Getting Started

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
01 Getting Started

Workshop

In the Getting Started module you will build the flowsheet around which most of the rest of this course is based.

The process is a Turbo Expander plant. LPG Product is obtained from a Feed Natural Gas stream by cooling, expanding, separating and distillation. The remaining gas is then re-compressed for export.

Learning Objectives

Once you have completed this section you should have revised your basic HYSYS knowledge. You will also be able to:

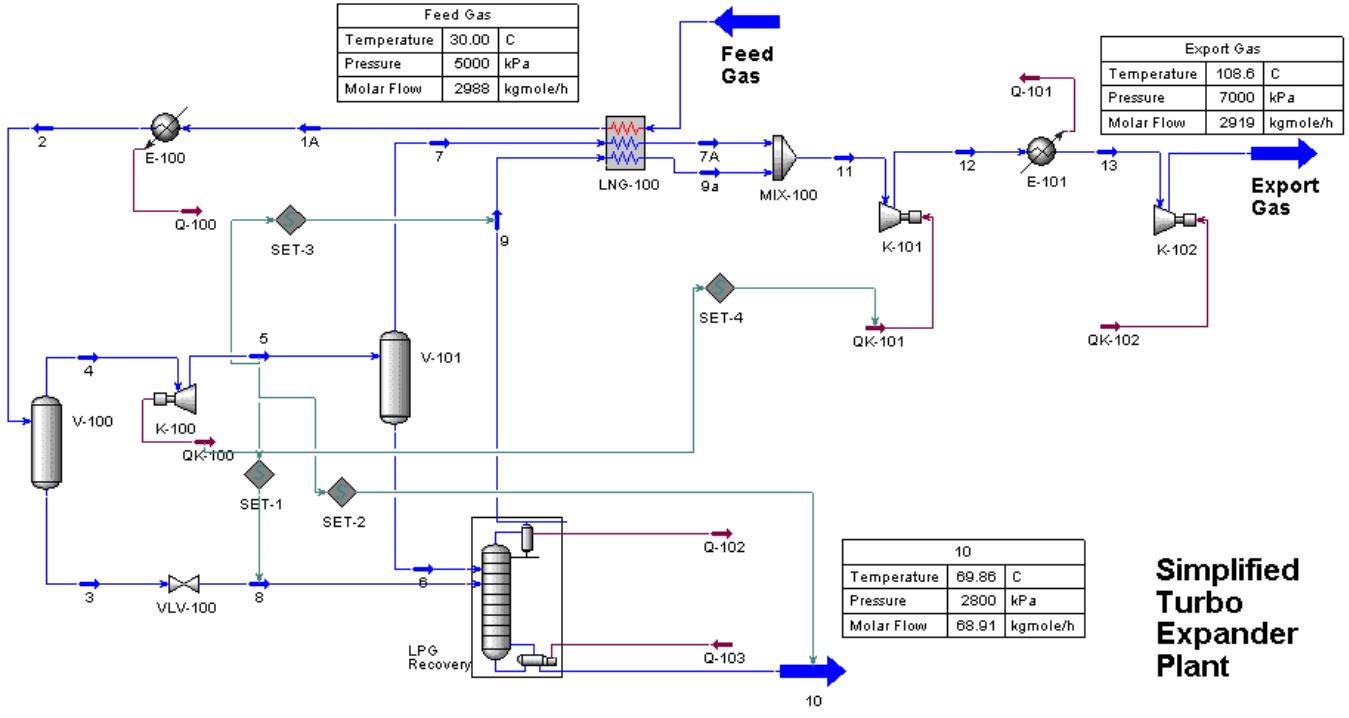
- Use the HYSYS LNG Exchanger to simulate multi-pass exchangers
- Add Columns using the Input Experts
- Add extra specifications to columns
- Customize the Workbook and PFD
- Use stream property correlations

Prerequisites

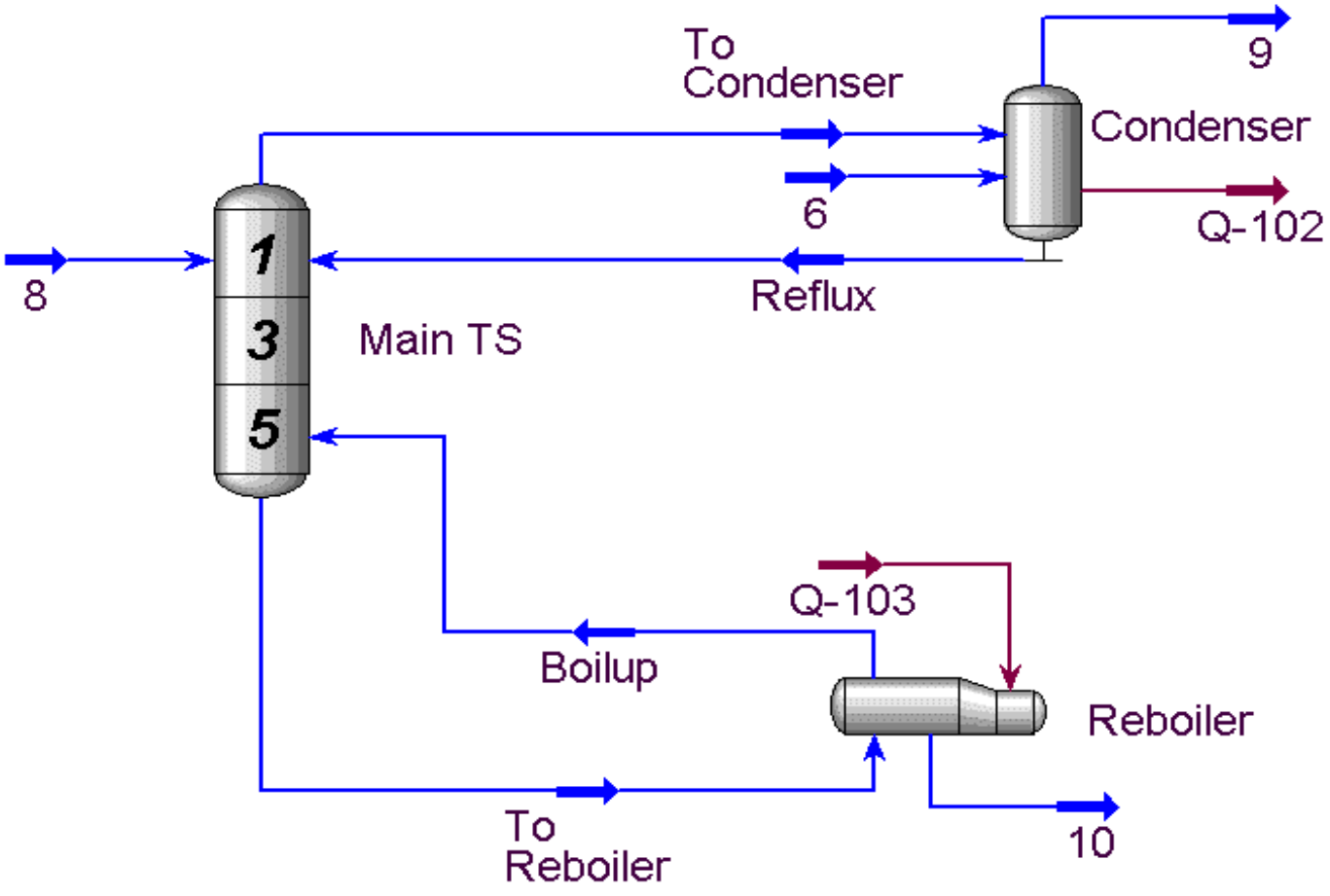
This course is aimed at people who have had some experience of using HYSYS before. Hence the instructions are deliberately brief in places where previous HYSYS users are likely to already know what to do.

