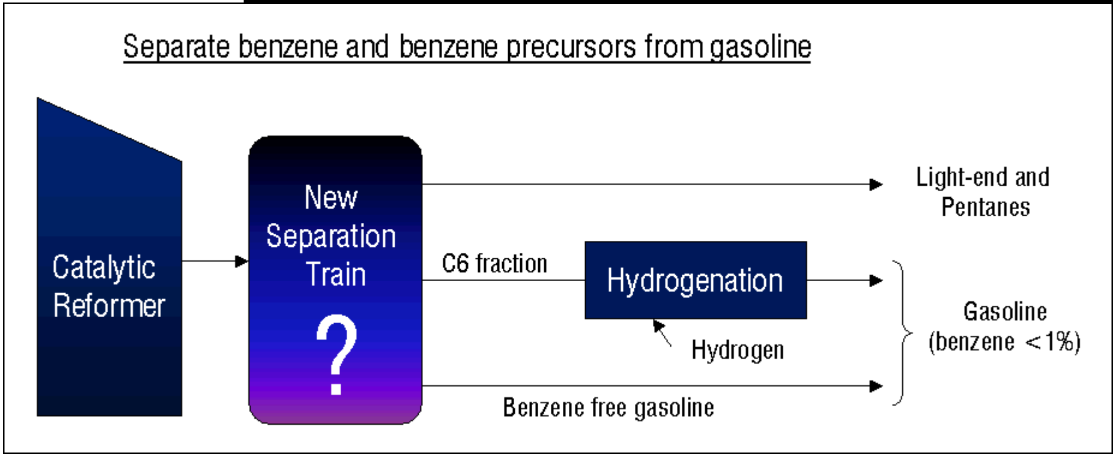
6.1 Introduction

This tutorial serves as an introduction to the DISTIL Three Product System operation.

In this tutorial, you will use DISTIL to help you choose the best column configuration to remove benzene from gasoline in a refinery process. The Clean Air Act states that “...benzene content in gasoline must be less than 1% by volume...”

The benzene content in the gasoline exiting the Separation Train is assumed to be 15%. One option is to separate the benzene and benzene precursors from the gasoline, and send only the benzene and benzene precursors into the Hydrogenation. In this case, a small amount of hydrogen is required in the Hydrogenation.

Figure 6.1



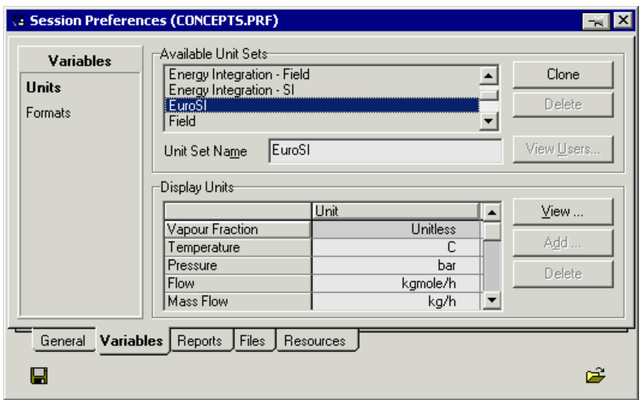
There are many possible column designs for the above separation system. DISTIL Three Product System (TPS) operation offers nine different complex column designs from which you can check and compare the designs to select the column configuration that meets your requirements.

6.2 Setting Unit Preferences

Before you begin the simulation, you will verify that the units currently selected in the DISTIL preferences are the ones you want to use. For this example, the temperatures are in Celsius and the cost per time is in Cost/year.

1. Start DISTIL if it is not already open.
2. From the **Tools** menu, select **Preferences**. The Session Preferences view appears.
3. Click the **Variables** tab, then select the **Units** page.

Figure 6.2



4. Select **SI** from the Available Unit Sets group.

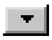
The text in the Unit column of the Display Units table are black. This indicates that you cannot modify the units associated with the variables.

The SI unit set is a default unit set and you cannot modify the units associated with the variables. You have to clone the default unit set, and modify the cloned unit set. You will do this in the next section.

Cloning a Default Unit Set

To clone a default unit set:

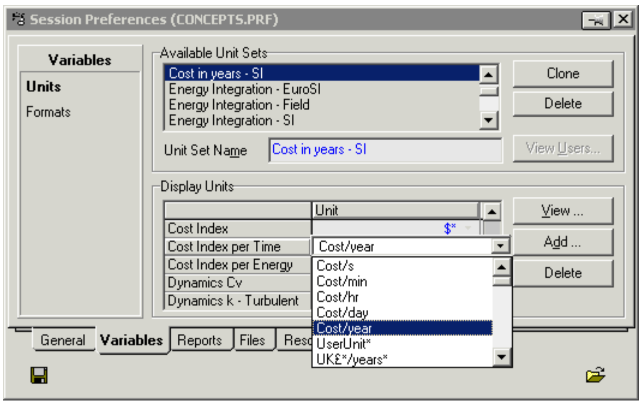
1. In the Available Unit Sets group, select the **SI** unit set and click the **Clone** button. A new unit set appears called New User.
2. In the Unit Set **Name** field, rename the New User unit set **Cost in years - SI**.
3. In the Display Units table, scroll and find the **Cost Index per Time** cell.



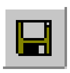
Down arrow icon

4. Click the down arrow in the Cost Index per Time cell, and select **Cost/year** from the drop-down list.

Figure 6.3



5. Repeat step #3 and #4 to change the **Energy** variable to **kW**.
6. Optional: At this point you can save the modified-cloned unit set in a new preference file, which will allow you to use the modified-cloned unit set in future cases.
   To save the preference file, click the **Save Preference File** icon. On the Save Preference File view, enter a file name and location, and click the **Save** button.



Save Preference File icon

Although you can overwrite the default preference set included with DISTIL, it is not recommended.

7. Click the **Close** icon  to close the Session Preferences view when you have completed the unit set modification.

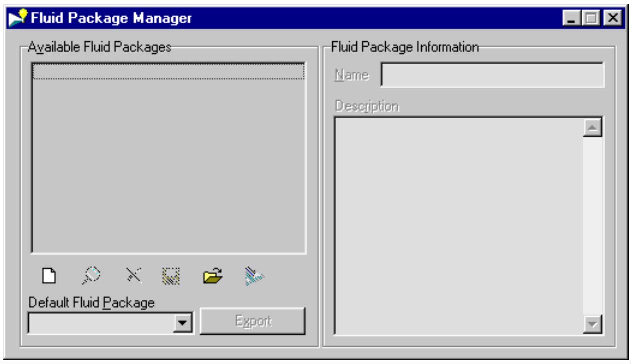
6.3 Defining the Fluid Package

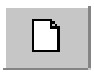
To use the Three Product System operation, you need to define a fluid package containing the property package and components required to simulate the feed stream entering the separation system.

Fluid Package Manager icon

1. Open the Fluid Package Manager view by clicking the **Fluid Package Manager** icon.

Figure 6.4

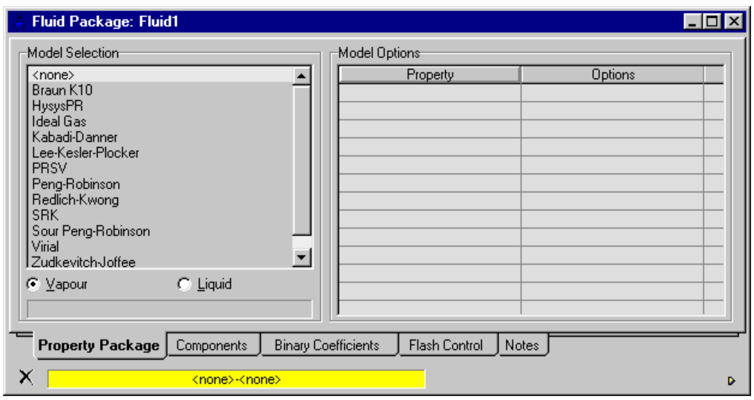




Add Fluid Package icon

2. In the Fluid Package Manager view, click the **Add Fluid Package** icon.
   The Fluid Package view will appear.

Figure 6.5



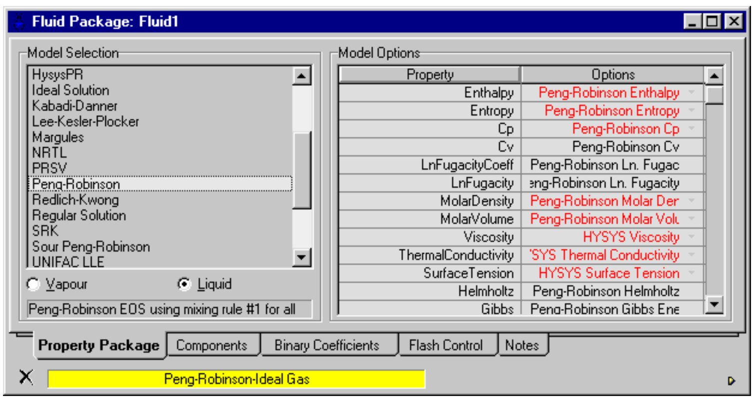
6.3.1 Selecting a Property Package

In the Fluid Package view, you will select the property package and components associated with the fluid package.

1. In the Fluid Package view, click the **Property Package** tab.
2. In the Model Selection group, select the **Vapour** radio button, then select **Ideal Gas** in the Model Selection list.
3. In the Model Selection group, select the **Liquid** radio button, then select **Peng-Robinson** in the Model Selection list.

The status bar at the bottom of the Fluid Package view should display Peng-Robinson-Ideal Gas as the selected property package, as shown in the figure below:

Figure 6.6



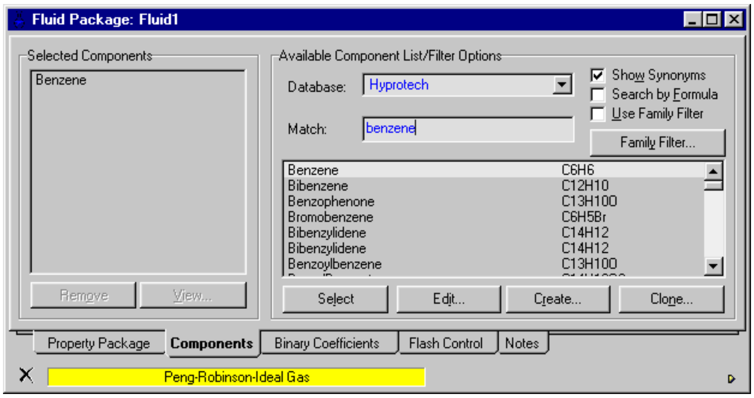
6.3.2 Selecting the Components

1. Click the **Components** tab.
2. In the Available Component List/Filter Options group, type **benzene** in the Match field.
   The list under the Match field will display all components that have names with the word **benzene** in them.

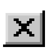
You can also press **ENTER** to add the selected component from the list into the Selected Components group.

3. Select **Benzene** from the list and click the **Select** button.
   The component benzene will be added to the Selected Components list.

Figure 6.7

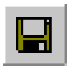


4. Repeat steps #2 to #3 to add the following components:
   - Toluene
   - i-C5
   - i-C4
   - n-C4
   - n-C5
   - 2-MC5
   - C6
   - C7
   - 124-M-BZ
   - m-Xylene
5. Click the **Notes** tab. In the **Name** field, enter a name for the fluid package (e.g., **PR**).



Close icon

6. Click the **Close** icon to close the Fluid Package view.

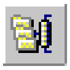


Save Case icon

7. Save the case as **BenzeneRemoval**. DISTIL will save the file with the ***.hcd** extension.

6.4 Creating a Three Product System

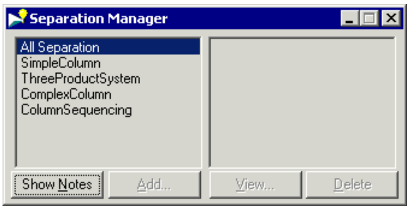
After specifying the fluid package for the simulation, you are ready to use the Three Product System operation.



Separation Technology Manager icon

1. Click the **Separation Technology Manager** icon to open the Separation Manager view.

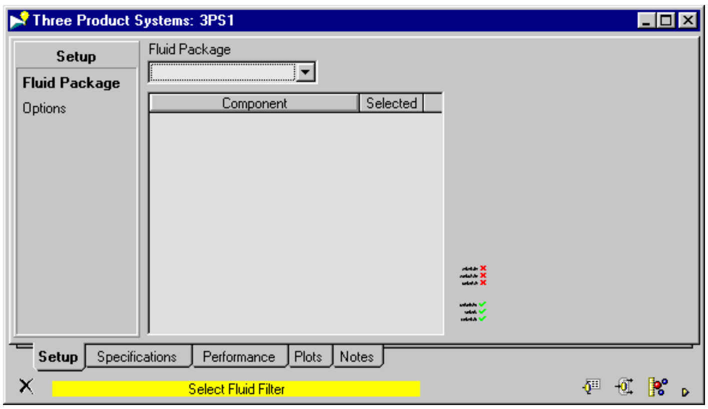
Figure 6.8



2. In the left list of the Separation Manager view, select **ThreeProductSystem**.
3. Click the **Add** button. The TPS property view will appear. DISTIL automatically names the TPS operation **3PS1**.

The name of the TPS operation will appear in the right list of the Separation Manager view.

Figure 6.9

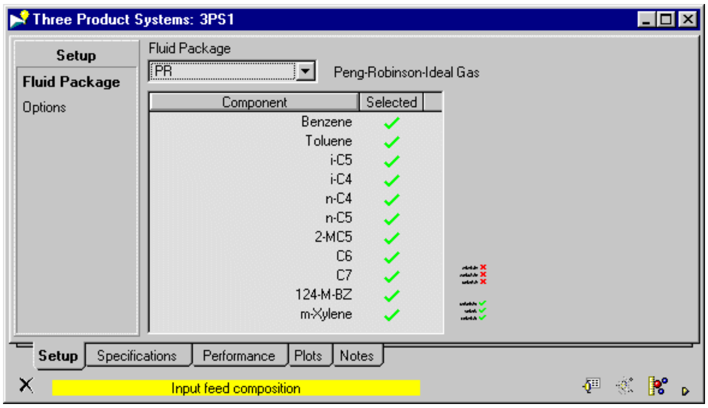


The status bar will also display an error message if there are any problems during calculations.

The status bar at the bottom of the TPS view indicates the status of the operation. If you have not specified the required information, the status bar will display a request for the missing information.

4. Click the **Setup** tab, then select the **Fluid Package** page.
5. On the **Fluid Package** page, use the **Fluid Package** drop-down list to select the fluid package for the simulation.

Figure 6.10

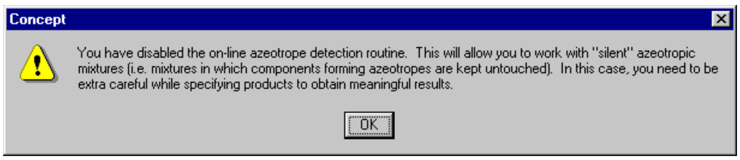


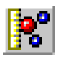
6.4.1 Modifying Column Configurations

In this tutorial, there are some limitations that need to be applied to the complex columns design calculations.

1. On the **Setup** tab, select the **Options** page.
2. In the Configurations group, ensure that all the checkboxes are checked.
3. In the Options group, check the **Disable Azeotrope Check** checkbox. The following warning message will appear:

Figure 6.11

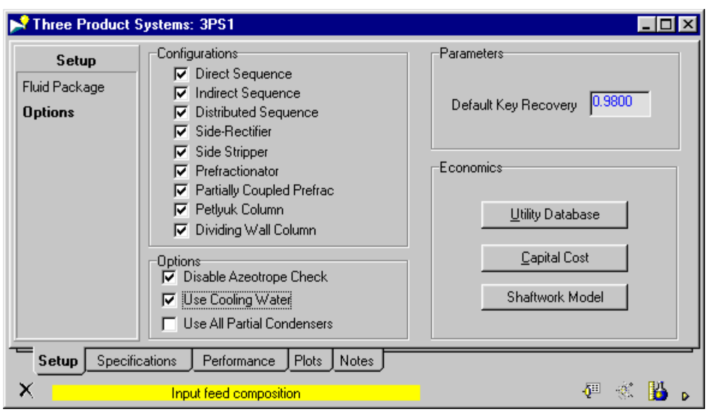




Molar Fraction Basis icon

4. Click the **OK** button.
5. Click the **Molar Fraction Basis** icon to open the Basis Selection view.
6. On the Basis Selection view, select the **LiqVol Fraction** radio button, then click the **Close** icon to close the Basis Selection view.
7. In the Options group, check the **Use Cooling Water** checkbox
8. In the **Default Key Recovery** field, enter **0.98**. The Options page should appear as shown in the figure below:

Figure 6.12

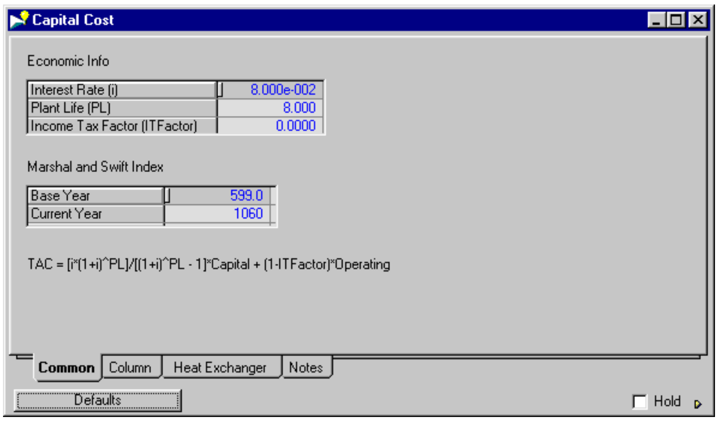


Modifying Capital Cost Calculation

In this section, you will modify the capital cost parameter values.