If you have problems at any stage you can ask the course instructor.

Process Overview



Column Overview



Building the Simulation

Some stream and operation names can be obtained by referring to the PFD at the start of this section.

Defining the Simulation Basis

1. Start a new case using the **Peng Robinson** Equation of State.
2. Use the following components: Nitrogen, CO2, Methane, Ethane, Propane, i-Butane, n-Butane, i-Pentane, n-Pentane and Hexane.
3. Enter the **Main Simulation Environment**.

Add the Feed Gas Stream

4. The **Feed Gas** stream has the following conditions and composition:

Name	Feed Gas
Temperature	30°C (86°F)
Pressure	5000 kPa (725.19psia)
Flow rate	2988 kgmole/h (6587.3 lbmole/hr)

Component	Mole Fraction
Nitrogen	0.0149
CO2	0.002
Methane	0.9122
Ethane	0.0496
Propane	0.0148
i-Butane	0.0026
n-Butane	0.002
i-Pentane	0.001
n-Pentane	0.0006
n-Hexane	0.0003

Add the Multi-pass Exchanger



LNG icon

Multi-pass exchangers are known as LNG Exchangers in HYSYS.

The **LNG** (Liquefied Natural Gas) exchanger model solves heat and material balances for multi-stream heat exchangers and heat exchanger networks. The solution method can handle a wide variety of specified and unknown variables.

The LNG allows for multiple streams, while the HEAT EXCHANGER allows only one hot side stream and one cold side stream.



The Tube Side and Shell Side streams can come from different Flowsheets. This is one way of using different fluid packages on each side of the exchanger

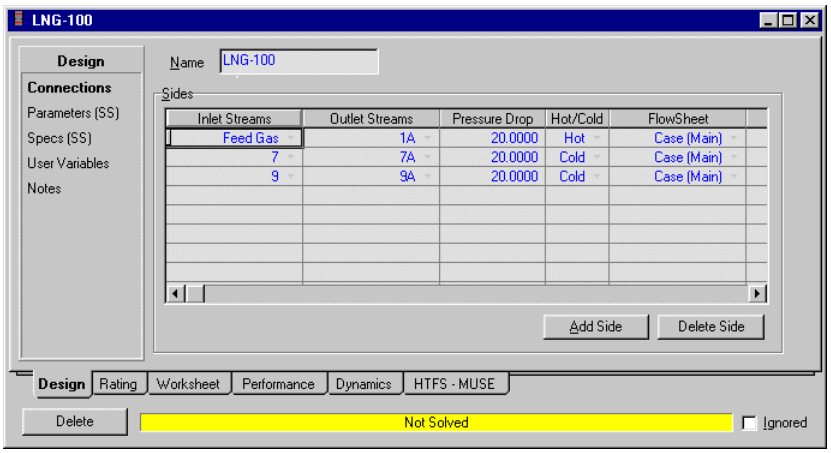
For the LNG exchanger, you can specify various parameters, including heat leak/heat loss, UA or temperature approaches. Two solution methods are employed; in the case of a single unknown, the solution is calculated directly from an energy balance. In the case of multiple unknowns, an iterative approach is used which attempts to determine the solution that satisfies not only the energy balance, but also any constraints, such as temperature approach or UA.

5. Add a **LNG** operation.
6. Select the **Connections** page on the **Design** tab and enter the following information:



Any number of Sides may be added simply by selecting the Add Side Button. To remove a side, select the Delete Side button after positioning the cursor in the appropriate row.

Figure 1



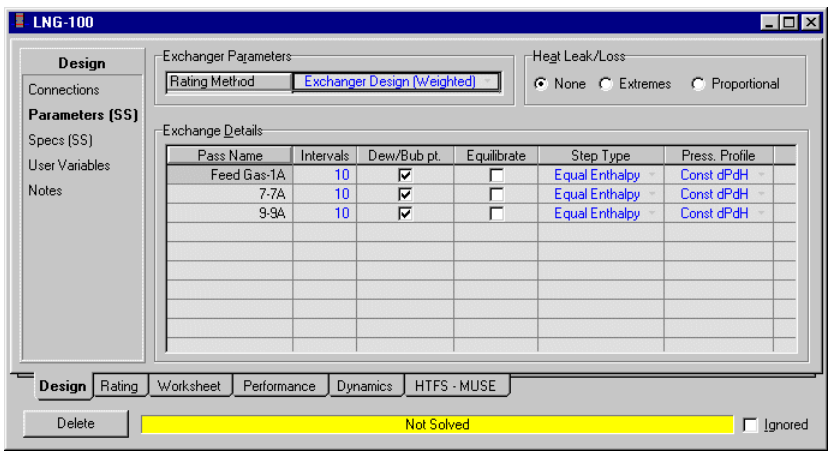
If you prefer you can add the other streams to the flowsheet manually. Alternatively typing the name of a new stream into the Sides box will create it.

For each exchanger side:

- An inlet stream and outlet stream are required.
- A **Pressure Drop** is required.
- The **Hot/Cold** designation can be specified. This will be used as an estimate for calculations and will also be used for drawing the PFD. If a designated hot pass is actually cold (or vice versa), the operation will still solve properly. The actual Hot/Cold designation (as determined by the LNG) can be found on the **Performance** tab in the **Results** page.