1. On the **Setup** tab, **Options** page, click the **Capital Cost** button in the Economics group.
   The Capital Cost view will appear:

Figure 6.13



2. In the **Common** tab, enter **1070** in the **Current Year** cell.

If you modify any settings other than the Current Year, the calculated results in the rest of this tutorial will differ from what you see on your screen.

You can modify the following parameters in the capital cost calculations, using the options in the Capital Cost view:

- **Column** tab: tray size, vessel thickness, vessel cost, tray cost, platform and ladder cost, and installation cost.
- **Heat Exchangers** tab: heat exchangers size, heat exchangers cost, and installation cost.

Close icon

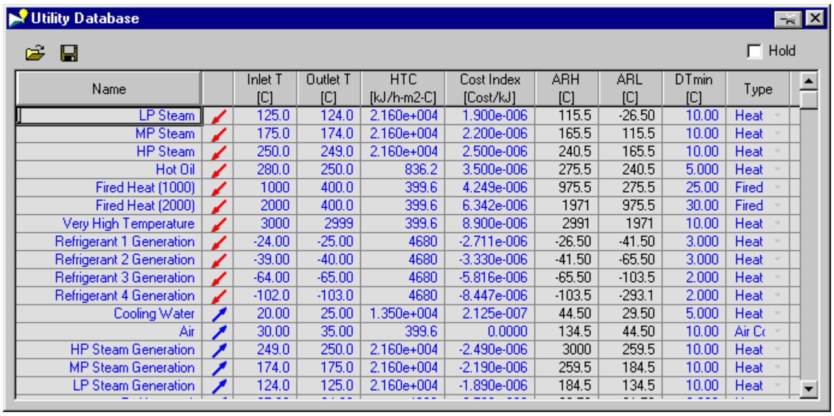
3. In this tutorial, the default values for the rest of the parameters are adequate. Click the **Close** icon to close the Capital Cost view.

Modifying the Utility Database

In this tutorial there is no Air utility, so you will remove the Air utility from the DISTIL default utility database.

1. On the **Setup** tab, **Options** page, click the **Utility Database** button in the Economics group.
   The Utility Database view will appear.

Figure 6.14



Name		Inlet T [C]	Outlet T [C]	HTC [kJ/h-m2-C]	Cost Index [Cost/kJ]	ARH [C]	ARL [C]	DTmin [C]	Type
LP Steam		125.0	124.0	2.160e+004	1.900e-006	115.5	-26.50	10.00	Heat
MP Steam		175.0	174.0	2.160e+004	2.200e-006	165.5	115.5	10.00	Heat
HP Steam		250.0	249.0	2.160e+004	2.500e-006	240.5	165.5	10.00	Heat
Hot Oil		280.0	250.0	836.2	3.500e-006	275.5	240.5	5.000	Heat
Fired Heat (1000)		1000	400.0	399.6	4.249e-006	975.5	275.5	25.00	Fired
Fired Heat (2000)		2000	400.0	399.6	6.342e-006	1971	975.5	30.00	Fired
Very High Temperature		3000	2999	399.6	8.900e-006	2991	1971	10.00	Heat
Refrigerant 1 Generation		-24.00	-25.00	4680	-2.711e-006	-26.50	-41.50	3.000	Heat
Refrigerant 2 Generation		-39.00	-40.00	4680	-3.330e-006	-41.50	-65.50	3.000	Heat
Refrigerant 3 Generation		-64.00	-65.00	4680	-5.816e-006	-65.50	-103.5	2.000	Heat
Refrigerant 4 Generation		-102.0	-103.0	4680	-8.447e-006	-103.5	-293.1	2.000	Heat
Cooling Water		20.00	25.00	1.350e+004	2.125e-007	44.50	29.50	5.000	Heat
Air		30.00	35.00	399.6	0.0000	134.5	44.50	10.00	Air C
HP Steam Generation		249.0	250.0	2.160e+004	-2.490e-006	3000	259.5	10.00	Heat
MP Steam Generation		174.0	175.0	2.160e+004	-2.190e-006	259.5	184.5	10.00	Heat
LP Steam Generation		124.0	125.0	2.160e+004	-1.890e-006	184.5	134.5	10.00	Heat

2. Select the **Air** cell in the **Name** column.
3. Press DELETE. DISTIL will remove **Air** from the utility database.
4. Click the **Close** icon to close the Utility Database view.

Close icon

6.4.2 Specifying Feed and Product Streams

Now that you have selected the fluid package and calculation options for the Three Product System operation, you will specify the feed and product streams characteristics.

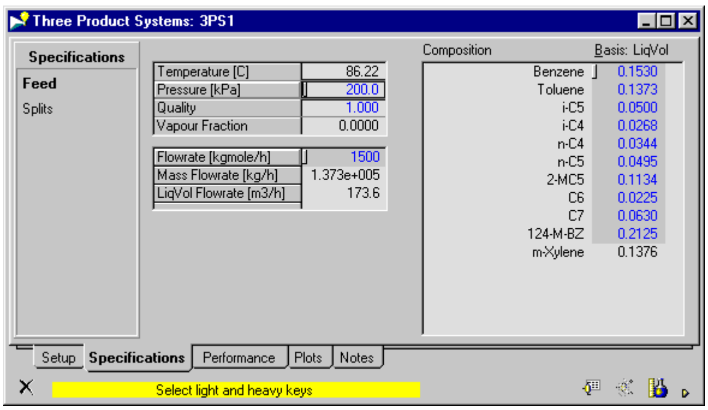
The Feed page allows you to specify the composition, pressure/temperature, and flow rate of the feed stream.

1. On the main TPS view, click the **Specifications** tab, then select the **Feed** page.
2. On the **Feed** page, enter the information from the table below:

Object	Value	Object	Value
Benzene cell	0.1530 mole	**C6 cell**	0.0225 mole
Toluene cell	0.1373 mole	**C7 cell**	0.0630 mole
i-C5 cell	0.0500 mole	**124-M-BZ cell**	0.2125 mole
i-C4 cell	0.0268 mole	**m-Xylene cell**	0.1376 mole
n-C4 cell	0.0344 mole	**Pressure cell**	200 kPa
n-C5 cell	0.0495 mole	**Flowrate cell**	1500 kgmole/h
2-MC5 cell	0.1134 mole		

The Feed page should appear as shown in the figure below:

Figure 6.15



3. Select the **Splits** page.
   The **Splits** page allows you to control the component splits in the mixture by selecting the components that will be in each of the three product streams.

The drop-down lists in the Light Split group allows you to select the components that will be in product A stream and the light component of product B stream.

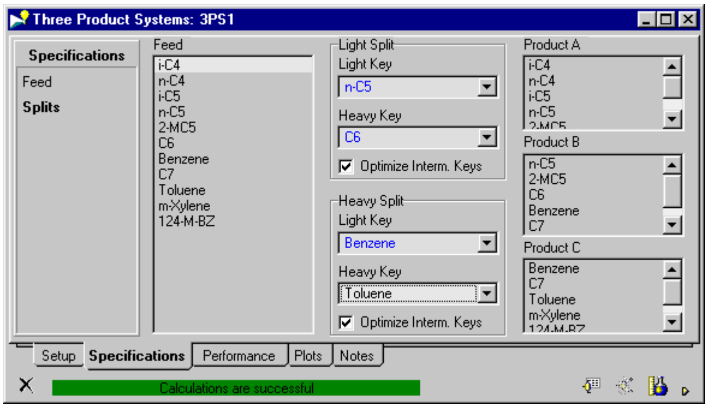
The drop-down lists in the Heavy Split group allows you to select the components that will be in the product C stream and the heavy component of product B stream.

4. In the Light Split group, select **n-C5** component from the **Light Key** drop-down list and select **C6** component from the **Heavy Key** drop-down list.
5. In the Heavy Split group, select **Benzene** component from the **Light Key** drop-down list and select **Toluene** component from the **Heavy Key** drop-down list.

> **After you finish selecting the key components for the split, DISTIL automatically begins calculating the variable values of the complex columns based on the specified product streams.**

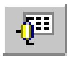
When the calculations and complex column designs are complete, the status bar will become green and display ‘Calculations are successful’. The Splits page should appear as shown in the figure below:

Figure 6.16



DISTIL may have already generated the variable values for the complex column designs, but the recovery percentage for each key component in the streams is set at the specified default values (i.e., **0.98** for the light and heavy key components in each product stream).

To change the recovery percentage for each key components you need to access the Recovery Matrix view:



Recovery Matrix icon

6. Click the **Recovery Matrix** icon at the bottom right corner of the view. The Recovery Matrix view appears.
7. Change the recovery percentage for C6 in the Product A stream by entering **0.04** in the cell intersecting the **C6** row and **Product A** column. The modified value appears in the figure below:

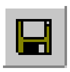
Figure 6.17

Recovery Matrix

Component	Product A	Product B	Product C
i-C4	1.0000	0.0000	0.0000
n-C4	1.0000	0.0000	0.0000
i-C5	1.0000	0.0000	0.0000
n-C5	0.9800	0.0200	0.0000
2-MC5	***	***	0.0000
C6	0.0400	0.9600	0.0000
Benzene	0.0000	0.9800	0.0200
C7	0.0000	***	***
Toluene	0.0000	0.0200	0.9800
m-Xylene	0.0000	0.0000	1.0000
124-M-BZ	0.0000	0.0000	1.0000

DISTIL will automatically recalculate the variable values of the complex column designs every time you modify any percentage values in the Recovery Matrix view.

8. Click the **Close** icon to close the Recovery Matrix view.
9. Click the **Save Case** icon to save the case.



Save Case icon

6.5 Comparing Complex Columns

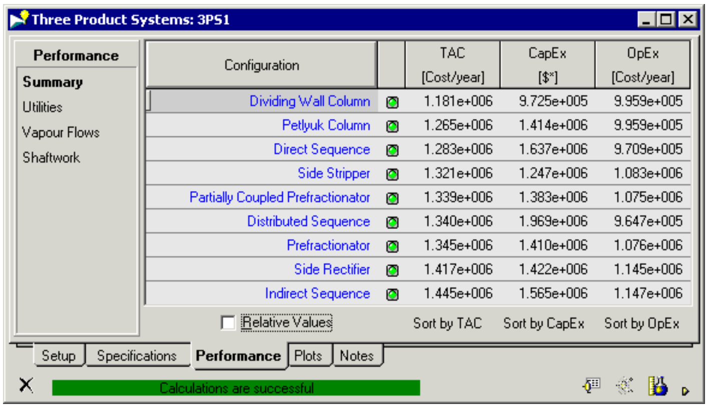
When the Three Product System operation has finish calculating the configuration variables for all nine complex column designs, you can then compare the summary cost, utilities, vapour flowrate, and number of trays for all designs.

In the Summary page, the default setting is to sort the complex column configuration in terms of total annualized cost (TAC), from lowest to highest cost.

1. Click the **Performance** tab, then select the **Summary** page.
2. Uncheck the **Relative Values** checkbox to view the summary cost information in cost per year.

The Direct Sequence design is currently ranked 3rd in terms of TAC.

Figure 6.18



You can double-click on the complex column's name, in the Configuration column, to view the parameters of the complex column in detail.

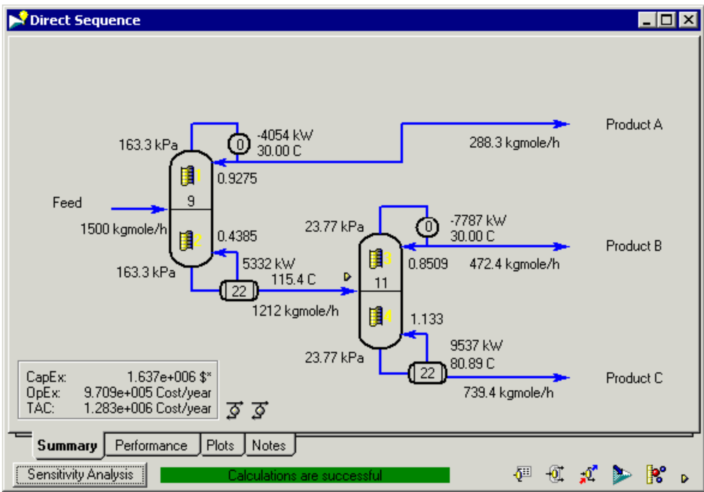
6.5.1 Direct Sequence Design

In this section, you will examine the details of the design parameters from the conventional Direct Sequence design.

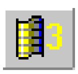
All major design parameters (like operating pressure, exchanger duties, exchanger temperatures, number of trays in each section, etc.) of the Direct Sequence column appear on this view.

1. Double-click the **Direct Sequence** cell in the **Configuration** column. The Direct Sequence view will appear.

Figure 6.19



The second column in the Direct Sequence design is operating at a low pressure (23.77 kPa), so you will change the column to operate at an atmospheric condition.

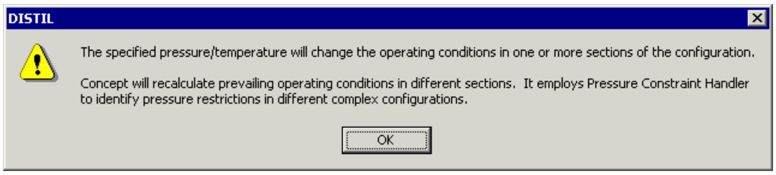


Section 3 icon

2. Click the **Section 3** icon to open the Section 3 view of the column. You can also access the Section 4 view, since section 4 is also located in the same column.
3. In the Section 3 view, locate the **Top Pressure** cell and enter **1 atm** as the new operating pressure value.

A warning view will appear informing you that any changes made to the operating pressure or temperature will result in DISTIL recalculating the values in the entire column design.

Figure 6.20

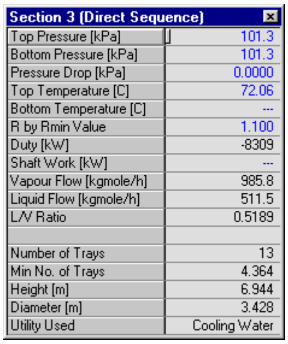


4. Click the **OK** button.

If the parameter value is blue, you can modify the value of that parameter.

5. The Section 3 view should appear as shown in the figure below:

Figure 6.21

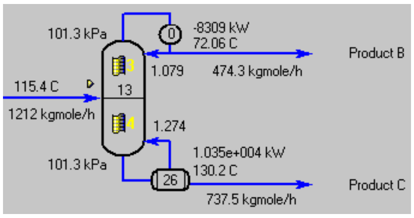


Close icon

6. Click the **Close** icon to close the Section 3 view.

In the Direct Sequence view, the design parameter values for the second column have been changed to accommodate the new operating pressure value.

Figure 6.22



Close icon

7. Click the **Close** icon to close the Direct Sequence view and return to the **Performance** tab, **Summary** page of the TPS view.

In forcing the second column to work at atmospheric conditions, the Direct Sequence configuration has been reduce in rank in terms of TAC. The original rank was **3rd**, and now the rank is **7th**.

Figure 6.23

Configuration		TAC [Cost/year]	CapEx [$*]	OpEx [Cost/year]
Dividing Wall Column		1.181e+006	9.725e+005	9.959e+005
Petlyuk Column		1.265e+006	1.414e+006	9.959e+005
Side Stripper		1.321e+006	1.247e+006	1.083e+006
Partially Coupled Prefractionator		1.339e+006	1.383e+006	1.075e+006
Distributed Sequence		1.340e+006	1.969e+006	9.647e+005
Prefractionator		1.345e+006	1.410e+006	1.076e+006
Direct Sequence		1.394e+006	1.427e+006	1.122e+006
Side Rectifier		1.417e+006	1.422e+006	1.145e+006
Indirect Sequence		1.445e+006	1.565e+006	1.147e+006

8. Repeat the steps you learned in this section to change the columns with low pressure (**< 1 atm**) to 1 atm for the **Indirect Sequence** and **Distributed Sequence** column designs.
   All columns in the Direct Sequence, Indirect Sequence, and Distributed Sequence designs should be operating at atmospheric conditions.
9. When you have finished making the modifications, return to the **Performance** tab, **Summary** page of the TPS view.

In forcing the other columns in the Indirect Sequence and Distributed Sequence column designs to atmospheric conditions, the rank of the complex column configurations have changed.

Figure 6.24

Configuration		TAC [Cost/year]	CapEx [$*]	OpEx [Cost/year]
Dividing Wall Column		1.181e+006	9.725e+005	9.959e+005
Petlyuk Column		1.265e+006	1.414e+006	9.959e+005
Side Stripper		1.321e+006	1.247e+006	1.083e+006
Partially Coupled Prefractionator		1.339e+006	1.383e+006	1.075e+006
Prefractionator		1.345e+006	1.410e+006	1.076e+006
Direct Sequence		1.394e+006	1.427e+006	1.122e+006
Side Rectifier		1.417e+006	1.422e+006	1.145e+006
Indirect Sequence		1.431e+006	1.435e+006	1.157e+006
Distributed Sequence		1.438e+006	1.773e+006	1.100e+006

10. Check the **Relative Values** checkbox to compare the complex column configurations in relative terms.

According to this table, the Dividing Wall Column is the most cost effective option (18% less than the Direct Sequence).

Figure 6.25

Three Product Systems: 3PS1

Performance: Summary, Utilities, Vapour Flows, Shaftwork

Configuration		TAC	CapEx	OpEx
Dividing Wall Column		1.000	1.000	1.000
Petlyuk Column		1.071	1.454	1.000
Side Stripper		1.118	1.282	1.088
Partially Coupled Prefractionator		1.134	1.422	1.080
Prefractionator		1.138	1.450	1.080
Direct Sequence		1.180	1.467	1.126
Side Rectifier		1.199	1.462	1.150
Indirect Sequence		1.211	1.476	1.162
Distributed Sequence		1.218	1.823	1.105

☑ Relative Values Sort by TAC Sort by CapEx Sort by OpEx

Setup | Specifications | Performance | Plots | Notes

Calculations are successful

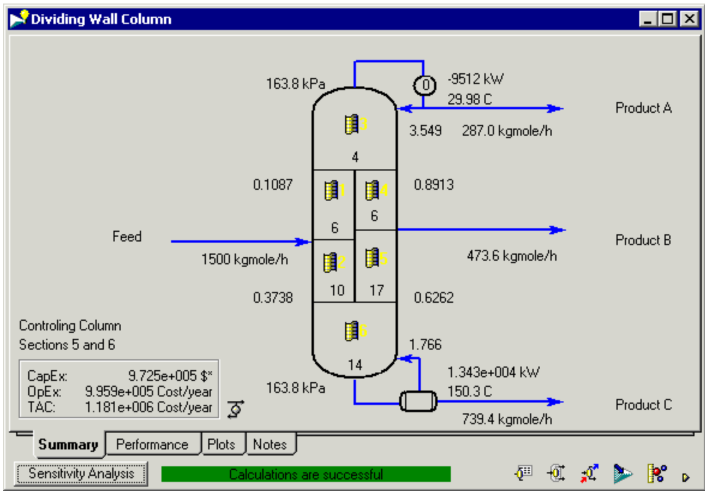
6.5.2 Dividing Wall Column Design

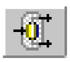
According to the previous DISTIL calculations, the Dividing Wall Column (DWC) is the most cost effective design. This section looks at the design parameters of the DWC in more detail.

1. Double-click on the **Dividing Wall Column** cell in the **Configuration** column.
   The Dividing Wall Column (DWC) view appears.

The DWC is capable of separating the mixture using only one column.

Figure 6.26

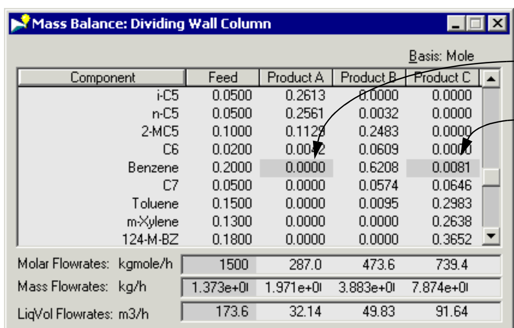




Overall Mass Balance icon

2. Click the **Overall Mass Balance** icon to view the mass balance of the DWC.

Figure 6.27



Mass Balance: Dividing Wall Column

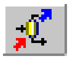
Basis: Mole

Component	Feed	Product A	Product B	Product C
i-C5	0.0500	0.2613	0.0000	0.0000
n-C5	0.0500	0.2561	0.0032	0.0000
2-MC5	0.1000	0.1129	0.2483	0.0000
C6	0.0200	0.0042	0.0609	0.0000
Benzene	0.2000	0.0000	0.6208	0.0081
C7	0.0500	0.0000	0.0574	0.0646
Toluene	0.1500	0.0000	0.0095	0.2983
m-Xylene	0.1300	0.0000	0.0000	0.2638
124-M-BZ	0.1800	0.0000	0.0000	0.3652
Molar Flowrates: kgmole/h	1500	287.0	473.6	739.4
Mass Flowrates: kg/h	1.373e+0l	1.971e+0l	3.883e+0l	7.874e+0l
LiqVol Flowrates: m3/h	173.6	32.14	49.83	91.64

As you can see, the benzene content for Product A and Product C streams are below 15%.

This confirms that the DWC design meets the Clean Air Act regulation.

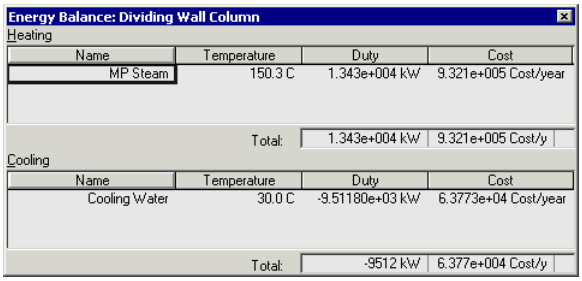
3. Click the **Close** icon to close the Mass Balance view.



Energy Balance icon

4. Click the **Energy Balance** icon to view the utility duties of the condenser and reboiler of the DWC.

Figure 6.28



Close icon

5. Click the **Close** icon to close the Energy Balance view.
6. Click the **Performance** tab in the DWC view.

The Performance tab is divided into six pages:

Blue text indicates that you can modify the value.

- **CapEx** page displays the capital expenditure, operating expenditure, TAC, column dimensions (number of tray, height, and diameter), column/vessel cost, heat exchanger areas, and heat exchanger costs.
- **OpEx** page displays the capital expenditure, operating expenditure, TAC, hot utility's duty and cost, cold utility's duty and cost, and vapour flow rates of condenser and reboiler.
- **Op. Conditions** page displays the operating pressure, temperature, utility used (if applicable), utility's duty (if applicable), and R/Rmin value for each section in the DWC.
- **Sections** page displays the number of trays, minimum number of trays, diameter, height, vapour and liquid flow rate, and the liquid/vapour ratio for each section in the DWC.
- **Volatilities** page displays the relative volatility of all the components in terms of the heaviest component in the mixture.
- **Options** page allows you to modify parts of the column configuration and/or add constraints in designing the column.

7. Return to the **Summary** tab.
8. In the column diagram, notice the number of trays on the left side of the column is **16**, while the number of trays on the right side is **23**.

It can be difficult to balance a column with an uneven load and pressure drop. DISTIL has a function that allows you to automatically balance the number of trays on either side of the dividing wall.

You can also modify the following parameters of the equating tray calculation:

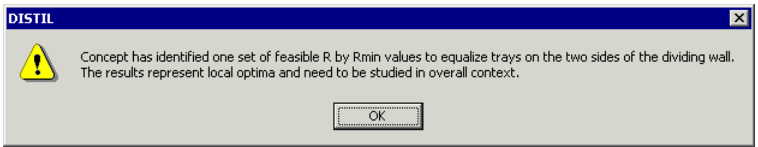
- Specify the number of calculation iterations in the **Max number of Iterations** field.
- Specify the step size of the operating reflux in the **Step size for R by Rmin** field.

9. Click the **Performance** tab, then select the **Options** page.
10. In the Options to Equate Trays on Two Sides of Wall group, click the **Equate Trays by changing R by Rmin** button.

DISTIL will now begin re-calculating the parameter and variable values of the column to different reflux values until a reflux value is found that allows the column to meet specifications and contain an equal number of trays on both sides of the wall.

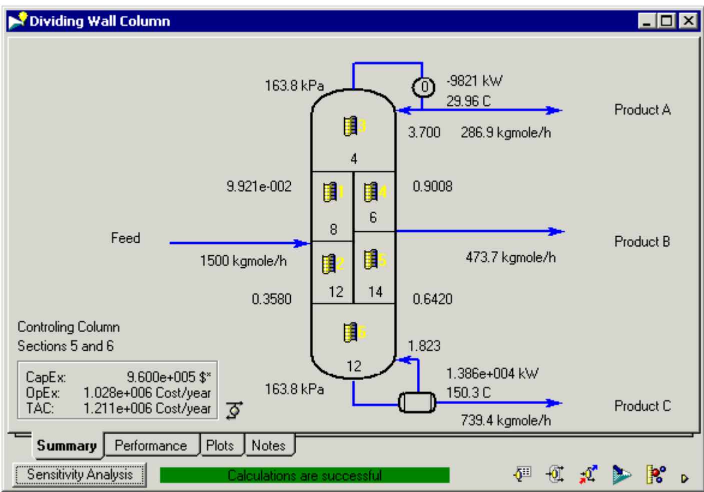
11. If a feasible reflux value is found that allows for equal trays on both side of the wall, a message view will appear to inform you. Click the **OK** button to close the message view.

Figure 6.29



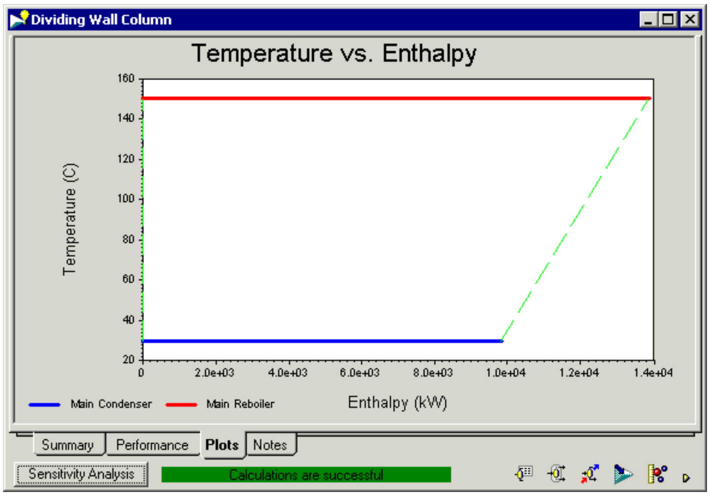
12. Click the **Summary** tab to view the new number of trays.

Figure 6.30



13. Click the **Plots** tab to view the temperature vs. enthalpy graph of the DWC system.

Figure 6.31

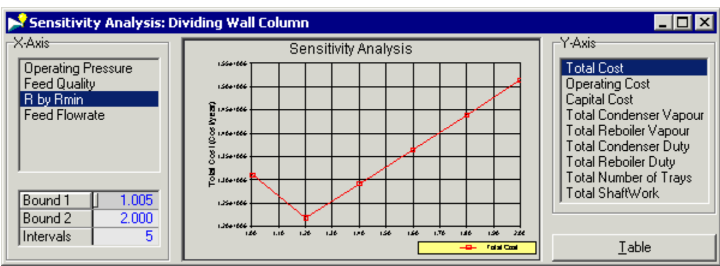


Sensitivity Analysis

DISTIL allows you to perform a sensitivity analysis on the main design variables. For example, you can view how changes in the operating pressure will affect the total cost of the DWC column.

1. Click the **Sensitivity Analysis** button located at the bottom left corner of the DWC view.
   The Sensitivity Analysis view will appear.

Figure 6.32

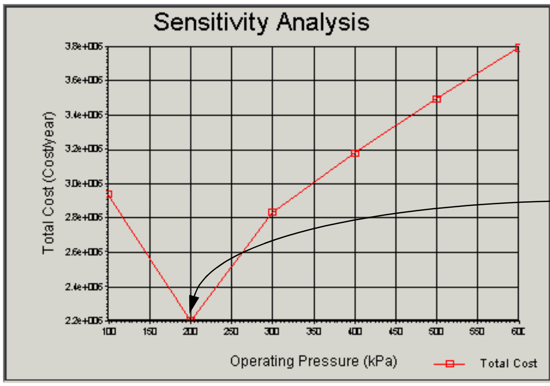


2. In the X-Axis group, select **Operating Pressure**.

3. In the **Bound 2** cell, increase the pressure range value by entering **600 kPa**.

Figure 6.33

At lower pressure, refrigeration is required, and cost increases.



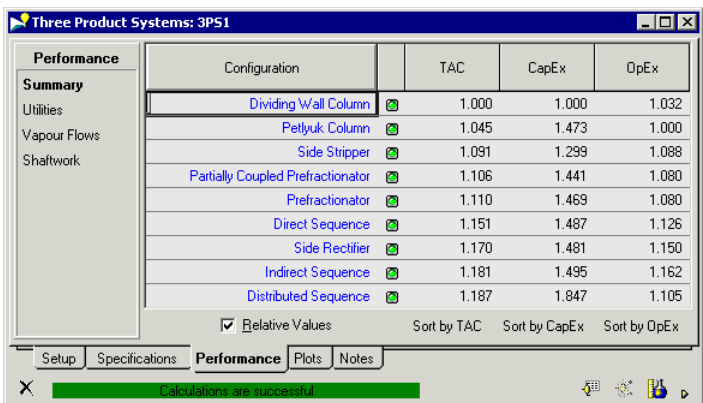
At higher pressure, more expensive hot utility is required, and cost also increases.

The optimum pressure is at 200 kPa, as shown in the plot.

Close icon

4. Click the **Close** icon to close the Sensitivity Analysis view.
5. Click the **Close** icon to close the DWC view and return to the **Performance** tab, **Summary** page of the TPS view.

Figure 6.34



As you can see from the information in the **Summary** page, the changes made in the Dividing Wall Column did not lower the configuration's rank in TAC.

In the next section, you will analyze the trade-off between quality and quantity of energy for different designs.

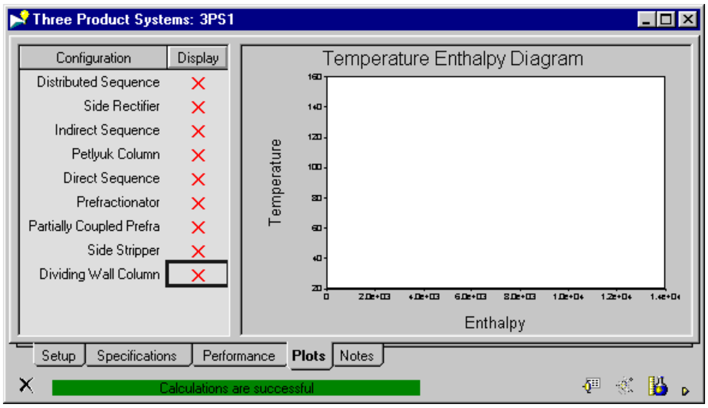
6.5.3 Quality vs. Quantity of Energy

The Three Product System operation contains a tab that displays the temperature vs. enthalpy plot for all complex columns, which allows you to compare the energy quality and energy quantity of each complex column.

The Temperature vs. Enthalpy plot is blank by default.

1. Click the **Plots** tab of the TPS view.

Figure 6.35

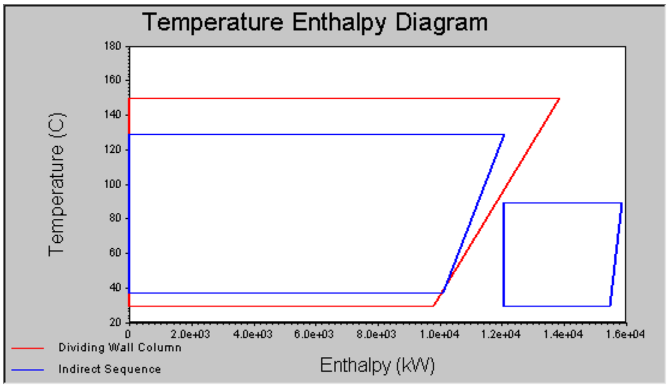


2. To plot the temperature vs. enthalpy data of the complex columns, click the icons in the **Display** column.
   - **Red X** icon indicates that the associated complex column data is removed from the plot. Click on this icon to change the icon to a green checkmark and display the associated complex column data on the plot.
   - **Green Checkmark** icon indicates that the associated complex column data is displayed on the plot. Click on this icon to change the icon to a red X and remove the associated complex column data from the plot.