7. Switch to the **Parameters (SS)** page. These values will be left unchanged.

Figure 2





The Weighted method is required for exchangers with more than two sides.

The **Exchanger Design (Weighted)** method is used to split the heating curves into intervals. (Rather than simply considering the end points) The number of intervals is set in the Exchange Details table. The **Step Type** parameter sets whether HYSYS splits the curves by temperature, enthalpy or automatically to minimise the errors.

By ticking the **Dew/Bub pt.** checkboxes HYSYS will include points on the heating curves for any phase changes that occur. The **Press. Profile** options set how HYSYS decides the pressure profile while iterating.

Heat Losses will not be considered. All streams on the same side with the **Equilibrate** checkbox checked are considered to be in equilibrium before entering the exchanger calculations. Thus the driving force will be reduced.

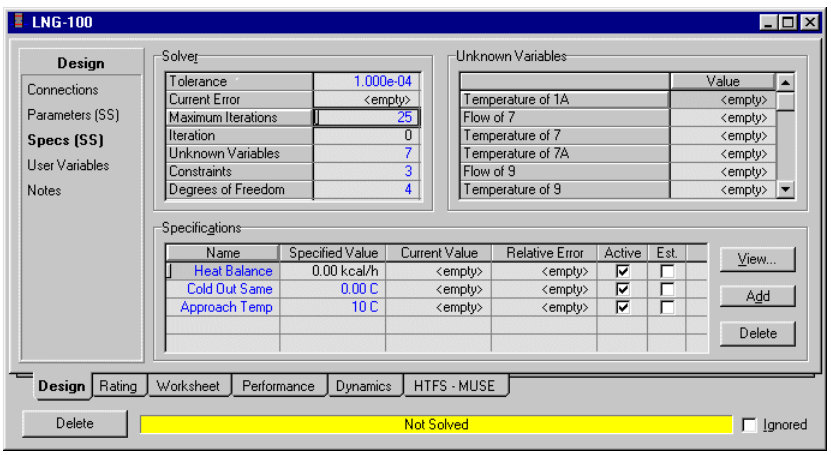
These options are discussed in more detail in the HYSYS Operations Guide. This is a good time to learn how to access/search the Acrobat PDF documentation.

8. Go to the **Specs (SS)** page.

Two extra specifications (in addition to the Heat Balance spec - i.e., conservation of energy) will be added.

Just after the streams have been added the **Degrees of Freedom** display shows 6 (i.e., 7 unknown variables, 1 constraint so far). By adding two new specs this is reduced to 4. Later on when more of the flowsheet is built, these 4 degrees of freedom will be used up and the exchanger will have enough information to solve.

Figure 3

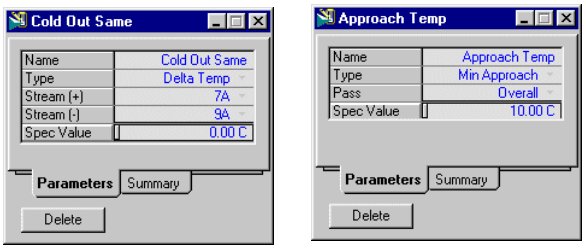


The two specs that will be added are:

- Cold Stream Outlet temperatures are the same (0°C or 32°F)
- Minimum approach temperature in the exchanger (10°C or 50°F)

9. Add these two specs as follows:

Figure 4



LNG exchangers, like normal HYSYS Heat Exchangers and Columns, allow the user to enter as many specifications as required. Only the active specifications are used to solve the operation.

If the **Est.** checkbox is checked then HYSYS will use the spec value as an initial estimate in its iterative solution. Hence specifications can be used for more than just being an active spec. They can:

- Provide an initial estimate only (Uncheck **Active**, Check **Est.**)
- Be used to monitor how important variables change as the operation iterates (Uncheck **Active** and **Est.**)

Add a Cooler

10. Add a **Cooler** to stream 1A. This should have the following parameters:

Name	E-100
Cooler Outlet	Stream 2
Pressure Drop	20 kPa (2.9psia)
Outlet Temperature	-62°C (-79.6°F)
Energy	Q-100

Note that in a later module the Cooler operation will be linked to a HYSYS sub-flowsheet that represents an ethane/propane refrigeration loop.

The Cooler outlet Stream 2 should now be fully defined.

Add a Separator

11. The Cooler product Stream 2 should be sent to a Separator vessel.
12. The liquid product is Stream 4 and the vapour product is Stream 3.

Add an Expander

13. The Separator vapour is then expanded to 2800 kPa.
14. The outlet is Stream 5.
15. The Expander should have the following properties:

Name	K-100
Outlet Pressure	2800 kPa (406 psia)
Adiabatic Efficiency	75% (This is the default)
Energy	QK-100

Add a Separator

16. Add another **Separator** to the Expander outlet.
17. The feed is the outlet from the Expander.
18. The vapour product is Stream 7 which has already been added and linked to the LNG exchanger.
19. The liquid product is Stream 6.

Add a Valve and Set

20. The liquid from the first Separator is flashed down to the same pressure as the Expander outlet. Add a HYSYS **Valve** operation to do this.
21. The valve outlet is Stream 8.

When the simulation is manipulated later the Turbo Expander outlet pressure is one of the process parameters that will be changed, hence it makes sense to link these parameters within HYSYS rather than manually changing both.

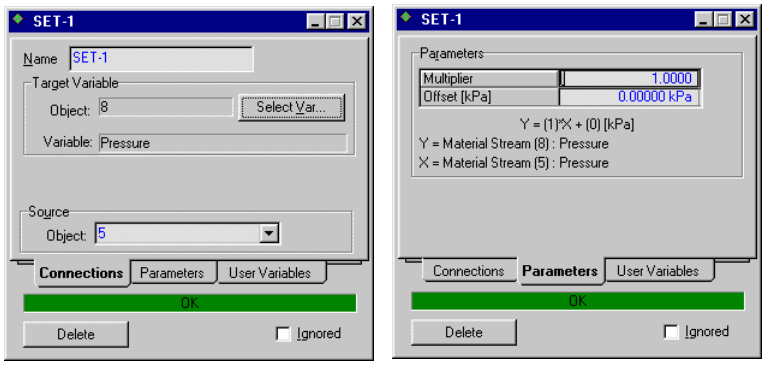
This is done using a HYSYS **Set** operation. This relates two HYSYS parameters. It can be used to make them identical or to force a fixed Multiplier and Offset between them.


Set icon

22. Install a **Set** with the following parameters:

Name	SET-1
Target Variable	Stream 8 Pressure
Source	Stream 5
Multiplier	1
Offset	0 kPa (0 psia)

Figure 5



LPG Recovery Column

This will be simulated using the HYSYS Distillation Column operation. This column has 5 stages, a Condenser and a Reboiler. Stream 8 is fed to the top tray and stream 6 is fed to the Condenser.

Rather than defining the column pressures within the column operation, as would be normal practice, HYSYS Set operations will be used to relate the column top and bottom pressures to the Turbo Expander outlet stream.

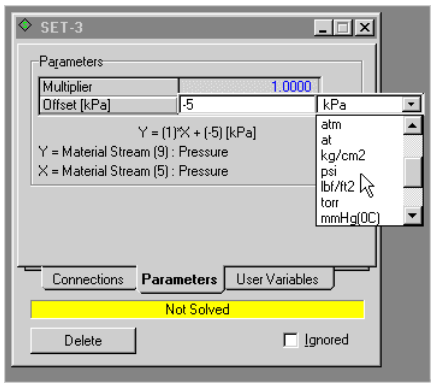
The purpose of this is to allow the flowsheet to be manipulated later by changing several key process parameters (including the Turbo Expander outlet pressure).

Set the Column Pressures

23. First add the bottoms liquid product stream to the PFD. (Stream 10).
24. Install a **Set** operation to 'Set' the pressure of Stream 10, so that it is the same as Stream 5.
25. Install another **Set** to make Stream 9's pressure 5 psi less than Stream 5's.

If psi is not the pressure unit HYSYS is currently using you can enter a psi value by using the unit drop-down to the right for the number entry field.

Figure 6



Add the Column



The Input Experts provide the new user with step-by-step instruction for defining a column. They can be switched off in the HYSYS Preferences.

26. Add a **Distillation Column** operation.

By default HYSYS includes an 'Input Expert' to guide the user through setting up the column. The Input Expert has four pages, you must enter all the required information on each page, before the **Next >** button will become active.

27. Double-click the **Column PFD** icon to start the Input Expert.
28. Enter the following information on the **Connections** page:



Distillation Column icon

Connections	
Column Name	LPG Recovery
No. of Stages	5
Feed Stream / Stage	8 / at stage 1
Condenser Type	Full Reflux
Overhead Vapour Product	9
Bottom Product	10
Condenser Energy Stream	Q-102
Reboiler Energy Stream	Q-103

29. Since the product stream pressures have already been set (with the Set operations), the **Pressure Profile** page automatically picks these up. Go to the **Optional Estimates** page by clicking the **Next >** button.

Temperature estimates are used to help the column come to a solution. If you already have an idea of the conditions in the column (e.g. if you are modeling an existing plant) then you can enter the information. In this case the temperatures are not known, so these estimates will be left blank.

30. Click the **Next >** button to move to the final page.

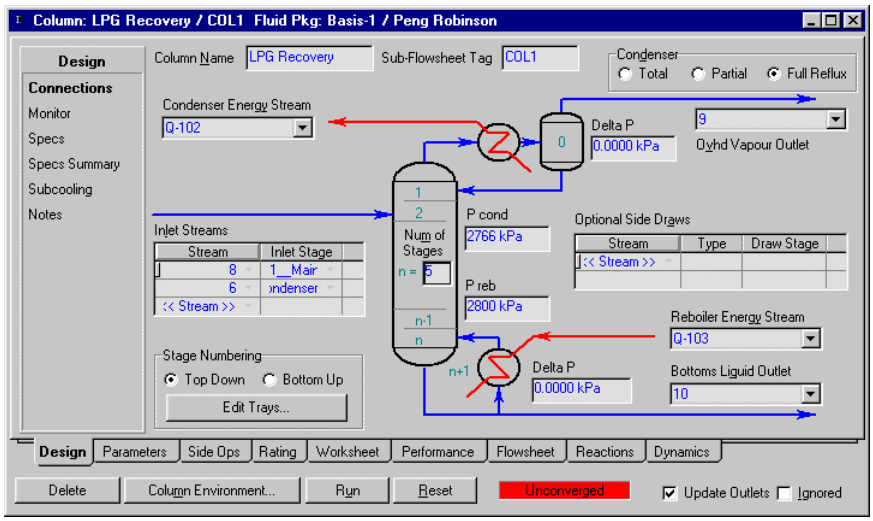


Once the Input Expert has been completed it cannot be accessed again unless the column is deleted and added back. However all the information that was entered can be changed using the column property view.

The final page of the Column Input Expert always has some typical specifications for the particular type of column. In this case, different specifications will be used. These must be added after the Input Expert has been completed.

31. Click the **Done** button to complete the Input Expert. The Column Property View will now appear.

Figure 7

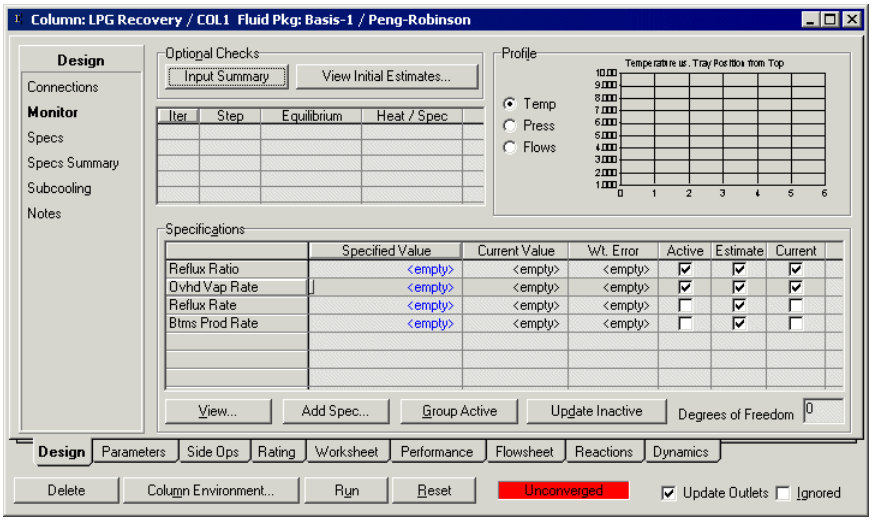


Before the column is ready to run, some more information must be entered:

- Additional Feed Stream
- New Specifications

32. Connect Stream 6 up in the **Inlet Streams** box. Stream 6 should enter the Condenser.
33. Go to the **Monitor** page on the **Design** tab. The existing column specs are visible.