In terms of TAC, the Indirect Sequence design was 22% more than the DWC design.

3. Click the icon in the **Display** column for the Dividing Wall Column (DWC) and the Indirect Sequence designs.

Figure 6.36

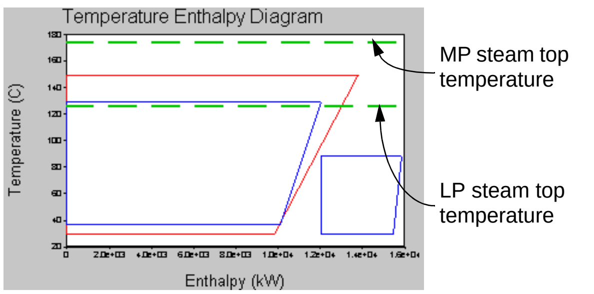


LP steam utility is slightly cheaper than MP steam utility.

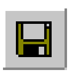
Based on the data plots from the figure above, you can make the following assumptions:

- The Indirect Sequence design requires more utility energy than the DWC design.
- The DWC design requires less utility energy than Indirect Sequence.
- The Indirect Sequence design uses both LP steam and MP steam hot utilities to heat the streams.
- The DWC design uses only MP steam to heat the streams.

Figure 6.37



The Indirect Sequence needs more energy (quantity), but requires a less expensive hot utility (quality). The DWC needs less energy (quantity), but requires a more expensive hot utility (quality).



Save Case icon

4. Click the **Save Case** icon to save the case.

6.6 Simulating the Design in HYSYS

Once you have decided which complex column design is the most promising candidate to separate the benzene from the gasoline product mixture, you can perform a more rigorous simulation on the selected complex column system.

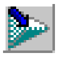
DISTIL allows you to extract information from the selected complex column from the Three Product System operation and export the information to HYSYS. Once in HYSYS, you can perform a more rigorous simulation.

To perform the simulation in HYSYS, you need HYSYS 3.0 or higher.

6.6.1 Extracting the Column Design

In this section you will extract the complex column system and export it to HYSYS.

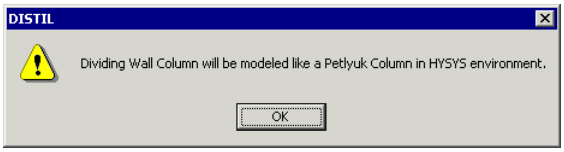
1. Click the **Performance** tab of the TPS view.
2. Double-click on the **Dividing Wall Column** cell in the Configuration column. The DWC view appears.



Simulate icon

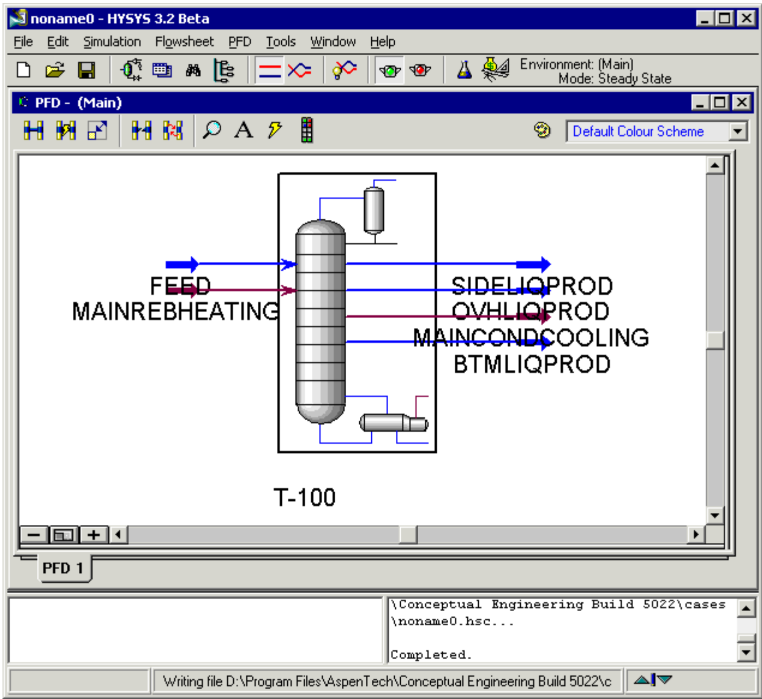
3. In the DWC view, click the **Simulate** icon.
4. A warning message appears to inform you that the DWC will be modeled as a Petlyuk column in HYSYS. Click the **OK** button

Figure 6.38



5. DISTIL will automatically activate HYSYS and send the fluid package and selected complex column design specifications to HYSYS.

Figure 6.39



6.6.2 Comparing Calculated Results

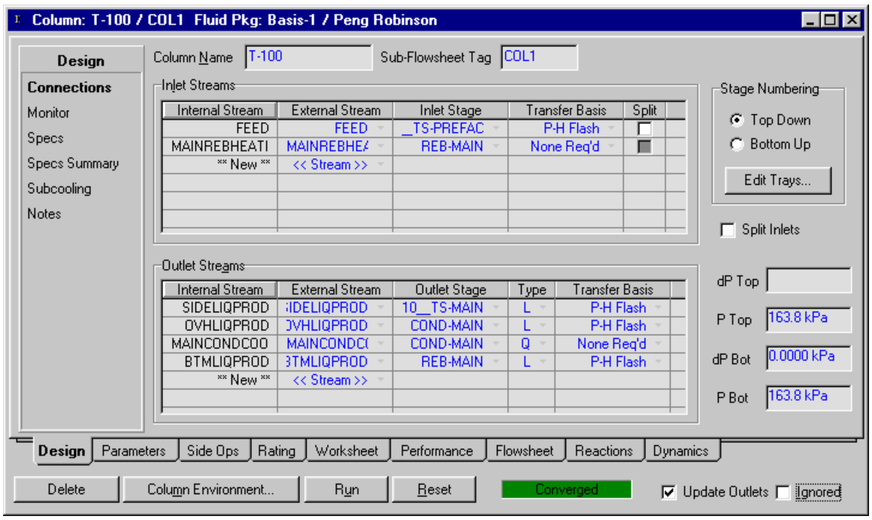
Once the selected complex column design is exported to HYSYS, HYSYS automatically performs an initial calculation on the entire separation system.

The re-calculation occurs because DISTIL assumptions are more relaxed than HYSYS assumptions.

Some of the values calculated by HYSYS will be different from the DISTIL values. To compare the calculated values, do the following:

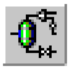
1. In HYSYS, double-click on the Column object in the PFD to open the Column property view.

Figure 6.40



2. Click the **Column Environment** button at the bottom of the Column view to enter the Column environment.

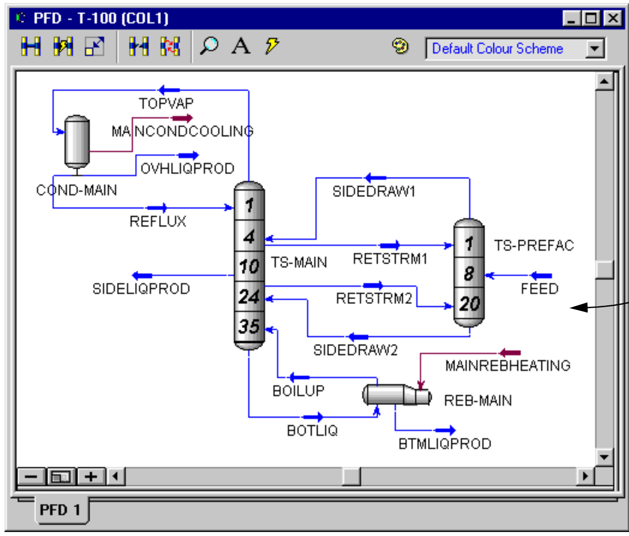
The Column environment provides more detailed views and information for the column configuration.



PFD icon

3. In the Column environment, click the **PFD** icon to open the PFD view of the Column environment (if it isn't already visible).

Figure 6.41



As the warning message had stated, the DWC would be modeled as a Petlyuk column in HYSYS.

4. In the Column environment PFD view you can do the following:
   - Double-click any of the Stream objects to access the properties and composition of the selected stream.
   - Double-click the Condenser or Reboiler object to access the properties and parameters of the condenser or reboiler.
   - Double-click the Column objects to access the properties and parameters of the columns.

In the object views (stream, column, and exchangers), you can change any values where the value text is blue or red.

- Blue text indicates the values were entered manually.
- Red text indicates the values are HYSYS and/or DISTIL default values.
- Black colour text indicates that the values are calculated by HYSYS. You cannot modify these values.

5. From the **File** menu, select **Save As**, and save the HYSYS case as **ImportedBenzeneRemovalCase**.

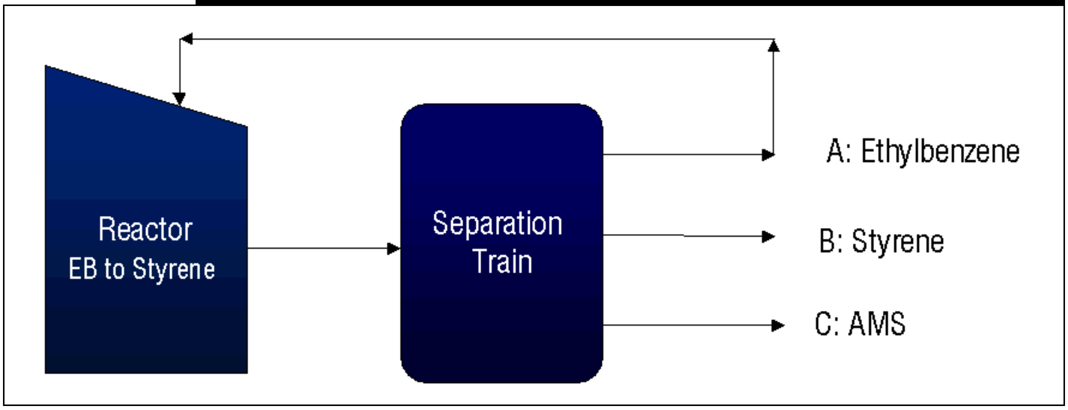
7 Styrene Separation

7.1 Introduction

This is an advanced tutorial. It is assumed that you know all the basic functions of DISTIL. For information about the basic functions, refer to the **User Guide** and **Separation** manual.

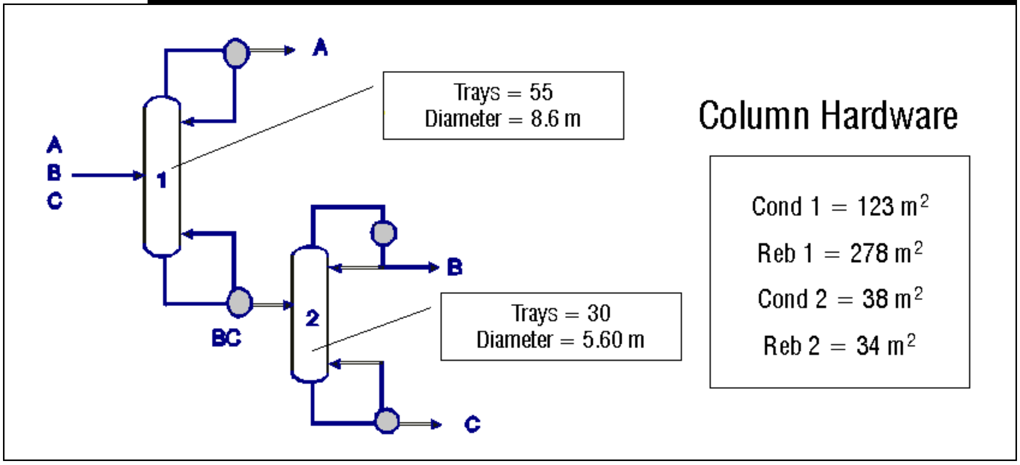
In this tutorial, you will use DISTIL to help you decide whether to replace or modify an existing separation train of a styrene separation process with a more efficient column configuration. A more efficient separation process is required because the profitability of the plant has dropped by 20% due to increase in energy cost.

Figure 7.1



The current separation train used in the styrene separation process is similar to a Direct Sequence design.

Figure 7.2



There are many possible column designs for the above separation system. DISTIL's Three Product System (TPS) operation offers nine different complex column designs from which you can compare the designs to select the column configuration that meets your requirements.

7.2 Setting Unit Preferences

Before you begin setting up the simulation, verify that the units currently selected in the DISTIL preferences are the ones you want to use.

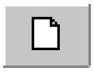
For this tutorial, the temperature is in Celsius and the Cost Index per Time is in $/year. Since you defined a unit set containing this variable in the previous tutorial (Cost in years-SI), you do not have to create a new one.

1. Activate DISTIL, if it is not already open.
2. From the **Tools** menu, select **Preferences**. The Session Preferences view appears.
3. Click the **Variables** tab, then select the **Units** page.
4. In the Available Unit Sets group, select the **Cost in years-SI** unit set.

7.3 Defining the Fluid Package

To use the Three Product System operation, you need to generate a fluid package containing the property package and components required to simulate the feed stream entering the separation system.

Fluid Package Manager icon



Add Fluid Package icon

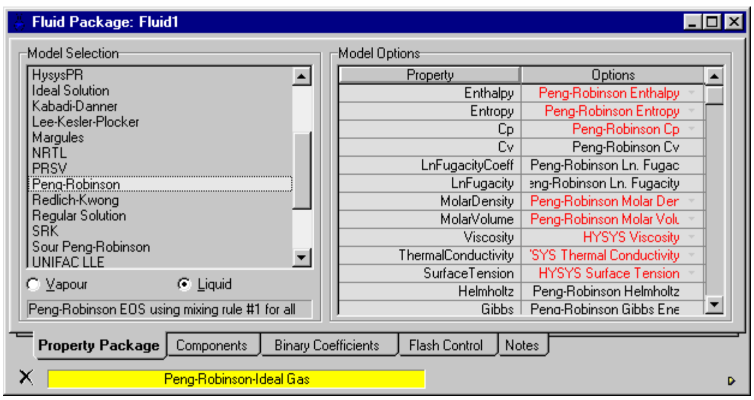
1. Activate DISTIL, if it is not already open.
2. Open the Fluid Package Manager view by click the **Fluid Package Manager** icon.
3. In the Fluid Package Manager view, click the **Add Fluid Package** icon.
   The Fluid Package view will appear.

7.3.1 Selecting a Property Package

In the Fluid Package view, you will select the property package and components associated with the fluid package.

1. In the Fluid Package view, click the **Property Package** tab.
2. In the Model Selection group, select the **Vapour** radio button, then select **Ideal Gas** in the Model Selection list.
3. In the Model Selection group, select the **Liquid** radio button, then select **Peng-Robinson** in the Model Selection list.

Figure 7.3



The status bar at the bottom of the Fluid Package view should display Peng-Robinson-Ideal Gas as the selected property package, as shown in the previous figure.

7.3.2 Selecting the Components

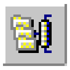
1. Click the **Components** tab.
2. In the Available Component List/Filter Options group, type **benzene** in the Match field.
   The list under the Match field will display all components that have names with the word **benzene** in them.
3. Select **Benzene** from the list and click the **Select** button.
   The component benzene will be added to the Selected Components list.
4. Repeat steps #2 to #3 to add the following components:
   - Toluene
   - Styrene
   - E-BZ
   - alpha-M-Styrene
5. Click the **Notes** tab. In the **Name** field, enter a name for the fluid package (e.g., **PR-styrene**).

Close icon

6. Click the **Close** icon to close the Fluid Package view.
7. Save the case as **StyreneSeparation**. DISTIL will save the file with the ***.hcd** extension.

7.4 Creating a Three Product System

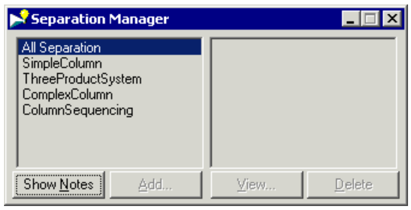
After specifying the fluid package for the simulation, you are ready to use the Three Product System (TPS) operation.



Separation Technology Manager icon

1. Click the **Separation Technology Manager** icon to open the Separation Manager view.

Figure 7.4

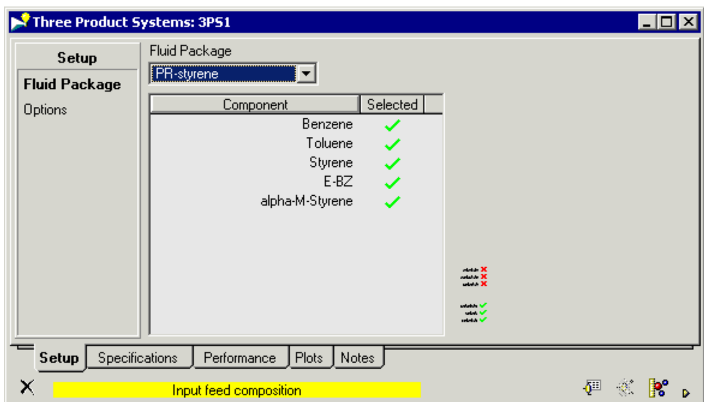


2. In the left list of the Separation Manager view, select **ThreeProductSystem**.
3. Click the **Add** button. The TPS property view will appear. DISTIL automatically names the TPS operation 3PS1.

The name of the TPS operation will appear in the right list of the Separation Manager view.

4. On the TPS view, select the **PR-styrene** fluid package from the **Fluid Package** drop-down list.

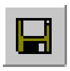
Figure 7.5



Modifying the Utility Database

In this tutorial, there is no Air utility, so you will remove the Air utility from the DISTIL default utility database.

1. On the **Setup** tab, **Options** page, click the **Utility Database** button in the Economics group. The Utility Database view appears.
2. Select the **Air** cell in the **Name** column.
3. Press DELETE. DISTIL will remove **Air** from the utility database.
4. Click the **Save Default Utilities to File** icon, the Save Heat Integration Defaults view appears.



Save Default Utilities to File icon

5. In the **File Name** field, enter **NoAir-utildata** and click the **Save** button to save the modified utility database.
6. Click the **Close** icon to close the Utility Database view.

Modifying the Tray Spacing

In this tutorial, the column's tray spacing is 18 inches. You must check and ensure the assumed value for tray spacing is 18 inches.

1. On the **Setup** tab, **Options** page, click the **Capital Cost** button in the Economics group. The Capital Cost view appears.
2. Click the **Column** tab, and select the **Sizing** page.
3. In the **Tray Spacing** field, check that the value is **18 inch**. If the value is incorrect, use the down arrow and select **18 inch** from the drop-down list.
4. Click the **Close** icon to close the Capital Cost view.

7.4.1 Specifying Feed and Product Streams

Now that you have selected the fluid package for the Three Product System operation, you will specify the feed and product streams characteristics.

The Feed page allows you to specify the composition, pressure/temperature, and flow rate of the feed stream.

1. On the main TPS view, click the **Specifications** tab, then select the **Feed** page.
2. Enter the information as shown in the table below:

Object	Value	Object	Value
Styrene cell	0.500	**alpha-M-Styrene**	0.0500
Benzene cell	0.0200	**Pressure cell**	30.00 kPa
Toluene cell	0.0300	**Flowrate cell**	1000
E-BZ cell	0.4000		

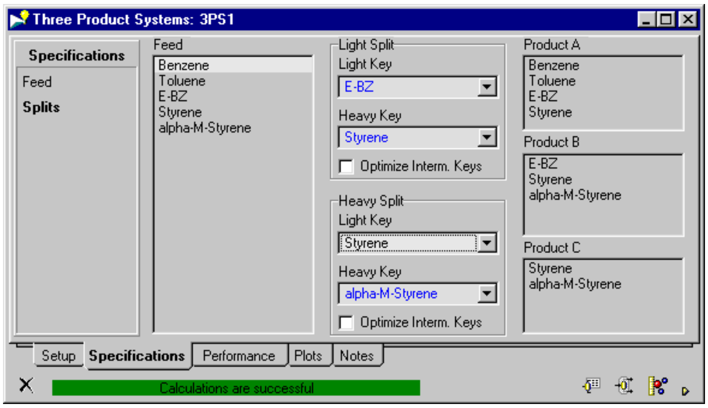
The Splits page allows you to control the component splits in the mixture by selecting the components that will be in each of the three product streams.

3. Select the **Splits** page.
4. In the Light Split group, select **E-BZ** component from the **Light Key** drop-down list and select **Styrene** component from the **Heavy Key** drop-down list.
5. In the Heavy Split group, select **Styrene** component from the **Light Key** drop-down list and select **alpha-M-Styrene** component from the **Heavy Key** drop-down list.

After you finish selecting the key components for the splits, DISTIL automatically begins calculating the variable values of the complex columns based on the specified product streams.

6. The Splits page should appears as shown in the figure below:

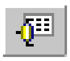
Figure 7.6



You can change the default value in the Default Key Recovery field of the Setup tab, Options page.

In this tutorial, the DISTIL default recovery percentage values for the key components are the desired values. The DISTIL default recovery percentage value is **0.95**.

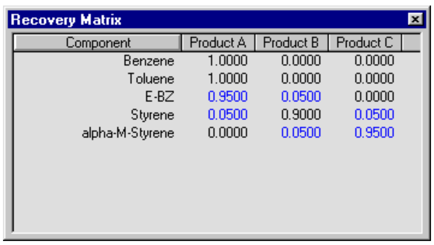
To view the recovery percentage values of the components for each product stream, open the Recovery Matrix view.



Recovery Matrix icon

7. Click the **Recovery Matrix** icon at the bottom right corner of the view to open the Recovery Matrix view.

Figure 7.7


Recovery Matrix

Component	Product A	Product B	Product C
Benzene	1.0000	0.0000	0.0000
Toluene	1.0000	0.0000	0.0000
E-BZ	0.9500	0.0500	0.0000
Styrene	0.0500	0.9000	0.0500
alpha-M-Styrene	0.0000	0.0500	0.9500

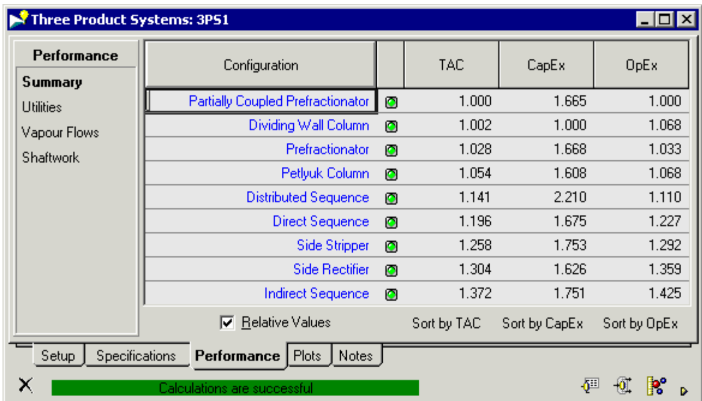
7.5 Comparing Complex Columns

When the Three Product System operation has finish calculating the configuration variables for all nine complex column designs, you can compare the summary cost, utilities, vapour flowrate, and number of trays for all designs.

In the Summary page, the default setting is to sort the complex column configuration in terms of total annualized cost (TAC), from lowest to highest cost.

1. Click the **Performance** tab, then select the **Summary** page.

Figure 7.8



Double-click on the complex column's name under the Configuration column to view the parameters of the complex column in detail.

Based on the calculated results in the Summary page, the top five ranking column configurations are prefractionation based designs. In these designs, the feed mixing losses are minimized.

The designs where the back mixing is important rank 7th and 8th place (Side Stripper and Side Rectifier).

7.5.1 Analyzing the Styrene Separation Process

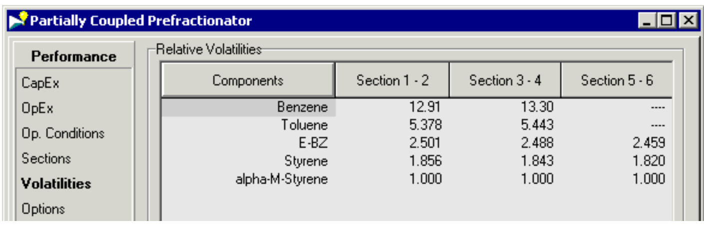
In this section, you will analyze the styrene separation process performance and the calculated results from the Three Product System operation compared with the performance to confirm that a complex column design would be a more efficient process.

For the styrene separation process, consider the following assumptions and conclusions:

- Since the effect of back mixing is less important, the Side Stripper and Side Rectifier designs need not be considered.
- Since the feed contains about 48% middle product, it is more efficient to use Prefractionator base columns to separate the mixture.
- The splits in the styrene separation process are difficult, as indicated by the relative volatility values calculated by DISTIL. You can check the components' relative volatility values in the individual complex column view on the Performance tab, Volatilities page.

Refer to **Section 5.3.4 - Complex Column Design Options** in the **Reference Guide** for more information.

Figure 7.9



Based on the values in the figure above, the relative volatility value for the first split is $\text{E-BZ}/\text{Styrene} = 1.35$, and for the second split is $\text{Styrene}/\text{alpha-M-Styrene} = 1.84$.

When the split ratio of the relative volatility value is less than 2, the Light and Heavy key components in the split are considered difficult to separate.

- There is a need to lower operating costs due to higher energy costs. Complex column configurations are known to lower the operating costs. For example, the Partially Coupled Prefractionator (PCP) design has the lowest operating cost because the configuration allows for the use of low temperature hot utilities.
- As mentioned previously, the existing separation train is similar to a Direct Sequence configuration. The calculated values for the complex column configuration (**Figure 7.8**) indicate that the PCP's TAC is 19% less than the Direct Sequence and the PCP's operating cost is 23% less than the Direct Sequence

In conclusion, a Prefractionator base column configuration would improve the efficiency of the separation train in the styrene separation process. The next step is to decide if the existing columns in the process need to be replaced or modified.

7.5.2 Replicating the Existing Separation Train

A good engineer should always see if some (ideally most) of the hardware elements in the existing column sequence can be reused in the modified energy efficient column configuration to reduce the cost of material needed to modify the existing column sequence.

Whether the existing hardware can be re-used will depend on the selection of the possible new column configuration.

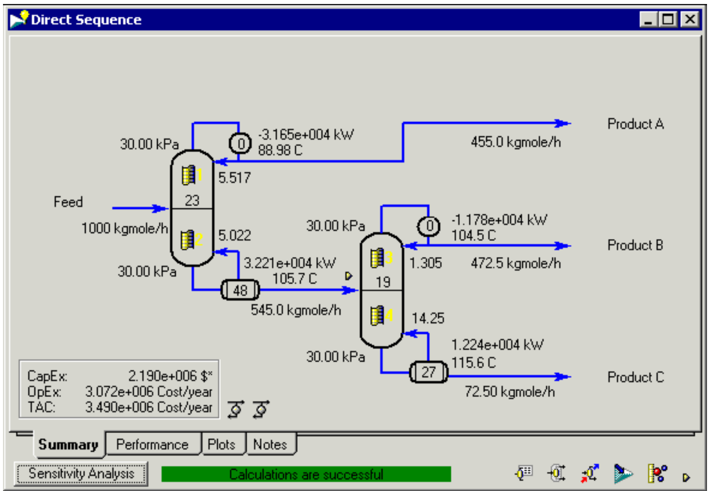
Which column configuration you choose will depend on your ability to properly compare the existing configuration with the possible improved configurations.

As mentioned at the beginning of this chapter, the existing separation train configuration is similar to the Direct Sequence configuration, however, the calculated cost values of the Direct Sequence configuration are based on the Three Product System operation parameter values, not on the existing separation train parameter values, so the cost values between the configurations are not accurate.

DISTIL allows you to manipulate the parameter values of the Direct Sequence in the TPS operation to match the existing separation train configuration, creating a more accurate configuration/design for comparison.

1. Open the Direct Sequence view.

Figure 7.10



According to the information in the Summary tab, the first column contains 48 trays and the second column contains 27 trays. The existing separation train's first and second columns contain 55 and 30 trays respectively.

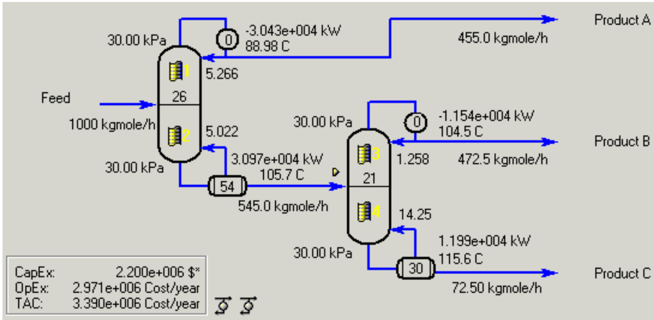
Refer to **Section 5.2.4 - Column Retrofit Options** in the **Reference Guide** for the detailed relationship between reflux ratio and number of trays in the column.

The number of trays in the Direct Sequence view needs to be increased. The number of trays in a column is inversely proportional to the reflux ratio value of the column, so decreasing the reflux ratio value of the column will increase the number of trays in the column.

2. Click the **Performance** tab.
3. Select the **Op. Conditions** page. The current reflux ratio value for the column is 1.100.
4. Decrease the reflux ratio value for the column and observe the change in the number of trays.

You will find that the values **1.05** for the first column (Sections 1 &2) and **1.06** for the second column (Sections 3 &4) work well for the reflux ratio value.

Figure 7.11



7.5.3 Modifying the PCP Configuration

As stated in the previous section, it is cheaper to take the existing column sequence and modify it to match the more cost efficient configuration.

- The existing separation train for the styrene separation process has two column shells, so from the list of possible complex columns, a configuration with two column shells is considered.
- The existing separation train is not energy efficient, so from the list of possible complex columns, the complex column with the lowest operating cost is considered.

From the above two requirements, the first configuration to be considered is the Partially Coupled Prefractionator (PCP). The columns in the PCP configuration will be modified on the assumption that the existing columns in the styrene separation process are still used for the separation.

Figure 7.12

Performance

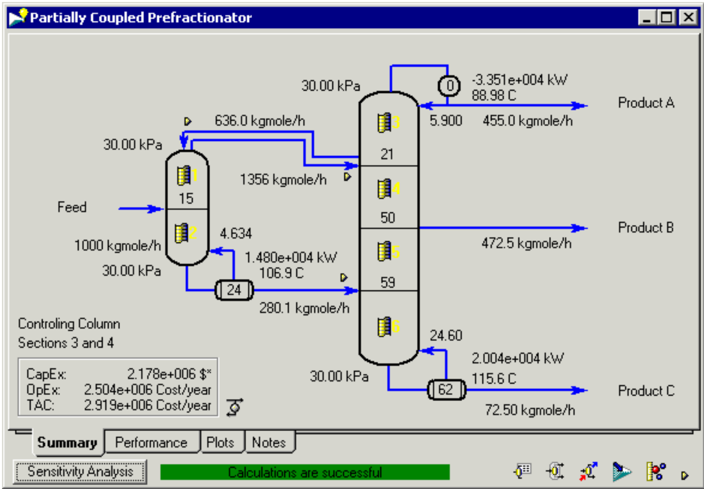
Summary
Utilities
Vapour Flows
Shaftwork

Configuration		TAC	CapEx	OpEx
Partially Coupled Prefractionator		1.000	1.665	1.000
Dividing Wall Column		1.002	1.000	1.068
Prefractionator		1.028	1.668	1.033
Petlyuk Column		1.054	1.608	1.068
Distributed Sequence		1.141	2.210	1.110
Direct Sequence		1.162	1.682	1.187
Side Stripper		1.258	1.753	1.292
Side Rectifier		1.304	1.626	1.359
Indirect Sequence		1.372	1.751	1.425

Notice that the Direct Sequence cost went down from 20% (see **Figure 7.8**) to 16.6% more than the PCP after the modification.

1. Open the PCP view.

Figure 7.13



The first column of the PCP contains 24 trays and the second column contains 62 trays.

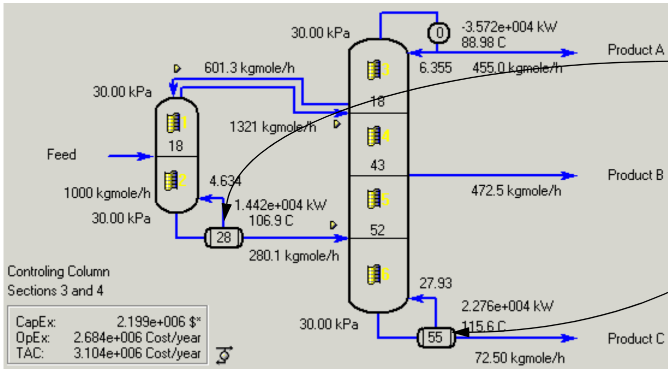
- There is a column in the existing separation train that contains 30 trays, which can be used as the first column (prefractionator) of the PCP configuration. The reflux ratio value of the first column of the PCP can be reduced to increase the number of trays.
- The other column in the existing separation train contains only 55 trays, so the reflux ratio will increase to reduce the number of trays in the second column of the PCP from 62 to 55.

Matching the Number of Trays

Click the **Section 1** and **Section 3** icons to open the section views.

1. On the **Summary** tab, open the Section 1 and Section 3 views.
2. Reduce the reflux ratio value in the Section 1 view to increase the number of trays in the first column.
   You will find that **1.04** is adequate for this tutorial.
3. Increase the reflux ratio value in the Section 3 view to decrease the number of trays in the second column.
   You will find that **1.20** is adequate for this tutorial.