Figure 8



Since no values were entered for the Specs in the Input Expert, no values are present here.

Note that since the column has fully defined feeds, the pressures are known, and 2 specs are activated the **Degrees of Freedom** display is showing 0.

This column will actually be solved to two different specifications.

- Condenser Duty of zero. Stream 6, the cold expanded liquid stream, enters the condenser so provides the necessary cooling effect.
- LPG product RVP (Reid Vapour Pressure) specification.



RVP (Reid Vapour Pressure) is a volatility measure commonly used in the Refining industry. It is the pressure at which the vapour and liquid have a 4:1 volume ratio at 100°F.

34. Add these two specs as follows:

First Spec	
Type	Column Duty
Name	Ovhd Duty
Energy Stream	Q-102 @ Col1
Spec Value	0.001 kJ/h

Second Spec	
Type	Column Vapour Pressure Spec
Name	Btms RVP
Stage	Reboiler
Type	Reid Vapour Pressure
Phase	Liquid
Spec Value	200 psia

An estimate of the column overhead vapour rate will also be provided. One of the specs already added by HYSYS will be used.

35. Modify the existing Ovhd Vap Rate spec as follows:

Existing Vent Rate Spec	
Name	Ovhd Vap Rate
Draw	9 @ Col1
Flow Basis	Molar
Spec Value	1600 lbmole/hr

36. Ensure the Ovhd Duty and Btms RVP specs are set to be **Active** and Estimates, and that the Ovhd Vap Rate spec is set as an **Estimate**.

The other unused specs can be deleted by going to the **Specs (SS)** page on the **Design** tab and pressing the **Delete** button, or by double-clicking the spec on the **Monitor** page on the **Design** tab and clicking **Delete**.

37. If the column hasn't run automatically, click the **Run** button. You should see some messages in the Trace Window (bottom right white pane) as the column solves. When the column has solved, the LNG should then be solved as now both Streams 7 and 9 are fully defined.



What is the flow rate of the Overhead vapour stream?

_____

Was the 'Vent Rate' estimate a good one?

_____

Completing the Simulation

The simulation is now almost complete. The two product gas streams from the LNG must be mixed and re-compressed for export.

38. Mix Stream 7A and 9A using a Mixer operation. The outlet is Stream 11.
39. Add a **Compressor**, keep the standard Adiabatic Efficiency of 75%. The outlet is Stream 12 and the energy is QK-101.
40. Install another **Set** to link the heat flow of the compressor energy stream (QK-101) to the Expander energy stream (QK-100).
41. Add another **Cooler** to cool the mixed compressed gases to 30°C. The Cooler has a 0.2 bar pressure drop.
42. Finally, install an Export Gas **Compressor** to take the product gas to 70 bar. The compressor has an Adiabatic Efficiency of 75%.

Save your case!

Compressor Degrees of Freedom

In this HYSYS case there are two compressors specified differently:

- K-101 has a fully specified inlet stream and a specified duty.
- K-102 has a fully specified inlet stream and a specified outlet pressure.

HYSYS can also solve for a flow rate given an otherwise fully specified feed stream, a duty and an outlet pressure.

Additionally HYSYS compressors can be supplied with head and efficiency curves. This is covered in a later module.

Advanced Modeling

The Column is a special type of sub-flowsheet in HYSYS. Sub-flowsheets contain equipment and streams, and exchange information with the parent flowsheet through the connected streams. From the main environment, the Column appears as a single, multi-feed multi-product operation. In many cases, you can treat the Column in exactly that manner.

The Column Sub-flowsheet provides a number of advantages:



The presence of the green 'Up Arrow' icon in the Button Bar and the Environment: Name (COL1) indicates that you are in the Column Sub-flowsheet.

The Object Palette is different in the Column Sub-flowsheet.

- **Isolation of the Column Solver** - The Column Build Environment allows you to make changes and focus on the Column without the re-calculation of the entire flowsheet.
- **Optional use of different Fluid Packages** - HYSYS allows you to specify a unique (different from the Main Environment) fluid package for the Column Sub-flowsheet. This may be useful in instances when a different fluid package is better suited to the Column (Gas Plant using PR may contain an Amine Contactor that needs to use the Amines Property Package), or the Column does not use all of the components used in the Main Flowsheet and so by decreasing the number of components in the column you may speed up column convergence.
- **Construction of custom templates** - In addition to the default column configurations, which are available as templates, you may define column set-ups with varying degrees of complexity. Complex custom columns and multiple columns may be simulated within a single sub-flowsheet using various combinations of SubFlowsheet equipment. Custom column



Enter Parent Simulation Environment icon

examples include, replacement of the standard condenser with a heat exchanger, or the standard kettle reboiler with a thermosyphon reboiler.

- **Ability to solve multiple towers simultaneously** - The Column Sub-flowsheet uses a simultaneous solver whereby all operations within the sub-flowsheet are solved simultaneously. The simultaneous solver permits the user to install multiple interconnected columns within the sub-flowsheet without the need for Recycle blocks.

You can enter the Column Sub-flowsheet by clicking the **Column Environment** button on the Column Property View. Once inside the Column Environment you can return to the Parent Environment by clicking either the **Parent Environment** button on the Column Runner view or the **Enter Parent Simulation Environment** button in the Button Bar.

Customizing the Workbook and PFD

HYSYS allows the user to customize the Workbook and PFD.

Customizing the Workbook

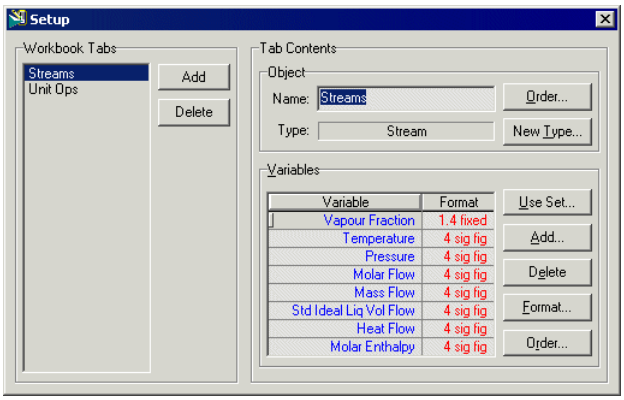


Workbook button

43. Show the workbook by pressing the **Workbook** button on the toolbar (or by using the **Workbook** option on the **Tools** menu).