Figure 7.14

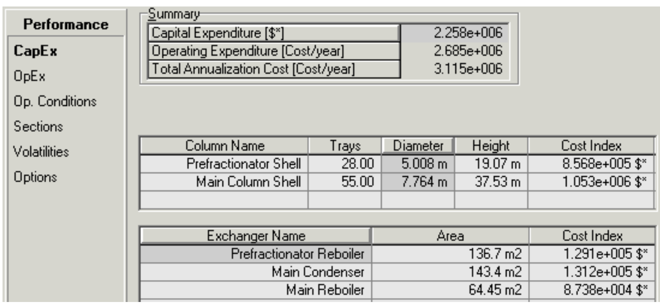


The first column now contains 28 trays, and the second column now contains 55 trays.

While the number of trays now appears suitable, there may be other column parameters that have been modified due to the changes in reflux ratio. The other column parameters, such as the tray diameter, should be checked.

4. Click the **Performance** tab, then select the **CapEx** page.

Figure 7.15



Capital Expenditure [$*]	2.258e+006
Operating Expenditure [Cost/year]	2.685e+006
Total Annualization Cost [Cost/year]	3.115e+006

Column Name	Trays	Diameter	Height	Cost Index
Prefractionator Shell	28.00	5.008 m	19.07 m	8.568e+005 $*
Main Column Shell	55.00	7.764 m	37.53 m	1.053e+006 $*

Exchanger Name	Area	Cost Index
Prefractionator Reboiler	136.7 m2	1.291e+005 $*
Main Condenser	143.4 m2	1.312e+005 $*
Main Reboiler	64.45 m2	8.738e+004 $*

The tray diameters for the PCP columns are about 5.817 m and 9.010 m. The tray diameter for the existing columns are 5.6 m and 8.6 m. Clearly the PCP columns are too fat, and the tray diameters need to be decreased.

Consider the factors that controls the tray diameter:

- **Vapour Density**. This variable can be manipulated; the value is not fixed.
- **Vapour/Liquid Traffic**. This variable cannot be manipulated; the value is fixed due to the required/specified composition of the product streams.

In this tutorial, the vapour density value will be manipulated to decrease the tray diameter. The vapour density has an inverse relationship with the tray diameter:

- When the vapour density value is small, the vapour in the column requires more space/volume, so the tray diameter needs to be large to increase the volume of the column.
- When the vapour density value is large, the vapour in the column requires less space/volume, so the tray diameter needs to be small to reduce the volume of the column.

The variable that increases or decreases the vapour density in the column is the operating pressure. Increasing the operating pressure will increase the vapour density, and decreasing the operating pressure will decrease the vapour density.

In this tutorial the operating pressure will be increased to increase the vapour density value and decrease the tray diameter.

The increase in operating pressure, however, may increase the difficulty in the separation process, which means more trays will be required in the column to achieve the required/specified separation.

DISTIL will automatically increase the number of trays in the column to compensate for the increased difficulty in the separation process. In order to match the number of trays in the simulated columns with the existing column, the reflux ratio value will have to be modified.

The above analysis relies on basic knowledge about how to manipulate tray diameter, however, this analysis has been done qualitatively. There are no numbers to support the reasoning.

DISTIL provides the *quantitative* analysis. The operating pressure value of the simulated columns can be increased or decreased to show how the quantitative changes to the value affect the rest of the simulated columns.

Matching the Column Diameters

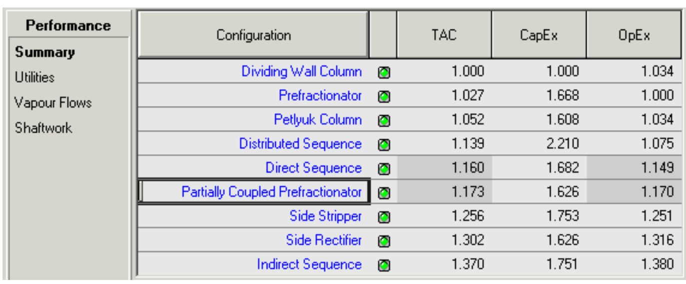
1. Click the **Summary** tab, then open the Section 1 view.
2. Increase the **Top Pressure** of the first column to **45 kPa**. The following results occur:
   - The tray diameter is reduced from **5.817 m** to **5.585 m**.
   - The number of trays increases from **28** to **29**.
3. Open the Section 3 view.
4. Increase the **Top Pressure** for the second column to **46 kPa**. The following results occur:
   - The tray diameter is reduced from **9.010 m** to **8.476 m**.
   - The number of trays increases from **55** to **57**.
5. On the Section 3 view, increase the reflux ratio value to **1.24**. The number of trays is reduced from **57** to **55**.

Analyze the Rank of the Modified PCP

Now that the simulated PCP configuration has been successfully modified to match the existing column's hardware (column vessel and trays), the modified PCP's cost will be compared to the existing separation column's cost.

1. On the main TPS view, click the **Performance** tab and select the **Summary** page.

Figure 7.16



Configuration	TAC	CapEx	OpEx
Dividing Wall Column	1.000	1.000	1.034
Prefractionator	1.027	1.668	1.000
Petlyuk Column	1.052	1.608	1.034
Distributed Sequence	1.139	2.210	1.075
Direct Sequence	1.160	1.682	1.149
Partially Coupled Prefractionator	1.173	1.626	1.170
Side Stripper	1.256	1.753	1.251
Side Rectifier	1.302	1.626	1.316
Indirect Sequence	1.370	1.751	1.380

According to **Figure 7.16**, the capital cost of the modified PCP (1.626) is still lower than the capital cost of the existing column (1.682), therefore, the operating cost has to be the only factor that increased the TAC of the PCP.

The modified PCP configuration has a lower total annualized cost (TAC) rank than it did before the modifications. The modified PCP appears to be more expensive than the existing column separation in both TAC and operating costs.

Considering that the PCP configuration was first chosen because it had the lowest operating cost, an analysis on why the PCP's operating cost increased so substantially must be performed.

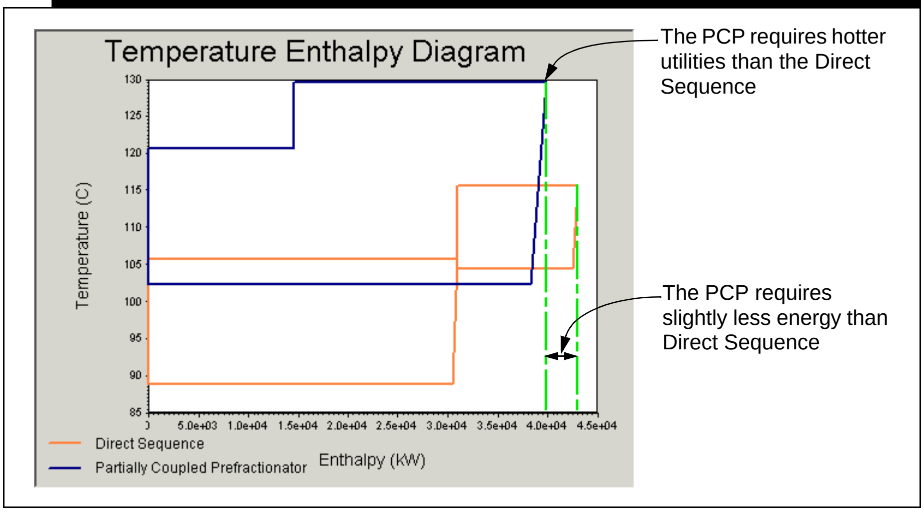
Factors that Increased the Operating Cost

Since the only column parameters that were changed in the modified PCP were the reflux ratio and the operating pressure, it can be assumed that the changes in the two parameters are the factors that increased the operating cost.

In this section you will analyze how the changes in the reflux ratio and operating pressure affected the cost.

1. On the TPS view, go to the **Plots** tab.
2. Select the **Direct Sequence** and **Partially Coupled Prefractionator** configurations to be displayed in the plot.

Figure 7.17



From the resulting plot, you can conclude the following regarding the reason for the large increase in the operating cost of the PCP:

- Operating costs increased because of the need for hotter utilities.
- Hotter utilities were required because of the increase in reflux ratio and operating pressure.
- Operating pressure and reflux ratio increased because of the need to increase the vapour density.

- Vapour density increased because of the need to reduce the diameter of the column.
- Column diameter was too fat because of the high vapour load.

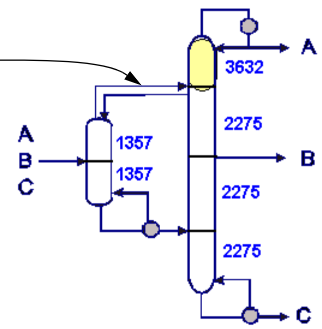
The analysis above indicates that the vapour load is the main factor contributing to the increased operation cost.

The next step is to find out why there is a high vapour load, and decide if there is a way to reduce the vapour load and thus reduce the operating cost.

From the configuration of the PCP design, you can see that the top trays of the second column (Section 3) will contain very high vapour flow rate.

Figure 7.18

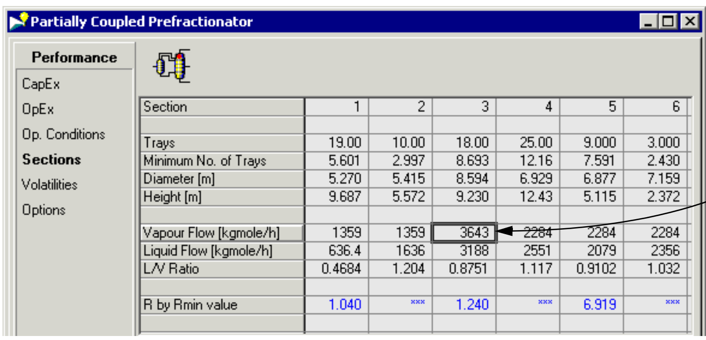
There is a large vapour flow rate in Section 3, because the columns are thermal coupled (only one condenser for two columns).



DISTIL confirms that Section 3 contains the highest vapour flow rate quantitatively.

3. On the PCP view, click the **Performance** tab, then select the **Sections** page.

Figure 7.19



Section	1	2	3	4	5	6
Trays	19.00	10.00	18.00	25.00	9.000	3.000
Minimum No. of Trays	5.601	2.997	8.693	12.16	7.591	2.430
Diameter [m]	5.270	5.415	8.594	6.929	6.877	7.159
Height [m]	9.687	5.572	9.230	12.43	5.115	2.372
Vapour Flow [kgmole/h]	1359	1359	3643	2284	2284	2284
Liquid Flow [kgmole/h]	636.4	1636	3188	2551	2079	2356
L/V Ratio	0.4684	1.204	0.8751	1.117	0.9102	1.032
R by Rmin value	1.040	***	1.240	***	6.919	***

The Vapour Flow row indicates that Section 3 contains the highest vapour flow rate.

To reduce the high vapour flow rate, the thermal coupling could be removed to redistribute the vapour flow rate more evenly throughout the configuration, so a new configuration needs to be considered, preferably one without thermal coupling.

7.5.4 Modifying the Prefractionator Configuration

The Prefractionator configuration is the next candidate for the separation train because it does not contain thermal coupling and it is similar to the PCP configuration.

Figure 7.20

Performance: **Summary** | Utilities | Vapour Flows | Shaftwork

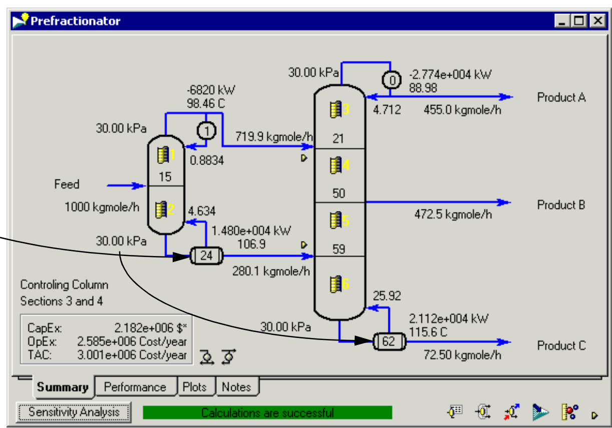
Configuration	TAC	CapEx	OpEx
Dividing Wall Column	1.000	1.000	1.034
Prefractionator	1.027	1.668	1.000
Petlyuk Column	1.052	1.608	1.034
Distributed Sequence	1.139	2.210	1.075
Direct Sequence	1.160	1.682	1.149
Partially Coupled Prefractionator	1.173	1.626	1.170
Side Stripper	1.256	1.753	1.251
Side Rectifier	1.302	1.626	1.316
Indirect Sequence	1.370	1.751	1.380

The Prefractionator configuration is currently ranked in 2nd place.

1. On the main TPS view, open the Prefractionator view.

Figure 7.21

The first column of the Prefractionator contains 24 trays and the second column contains 62 trays.



Just like the PCP, the Prefractionator column parameters will be modified to match the parameters of the existing column.

2. For the first column, reduce the reflux ratio value to **1.04**. The number of trays increases to 28 and the tray diameter changes to 5.747 m.
3. For the second column, increase the reflux ratio value to **1.2**. in the second column to decrease the number of trays. The number of trays decreases to 55 and the tray diameter changes to 8.23 m.

The tray diameter for the first column is slightly too large, so like the PCP configuration, the first column's operating pressure needs to be increased to reduce the vapour density, and therefore reduce the tray diameter.

4. For the first column, increase the operating pressure to **39 kPa**. The tray diameter changes to 5.446 m.

Analyze the Rank of the Modified Prefractionator

Now that the simulated Prefractionator configuration has been successfully modified to match the existing column's hardware (column vessel and trays), the modified Prefractionator's cost will be compared to the existing separation column's cost.

1. On the TPS view, go to the **Performance** tab and select the **Summary** tab.

Figure 7.22

Configuration		TAC	CapEx	OpEx
Dividing Wall Column		1.000	1.000	1.000
Petlyuk Column		1.052	1.608	1.000
Prefractionator		1.090	1.701	1.033
Distributed Sequence		1.139	2.210	1.039
Direct Sequence		1.160	1.682	1.111
Partially Coupled Prefractionator		1.173	1.626	1.131
Side Stripper		1.256	1.753	1.210
Side Rectifier		1.302	1.626	1.272
Indirect Sequence		1.370	1.751	1.334

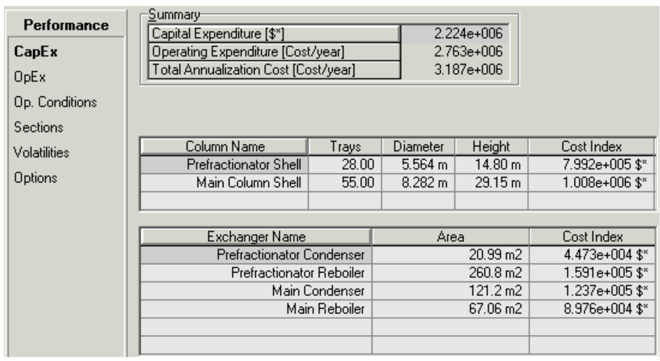
- The Prefractionator's operating cost is only 3.3% more than the lowest operating cost.
- The Prefractionator ranks 3rd in the TAC value.
- The Prefractionator operating cost is about 8% less than the operating cost of the existing separation train configuration.

7.5.5 Checking the Rest of the Column Hardware

The Prefractionator configuration has been chosen and modified so that the existing vessels and trays in the separation train can be reused. The rest of the existing column configuration may have to be changed.

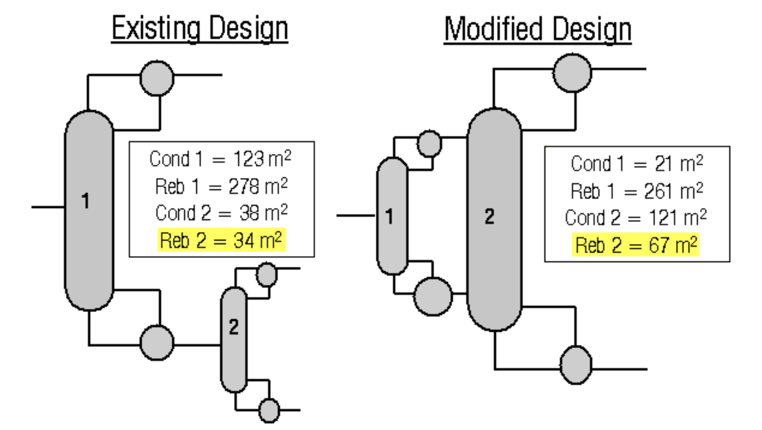
1. Open the Prefractionator view, go to the **Performance** tab and select the **CapEx** page.