44. From the **Workbook** menu, select **Setup**. The Setup view appears as shown below:

Figure 9



The left side of this section allows you to add new tabs to the Workbook. The right side allows you to configure the properties that appear on each tab.

45. **Add** a new Workbook tab. Choose the object type to be **Stream-Material Stream** on the window that appears.
46. Change the tab Name to **Other Props**.
47. Change the tab to show the following variables: **Molecular Weight, Mass Heat Capacity and vapour phase Cp/Cv**.



Phase specific properties all start with 'Phase…' in the alphabetical variable list.

By using the **Order/Hide/Reveal Objects** option on the **Workbook** menu it is possible to customize which objects appear on each tab.

48. Change the **Other Props** tab so that it displays only the terminal streams (Feed Gas, Export Gas and 10).

Customizing the PFD

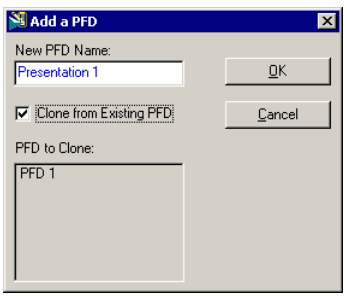
For each flowsheet, HYSYS allows multiple PFD views to be configured. These PFDs are just different views on the same set of objects, so deleting a stream from one PFD will mean it is deleted from all the PFDs. Using multiple PFDs allows various possibilities:

- Creation of a 'Presentation' PFD that has some streams/ operations hidden to produce a PFD ready for output.
- Define a number of views on the same flowsheet (e.g. a zoomed out view and a number of views zoomed in to particular areas of the process).

When using multiple PFDs it is a good idea to have one 'working' PFD that shows all the streams and operations.

49. Create a PFD named **Presentation** using the **Add a PFD** option from the **PFD** menu. Choose to **Clone** the existing PFD.

Figure 10





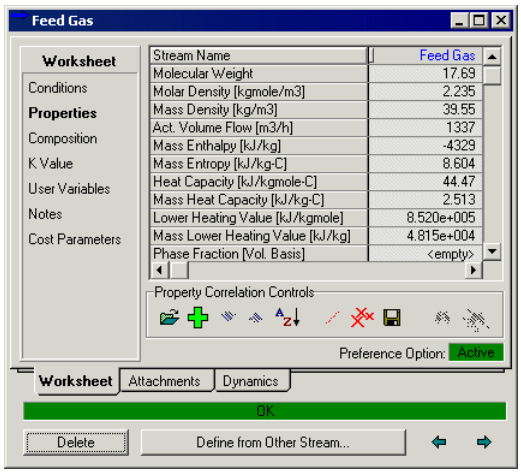
To unhide objects that have been hidden, right-click on the PFD background and choose R**eveal Hidden Objects**.

50. Hide all the Set operations on the Presentation PFD by right-clicking on their PFD icons and choosing **Hide** on the object inspect menu.
51. Add a PFD workbook table for the **Other Props** Workbook tab by object inspecting the PFD background and choosing the **Add Workbook Table** option.

Customizing Stream Properties

The Properties page of the material stream property view can be customized.

Figure 11



The user can:

- Add or remove properties (these are also referred to as Property Correlations)
- Change the order of properties
- Save sets of property correlations and apply them to other streams, or to the whole case

There are two places in HYSYS that these correlations are controlled:

- Property Correlation Controls section on the stream Properties page – Allows the correlations for an individual stream to be customized. Also allows sets of correlations to be saved.
- Correlation Manager on the Tools menu – Allows changes to be made to the property correlations in use for the whole case.

The property correlations displayed for a particular stream are controlled using the buttons at the bottom of the stream window on the Properties page.

Figure 12



These have the following functions:

Button	Flyby Text	Notes
	View Correlation Set List	Allows the user to pick from a list of previously defined correlation sets.
	Append New Correlation	Brings up a window where all correlations are displayed in a tree, and can be selected and added. Correlations are added to the bottom of the list.
	Move Selected Correlation Down	
	Move Selected Correlation Up	
	Sort Ascending	
	Remove Selected Correlation	
	Remove All Correlations	
	Save Correlation Set to File	The Correlation Set can then be loaded with the View Correlation Set List function.

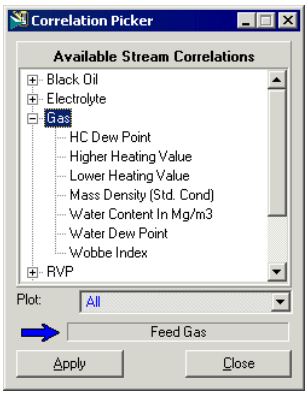
	View Selected Correlation	See settings specific to the selected correlation.
	View All Correlation plots	See all correlation plots for the stream. Currently greyed out as none of the correlations have plots.

1. Open the property view of the Feed Gas stream. Click the **Remove All Correlations** button to clear all the correlations from the stream.
2. Using the **Append New Correlation** button, add the following properties:
   - Gas - HC Dew Point
   - Gas - Higher Heating Value
   - Gas - Lower Heating Value
   - Gas - Wobbe Index
   - Standard - Act. Volume Flow



Don't forget to click **Apply** to add these properties.

Figure 13



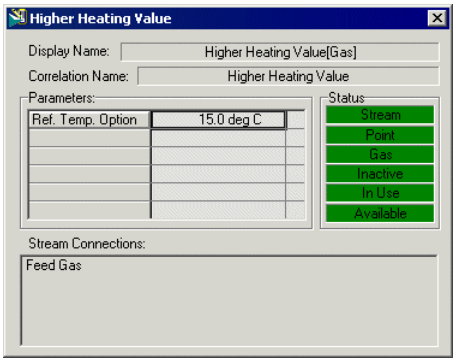


By clicking the blue stream arrow button next to the stream name, a different stream can be selected.

3. On the Feed Gas stream Properties page, select the **Higher Heating Value [Gas]** row in the table.

4. Click the **View Selected Correlation** button.

Figure 14





Property correlation parameters can only be edited using the Correlation Manager.

The meaning of the Status group is explained below.

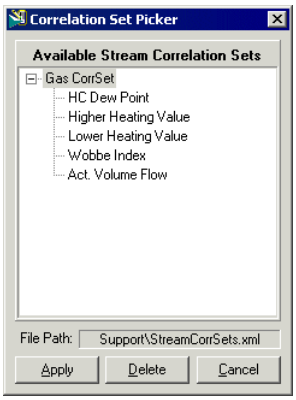
A window appears giving details of the property, note that here the Reference temperature option cannot be changed (i.e., it appears in black).

5. Click the **Save Correlation Set to File** button to save the properties in this stream as a correlation set called **Gas CorrSet**.
6. Open the **Properties** page for the Export Gas stream. Remove all the existing correlations and add the Gas CorrSet correlation set to the stream using the **View Correlation Set List** button.



If the existing correlations are not first removed, then any new ones in the Correlation Set are added to the bottom of the list.

Figure 15



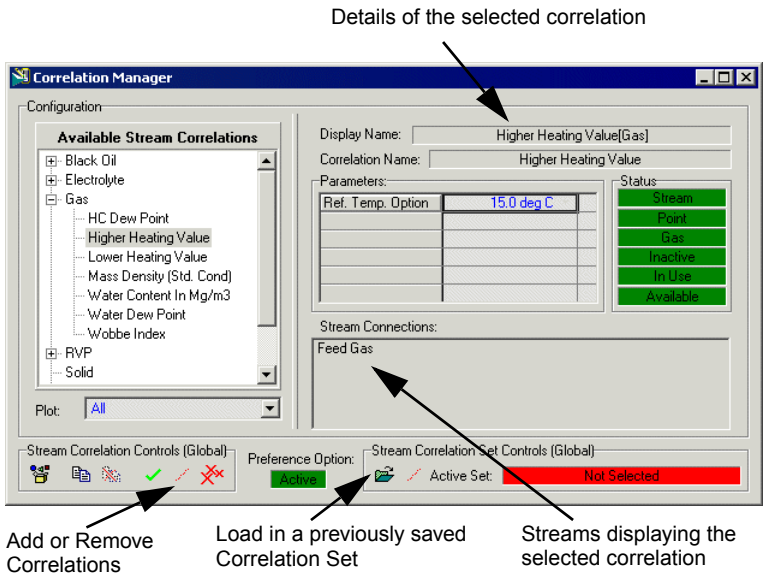
All user defined correlation sets are stored, by default, in the file **StreamCorrSets.xml** in the \Support subdirectory of the HYSYS

installation. The name and location of this file can be configured on the **Files-Locations** page of the Preferences window (**Tools-Preferences** menu option). This file is not created until a correlation set is added.

Customizing Properties for the Whole Case

HYSYS includes a Correlation Manager where global changes for the whole case can be made. This is accessed from the **Tools-Correlation Manager** menu item.

Figure 16



The meaning of the Stream Correlation Controls (Global) buttons are similar to those for the individual stream, except any changes apply globally (i.e., to all the streams in the case). The functions of the buttons are as follows:

Button	Flyby Text	Notes
	Scan System Correlations	Click this icon to manually scan the system registry and build a list of available property correlations.

	Clone Selected Correlation	Only enabled when a property with variable parameters is selected (details are given in the following sections).
	Delete Cloned Correlation From List	Only enabled when a previously cloned property is selected in the Clone part of the tree, and when no streams are showing the property.
	Activate Selected Global Correlation	Adds the selected property correlation to all the streams in the case.
	Remove Selected Global Correlation	Removes the selected property correlation from all the streams in the case.
	Remove All Global Correlations	Removes all the property correlations from all the streams in the case.

Some properties (like the Higher Heating Value property viewed above) have user adjustable parameters.

1. Open the **Correlation Manager** using the **Tools** menu.
2. Select the **Gas-Higher Heating Value** property in the tree. The right side of the window now shows a similar view to that seen previously. However, now the reference temperature can be changed.
3. Change the reference temperature and note that the values calculated in the Feed Gas and Export Gas streams change.

When you change a correlation's parameters, all the streams displaying the correlation will use the new parameters. Hence HYSYS has a clone feature that allows multiple copies of the same correlation to be used. Cloned correlations can have different parameter values to the original correlation.



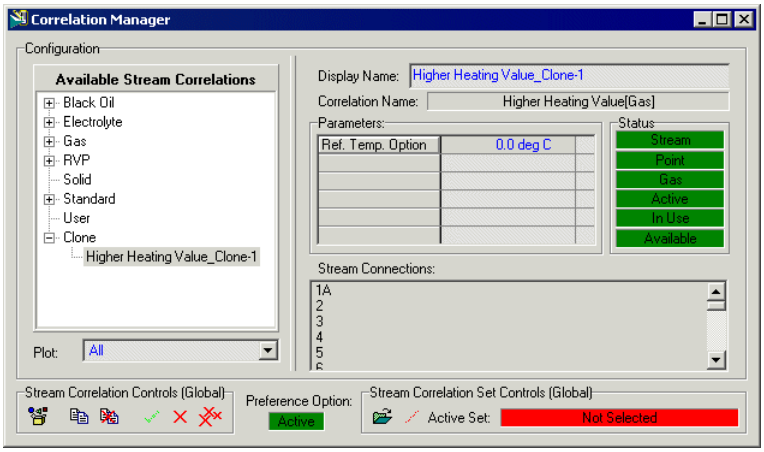
Note that now the cloned Higher Heating Value correlation is now present in all the streams in the case.'

Cloned correlations can be renamed by typing a new value into the Display Name cell.

4. Click the **Clone Selected Correlation** button to clone the Higher Heating Value property.
5. Find the new cloned property in the Clone section of the tree.
6. Add this correlation to all the streams in the case using the **Activate Selected Global Correlation** button.
7. Set the reference temperature for the cloned correlation so that it is different from the original correlation.

Compare the values of the two correlations for the Feed Gas and Export Gas streams.

Figure 17



The following table describes the six bars contained in the Status group:

Status Bar	Description
Stream	Indicates that the correlation can only be applied to material streams.
Point/Plottable	Indicates whether the property correlation is a point or plottable property.
Black Oil/Electrolyte/ Gas/RVP/Solid/ Standard/User/Clone	Indicates which correlation type the property correlation resides within in the Available Correlations list.
Active/Inactive	Indicates whether the property correlation has been activated by the correlation manager. If the status bar is green, any new stream added to the flowsheet with the same fluid type as the correlation will automatically have the property correlation added.
In Use/Not in Use	Indicates whether the property correlation is being used by a stream in the case.
Available/Unavailable	Indicates whether the property correlation exists in the window registry of the system.