Figure 7.23



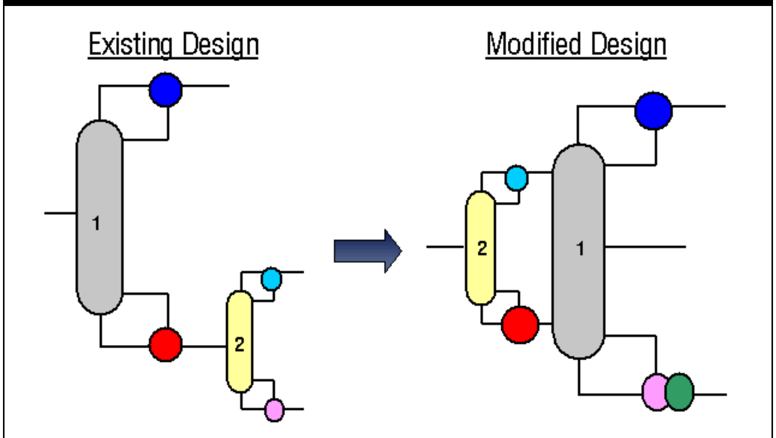
In the Exchangers table, the estimated sizes of the condensers and reboilers in the Prefractionator column are similar to the existing condensers and reboilers, except for the Main/2nd Reboiler.

Figure 7.24



The piping connections/layout of the existing separation train will also need to be modified.

Figure 7.25



In conclusion, the following modifications need to be made to the existing separation train:

- The main/2nd reboiler needs to be replaced with a larger reboiler. The heat transfer area needs to be increased by about 33 m^2.
- The pipes network needs to be modified.

The cost of modifying the pipe network would be small compared to the cost of replacing the reboiler, so the payback time for the cost of the reboiler only will be analyzed.

2. Open the Direct Sequence view.
3. Arrange both Direct Sequence and Prefractionator views so you can see the calculated results from both views.
4. On the Direct Sequence view, click the **Performance** tab, then select the **CapEx** page.

Assume that the calculated operating cost of the Direct Sequence view matches the current separation train operating cost.

5. Subtract the operating cost of the Prefractionator view (**$2.763e^6$/ year**) from the operating cost of the Direct Sequence view (**$2.971e^6$/ year**).
6. The operating cost saved per year is **$2.08e^5$** or **$1.733e^4$** per month.

Assume that the current reboiler can be sold at the capital cost calculated by DISTIL.

7. Subtract the capital cost of the second reboiler of the Direct Sequence view (**5.836^4**)from the second reboiler of the Prefractionator view (**$8.976e^4$**).
8. The cost difference to purchase the reboiler for the Prefractionator is **$3.14e^4$**.

About $31,400 needs to be invested to purchase the new reboiler, and the new separation train (Prefractionator) will save about $17,333 per month in operating costs.

In roughly three months of operation time, the new separation system will pay for itself.

7.6 Simulating the Design in HYSYS

Once you have decided which complex column design is the most promising candidate to for the styrene separation process, you can perform a more rigorous simulation on the selected complex column system using HYSYS.

DISTIL allows you to extract information from the selected complex column from TPS operation and export the information to HYSYS. Once in HYSYS, you can perform a more rigorous simulation.

To simulate the column design in HYSYS, you will require HYSYS 3.0 or higher.

Simulate icon

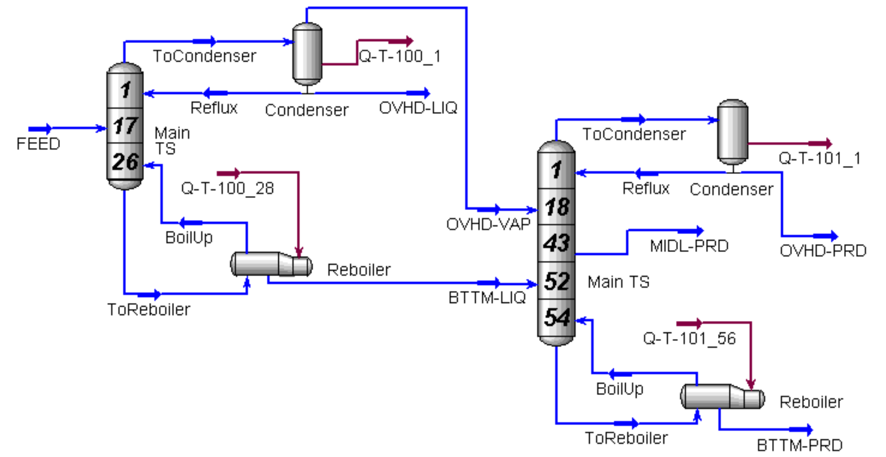
1. To export the Prefractionator column information from DISTIL to HYSYS, open the Prefractionator view, then click the **Simulate** icon.

DISTIL will automatically activate HYSYS, add the fluid package, populate the stream sequence, initialize the column and perform robust covergence operations.

The extraction option performs a seamless data transfer that reduces possible copy/paste errors and time required to replicate the complete column simulation in HYSYS.

2. Double click any of the PFD objects or streams to view the details.

Figure 7.26



The figure is a combination of the PFD view from the column environment of both columns.

Refer to HYSYS manuals for information on using HYSYS.

Once the selected complex column design is exported to HYSYS, some of the values calculated by HYSYS will be different from the DISTIL' values.

The difference in values is because DISTIL assumptions are more relaxed than HYSYS assumptions.

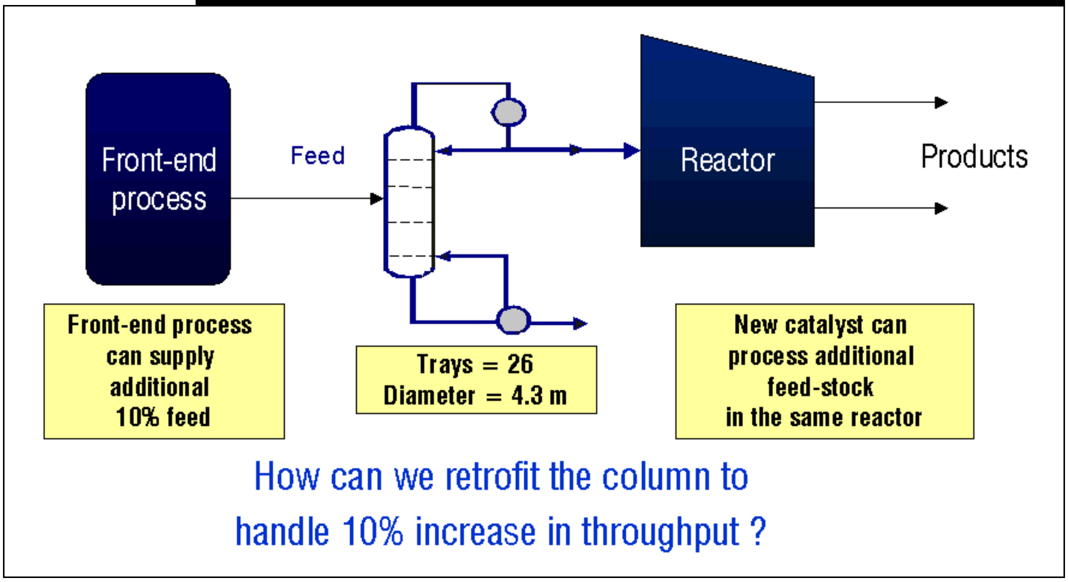
3. From the **File** menu, select **Save As**, and save the HYSYS case as **ImportedStyreneSeparationCase**. HYSYS will save the file with ***.hcs** extension.

8 Increasing Column Throughput

8.1 Introduction

In this tutorial, you will use DISTIL to help you retrofit a column to accommodate for a 10% increase in the throughput.

Figure 8.1



This is an advanced tutorial. It is assumed that you know all the basic functions of DISTIL. For information about the basic functions, refer to the User Guide and Separation manual.

8.1.1 Setting Unit Preferences

Before you begin setting up the simulation, verify that the units currently selected in the DISTIL preferences are the ones you want to use.

For this tutorial, the temperature is in Celsius and the cost per time is in $/year.

1. From the **Tools** menu, select **Preferences**.
2. Click the **Variables** tab, then select the **Units** page.
3. In the Available Unit Sets group, select the **Separation - SI** unit set.

8.1.2 Defining the Fluid Package

To use the Simple Column operation, you have to generate a fluid package containing the property package and components required to simulate the feed stream entering the separation system.

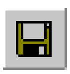
1. Open the Fluid Package Manager.
2. In the Fluid Package Manager view, and create a fluid package with the following parameters and components.

Fluid Package Manager icon

Property Package	Components
Peng-Robinson (Ideal)	C6, C7, C8, C9

8.1.3 Saving the Case

To save the case, do one of the following:



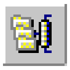
Save Case icon

- Click the **Save Case** icon.
- Select **File-Save** command from the menu bar.
- Select **File-Save As** command from the menu bar.

In the Save Case view, select the saving location and enter the file name for the case. DISTIL will save the file with ***.hcd** extension.

8.2 Installing a Simple Column

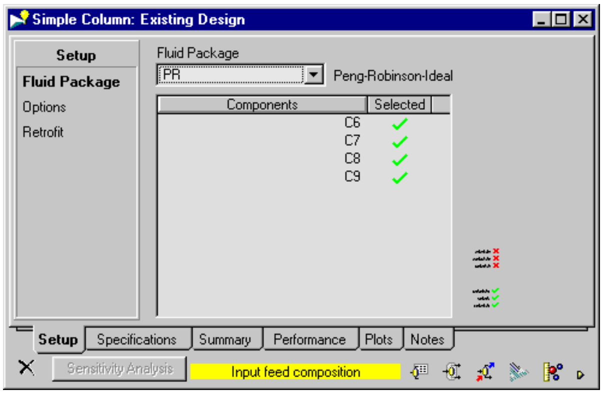
After specifying the fluid package for the simulation, you are ready to use the Simple Column operation.



Separation Technology Manager icon

1. Open the Separation Manager view.
2. In the Separation Manager view, select and add the simple column operation.
3. On the Simple Column view, rename the simple column operation **Existing Design** and select the appropriate fluid package.

Figure 8.2



8.2.1 Modifying the Tray Spacing

In this tutorial, the column's tray spacing is 18 inches. You must check and ensure the assumed value for tray spacing is 18 inches.

1. On the **Setup** tab, **Options** page, click the **Capital Cost** button in the Economics group. The Capital Cost view appears.
2. Click the **Column** tab, and select the **Sizing** page.
3. In the **Tray Spacing** field, check that the value is **18 inch**. If the value is incorrect, use the down arrow and select **18 inch** from the drop-down list.
4. Click the **Close** icon to close the Capital Cost view.

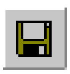
8.2.2 Modifying Available Utilities

In this tutorial, there are only six utilities available in the plant system, so you will remove almost all the utilities from the DISTIL default utility database.

1. Open the Utility Database view associated with the Existing Design view (**Setup** tab, **Options** page).
2. Remove all the utilities except for the following:
    - LP Steam
    - MP Steam
    - HP Steam (Needs to be modified)
    - Very High Temperature
    - Cooling Water
    - Very Low Temperature
3. Modify the HP Steam so the values as follows:
    - Outlet T: 214°C
    - Inlet T: 215°C
    - Cost Index: 8.22e-003 Cost/kW-h

Figure 8.3

Name		Inlet T [C]	Outlet T [C]	HTC [kJ/h-m2-C]	Cost Index [Cost/kW-yr*]	ARH [C]	ARL [C]	DTmin [C]	Viscosity [cP]	Conductivity [W/m-K]	Density [kg/m3]	Eff. Cp [kJ/kg-C]	Type
LP Steam		125.0	124.0	2.160e+004	59.92	115.5	-273.1	10.00	0.013	0.027	558.00	2196.40	Steam Reboiler
MP Steam		175.0	174.0	2.160e+004	69.38	165.5	115.5	10.00	0.015	0.031	493.00	1981.40	Steam Reboiler
HP Steam		215.0	214.0	2.160e+004	72.01	205.5	165.5	10.00	0.018	0.038	421.00	1703.10	Steam Reboiler
Very High Temperature		3000	2999	399.6	280.7	2991	205.5	10.00	0.000	0.000	1.23	1.000	Fired Heater
Cooling Water		20.00	25.00	1.350e+004	6.700	3000	29.50	5.000	1.000	0.603	998.00	4.183	Shell & Tube
Very Low Temperature		-270.0	-269.0	4680	280.7	29.50	-267.5	2.000	0.007	0.012	2.27	1.341	Shell & Tube
New		---	---	---	---	---	---	---	---	---	---	---	Shell & Tube



Save Default Utilities to File icon

4. Click the **Save Default Utilities to File** icon, the Save Heat Integration Defaults view appears.
5. In the **File Name** field, enter **Tutorial8-utildata** and click the **Save** button to save the modified utility database.
6. Click the **Close** icon ☒ to close the Utility Database view.

8.2.3 Specifying Feed and Product Streams

Once you have selected the fluid package and calculation options for the Existing Design view, you will specify the feed and product streams characteristic.

The Feed page allows you to specify the composition, pressure/temperature, and flow rate of the feed stream.

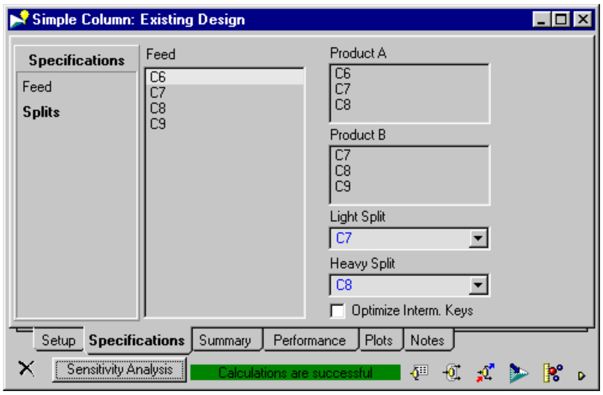
1. Click the **Specifications** tab, then select the **Feed** page.
2. Enter the information as shown in the table below:

Object	Value	Object	Value
C6	0.250	**C9**	0.200
C7	0.200	**Pressure cell**	100 kPa
C8	0.350	**Flowrate cell**	1000 kgmole/h

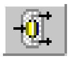
The Splits page allows you to control the component splits in the mixture by selecting the components that will be in each of the two product streams.

3. Select the **Splits** page.
4. Define the splits information as follows:
   - Light Split: **C7**
   - Heavy Split: **C8**

Figure 8.4



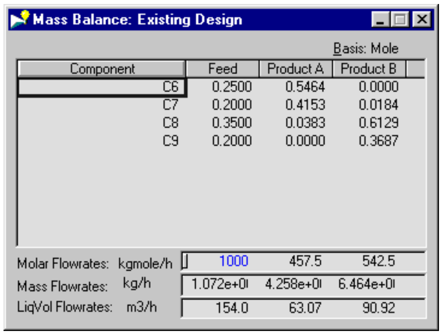
DISTIL automatically begins calculating the variable values of the complex columns based on the specified product streams.



Recovery Matrix icon

5. Click the **Overall Mass Balance** icon at the bottom right corner of the Simple Column view to open the Mass Balance view.

Figure 8.5



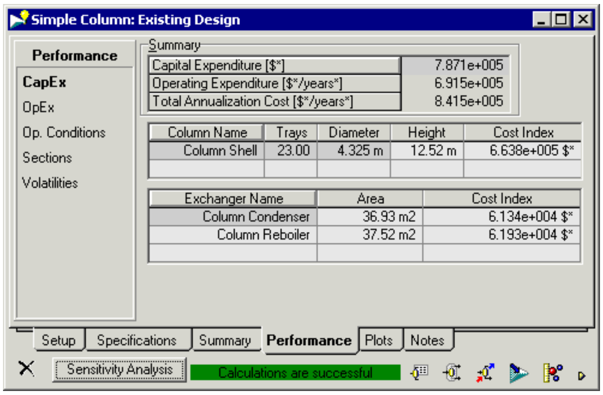
The Mass Balance view displays the molar, mass, and liquid volume flow rate of the feed and product streams.

8.2.4 Replicating the Existing Simple Column

The existing column has 26 trays and the tray diameter is 4.3 m. The number of trays and tray diameter in the Existing Design view needs to be checked and/or modified to match.

1. Click the **Performance** tab, then select the **CapEx** page.

Figure 8.6



The Existing Design column contains only 23 trays and shows 4.325 m for the tray diameter.

The Existing Design's tray diameter is just a little higher than the existing column's tray diameter, so it shall be left alone. The number of trays in the Existing Design view, however, needs to be modified as it is three trays short.

Refer to **Section 5.2.4 - Column Retrofit Options** in the **Reference Guide** for the detailed relationship between reflux ratio and number of trays in the column.

The number of trays in the Existing Design column needs to be increased. The number of trays in the column has an inverse relationship with the reflux ratio value of the column, so decreasing the reflux ratio will increase the number of trays in the column.

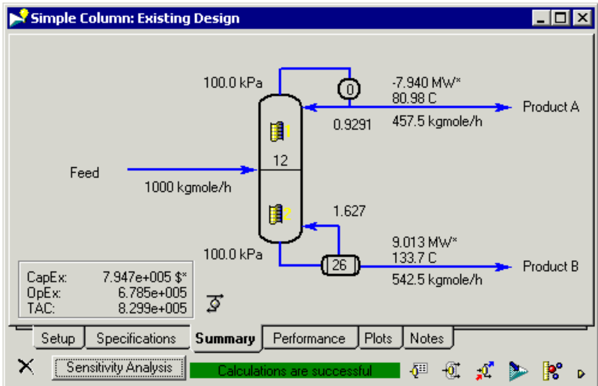
The Simple Column operation in DISTIL contains an option that will automatically calculate/modify the reflux ratio based on the number of trays you specified for the column.

2. Click the **Setup** tab, then select the **Retrofit** page.

3. Enter **26** in the **Target # of Trays** field, then click the **Modify R/RMin** button to begin calculation of the reflux ratio for the specified number of trays.

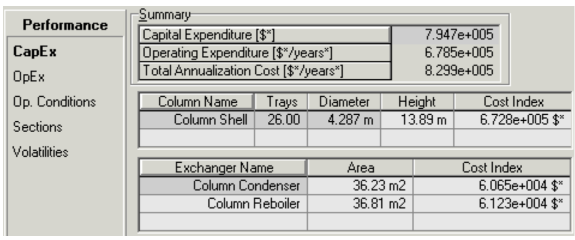
On the Summary tab, the column in the Existing Design view now has 26 trays.

Figure 8.7



Check the rest of the column's parameter to make sure they match the existing column parameters.

Figure 8.8



The tray diameter is now 4.287 m.

The heat transfer area of the reboiler is 36.23 m^2 and the heat transfer area of the condenser is 36.81 m^2.

The second Simple Column operation will be the column with the increased feed flow rate.

4. Repeat steps #1 in **Section 8.2 - Installing a Simple Column** to the step above (#3) and generate another Simple Column operation which will be the same as the existing column.
5. Name the replicated Simple Column operation **Modified Design**.

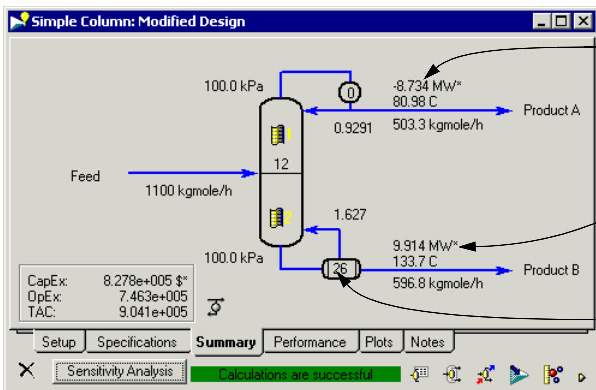
8.3 Increasing the Feed Flow Rate

The simple column operation will automatically make modifications to the column configuration to handle the increase in feed flow rate.

Now that there are two Simple Column operations replicating the existing column, the Modified Design column's feed stream flow rate will be increased by 10%.

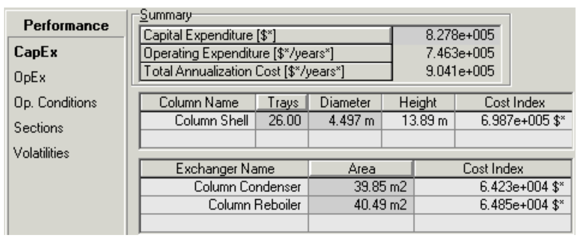
1. On the Modified Design view, enter **1100 kgmole/h** for the feed flow rate. (**Specifications** tab, **Feed** page)

Figure 8.9



2. Click the **Performance** tab, then select the **CapEx** page.

Figure 8.10



The tray diameter has increased to 4.497 m.

The reboiler and condenser heat transfer area has increased to 39.85 m^2 and 40.49 m^2, respectively.

The tray diameter in the Modified Design column has been increased to compensate for the new feed flow rate. Unfortunately, the cost of replacing the column vessel and trays is not cost effective. Some other cost effective column parameter needs to be modified to reduce the tray diameter.

8.3.1 Modifying the Pressure

To reduce the tray diameter, consider the factors that affect the tray diameter:

- **Vapour Density**. This variable can be manipulated. The value is not fixed.
- **Vapour/Liquid Traffic**. This variable cannot be manipulated. The value is fixed, due to the required/specified composition of the product streams.

The vapour density has an inverse relationship with the tray diameter:

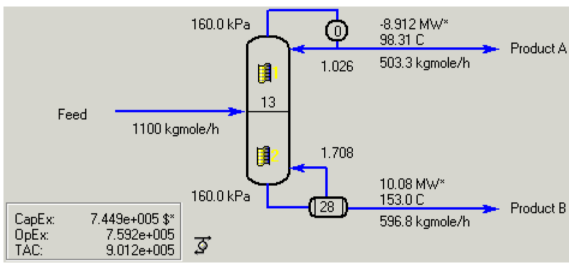
- When the vapour density value is small, the vapour in the column requires more space/volume, so the tray diameter needs to be large to increase the volume of the column.
- When the vapour density value is large, the vapour in the column requires less space/volume, so the tray diameter needs to be small to reduce the volume of the column.

In this tutorial, the vapour density value will be manipulated to reduce the tray diameter. The column parameter that manipulates the vapour density is the operating pressure. Increasing the operating pressure will increase the vapour density, and decreasing the operating pressure will decrease the vapour density.

In this tutorial the operating pressure will be increased to increase the vapour density value, and thus reduce the tray diameter.

1. On the Modified Design view, click the **Summary** tab.
2. Open the Section 2 view, and increase the operating pressure of the column to **160 kPa**. The following changes occur.
   - The tray diameter changes to 4.29 m.
   - The number of trays increases to 28.

Figure 8.11



When the operating pressure of the column increases, the difficulty in the separation process also increases, so more trays are required in the column to achieve the required separation.

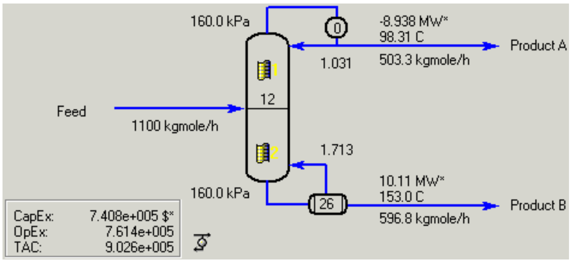
The Simple Column operation automatically increases the number of trays in the column to compensate for the increased difficulty in the separation process.

8.3.2 Modifying the Reflux Ratio

In the existing column have 26 trays, so the modified column is limited to 26 trays. The automatic modification of the reflux ratio option will be used again to reduce the number of trays in the column.

1. On the Modified Design view, click the **Modify R/RMin** button.

Figure 8.12



The Modified Design column contains the correct number of trays and tray diameter, and is a feasible separation system even when the feed stream flow rate has increased by 10%. The energy required by the reboiler and condenser has not changed much either.

The rest of the column, however, should be checked to see if there were any adverse modifications made to the column configuration to meet the required separation specifications.

2. On the Modified Design view, go to the **Performance** tab and select the **CapEx** page.
3. On the Existing Design view, go to the **Performance** tab and select the **CapEx** page.

Figure 8.13

Existing Design

Summary

Capital Expenditure [$*]	7.947e+005
Operating Expenditure [$*/years*]	6.785e+005
Total Annualization Cost [$*/years*]	8.299e+005

Column Name	Trays	Diameter	Height	Cost Index
Column Shell	26.00	4.287 m	13.89 m	6.728e+005 $*

Exchanger Name	Area	Cost Index
Column Condenser	36.23 m2	6.065e+004 $*
Column Reboiler	36.81 m2	6.123e+004 $*

Modified Design

Summary

Capital Expenditure [$*]	7.408e+005
Operating Expenditure [$*/years*]	7.614e+005
Total Annualization Cost [$*/years*]	9.026e+005

Column Name	Trays	Diameter	Height	Cost Index
Column Shell	26.00	4.296 m	13.89 m	5.852e+005 $*

Exchanger Name	Area	Cost Index
Column Condenser	31.45 m2	5.583e+004 $*
Column Reboiler	78.24 m2	9.982e+004 $*

In comparing the heat transfer area between the Modified Design and Existing Design, you will observe the following:

- The heat transfer area of the condenser from the Modified Design is slightly less than the Existing Design (31.45 m^2 < 36.23 m^2).
- The heat transfer area of the reboiler from the Modified Design is a lot more than the Existing Design (78.24 m^2 > 36.81 m^2).

The reason for slight decrease and large increase in heat transfer area can be deduced through the temperature differences between the streams temperatures and the utilities temperatures.

4. On the Existing Design view, select the **Op. Conditions** page.
5. On the Modified Design view, select the **Op. Conditions** page.

Figure 8.14

Simple Column: Existing Design

Performance: CapEx, OpEx, **Op. Conditions**, Sections, Volatilities

Section	1	2
Top Pressure [kPa]	100.0	100.0
Bottom Pressure [kPa]	100.0	100.0
Pressure Drop [kPa]	0.0000	0.0000
Top Temperature [C]	80.98	---
Bottom Temperature [C]	---	133.7
R/Rmin	1.058	***
Utility Used	ooling Water	MP Steam
Duty [MW*]	-7.940	9.013
Shaftwork [MW*]	0.0000	0.0000

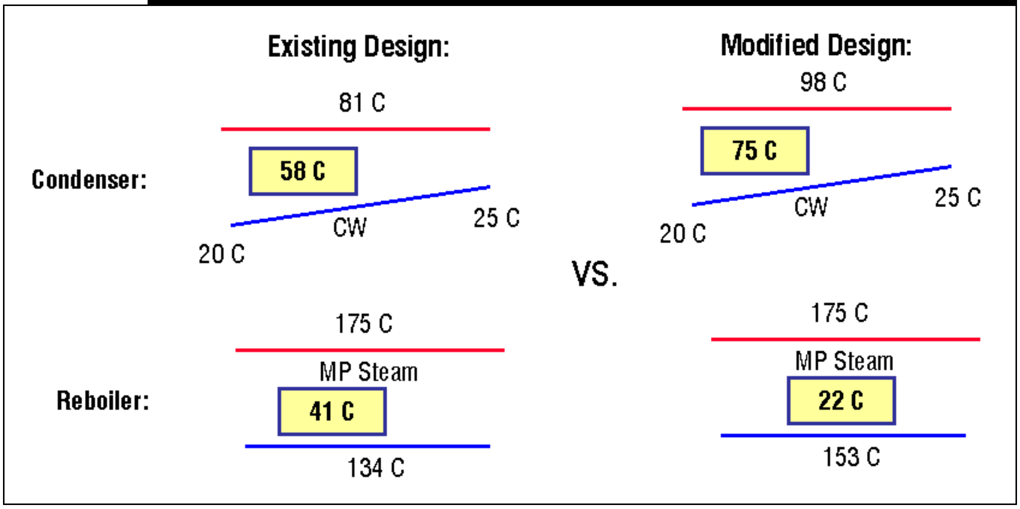
Simple Column: Modified Design

Performance: CapEx, OpEx, **Op. Conditions**, Sections, Volatilities

Section	1	2
Top Pressure [kPa]	160.0	160.0
Bottom Pressure [kPa]	160.0	160.0
Pressure Drop [kPa]	0.0000	0.0000
Top Temperature [C]	98.31	---
Bottom Temperature [C]	---	153.0
R/Rmin	1.064	***
Utility Used	ooling Water	MP Steam
Duty [MW*]	-8.938	10.11
Shaftwork [MW*]	0.0000	0.0000

6. Compare the temperature differences, in both reboiler and condenser, for the existing column vs. the simple column.

Figure 8.15

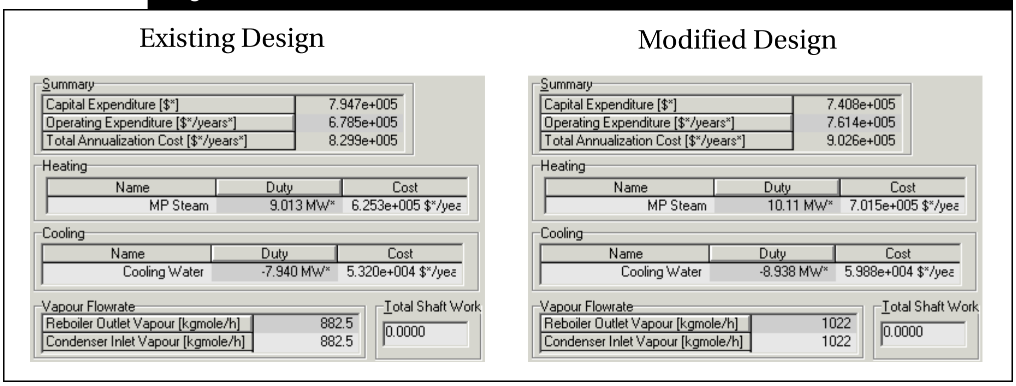


From the above figure you can observe the following:

- The temperature difference in the condenser has increased slightly, so the driving force in the condenser also increases. A slightly smaller heat transfer area is required to accomplish the heat transfer.
- The temperature difference in the reboiler has decreased by half, so the driving force in the reboiler also decreases. A larger heat transfer area is required to achieve the heat transfer.

7. On the Existing Design view, select the **OpEx** page.
8. On the Modified Design view, select the **OpEx** page.

Figure 8.16



9. Compare the total operating cost between the Modified Design and the Existing Design.

The total operating cost for the Existing Design view is $6.785e^5/year, and the total operating cost for the SC1 view is $7.614e^5/year, so the total operating cost needs to be increased by about 12% to accommodate the 10% increase in feed flow rate.

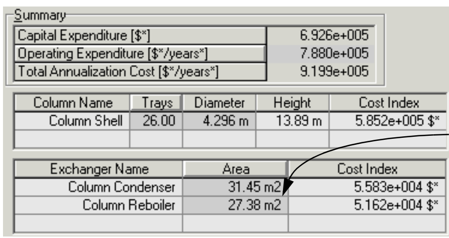
8.3.3 Changing Utilities

The need to change the reboiler to a larger reboiler may not be recommended due to lack of space or cost effectiveness.

To continue using the same reboiler, the utility used in the reboiler will be changed to a hotter utility. If a hotter utility is used, there is a greater driving force in heat transfer, reducing the heat transfer area required.

1. On the Modified Design view, go to the **Performance** tab and select the **Op. Conditions** page.
2. Change the utility used in Section 2 to **HP Steam**.
3. Click the **CapEx** page.

Figure 8.17


Summary

Capital Expenditure [$*]	6.926e+005
Operating Expenditure [$*/years*]	7.880e+005
Total Annualization Cost [$*/years*]	9.199e+005

Column Name	Trays	Diameter	Height	Cost Index
Column Shell	26.00	4.296 m	13.89 m	5.852e+005 $*

Exchanger Name	Area	Cost Index
Column Condenser	31.45 m2	5.583e+004 $*
Column Reboiler	27.38 m2	5.162e+004 $*

The change in utility from MP Steam to HP Steam has reduce the reboiler heat transfer area to 27.38 m^2, removing the need for capital investment for a new reboiler.

The change in utility would also affect the operating cost because the cost of HP Steam utility is slightly higher than the cost of MP Steam utility. The total operating cost, when HP Steam is used, increases to $7.880e^5/year.

In comparing the total operating cost between the Modified Design and the Existing Design, the total operating cost increases by about 16.1% to accommodate the 10% increase in feed flow rate.

8.4 Comparing Options

DISTIL allows you to evaluate different options quantitatively, present different options with minimal effort, and access information quickly and efficiently.

8.4.1 Option 1

Based on the analysis from **Section 8.3.2 - Modifying the Reflux Ratio**, one of the options to accommodate the 10% increase in feed flow rate is as follows:

- Increase the operating cost by 12%.
- Increase the heat transfer area of the reboiler to 78.24 m^2.

This option requires a minimum capital investment of $38, 590 to purchase the new reboiler, assuming the existing reboiler is sold at the suggested price.

Figure 8.18

Existing Design heat exchanger capital cost

Exchanger Name	Area	Cost Index
Column Condenser	36.23 m2	6.065e+004 $*
Column Reboiler	36.81 m2	6.123e+004 $*

SC1 heat exchanger capital cost

Exchanger Name	Area	Cost Index
Column Condenser	31.45 m2	5.583e+004 $*
Column Reboiler	78.24 m2	9.982e+004 $*

8.4.2 Option 2

Based on the analysis from **Section 8.3.3 - Changing Utilities**, another option to accommodate the 10% increase in feed flow rate is as follows:

- Change the reboiler utility to HP Steam.
- Increase the operating cost by 16.1%.

This option requires no capital investment, however, there is a greater increase in the operating cost.