View Global Correlation Set List button

The **View Global Correlation Set List** button in the Stream Correlation Set Controls (Global) group allows a previously saved Correlation Set to be displayed for all the streams in the case.

8. Using the Correlation Manager, remove all the existing correlations for all the streams in the case, then load in the previously created **Gas CorrSet** Correlation Set so that it is used by all the streams.

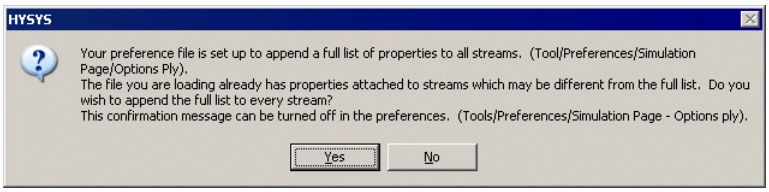
Warning Message: Loading a Case

When you load a previously saved case, you may see the following message:



The wording of this message and the Preferences options are slightly different for HYSYS versions before 3.2, although the effect of choosing each option is the same.

Figure 18

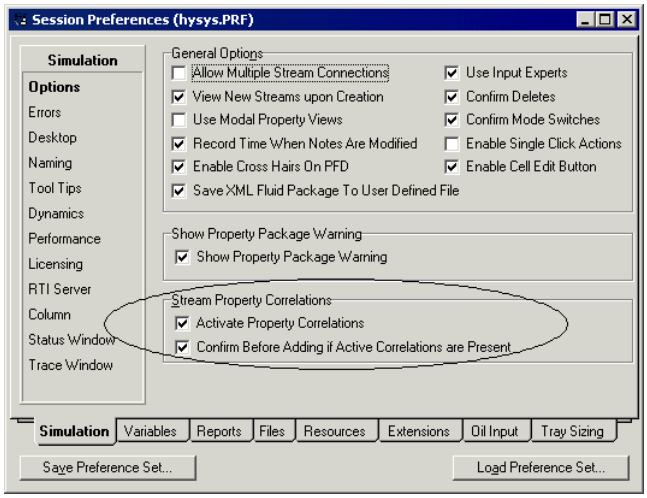


- **Yes** will append the standard set of properties to any streams that may have had properties removed. Any custom properties added will remain.
- **No** will leave all stream property views as they were when the case was saved.

In most cases the standard property set will be being used anyway, so it does not matter which option is chosen.

The HYSYS preferences include several options relevant to this message.

Figure 19

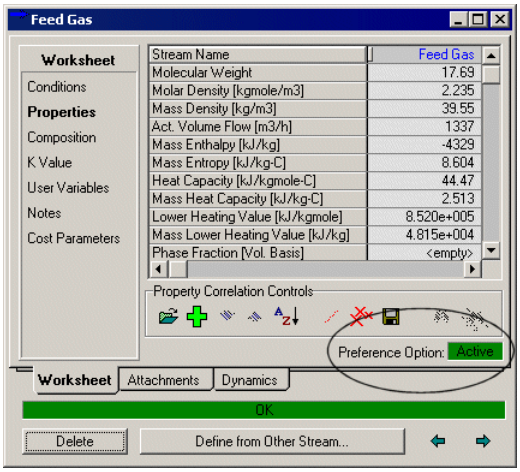


- Checking **Activate Property Correlations** tells HYSYS to add the standard correlations upon opening a case.
- Checking **Confirm Before Adding if Active Correlations are Present** makes HYSYS show the previous message when every case is loaded.
- Unchecking **Confirm Before Adding if Active Correlations are Present** is equivalent to clicking **Yes** each time the message appears.

The safest choice of options is the default. Unchecking the **Confirm Before Adding if Active Correlations are Present** is probably worthwhile, unless you are dealing with cases where the correlations have been customized.

Each stream has a status indicator on the Properties page that indicates whether the **Activate Property Correlations** option is checked in the Preferences. The Correlation Manager window also has a similar status indicator.

Figure 20



9. Save and close your case.
10. Go to the **Simulation-Options** page of the HYSYS Preferences (select the Tools-Preferences menu option).
11. Ensure that the two checkboxes in the Stream Property Correlations group are activated.
12. Reload the case. Click **Yes** on the message box. Notice that all the streams in the case now have the standard set of correlations in addition to any customizations.
13. Reload the case again. Click **No** on the message box. Notice that now the streams only show the correlations in the Gas CorrSet correlation set.

Extensions

© 2004 AspenTech - All Rights Reserved.

Introduction

One of the most powerful features of HYSYS is that users are able to create and add their own unit operations to the program through extensibility. In this module, the power of this feature will be demonstrated, however the process required to build an extension in HYSYS will not be covered.

If you want to learn more about creating unit operation extensions or other tools, using the extensibility feature of HYSYS; AspenTech offers another course that will meet your needs. For more information, ask the instructor.

With unit operation extensions users can create models for unit operations that are not available in HYSYS. Unit operations can also be used to perform calculations, much like the depressuring utility also examined in this course.

The Virtual Stream extension to be used in this module allows the user to transfer information from one stream to another, creating a “Live Link” between them.

Learning Objectives

By completing this module, you will learn how to:

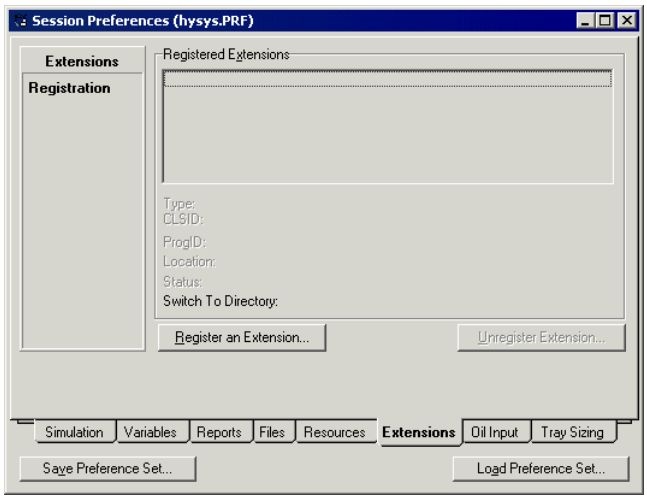
- Register extensions in HYSYS.
- Use a prebuilt extension in a HYSYS simulation.

Registering Extensions

Before extensions can be used in a simulation, they must be registered.

1. Open the **Tools** menu and select **Preferences**.
2. Click the **Extensions** tab.

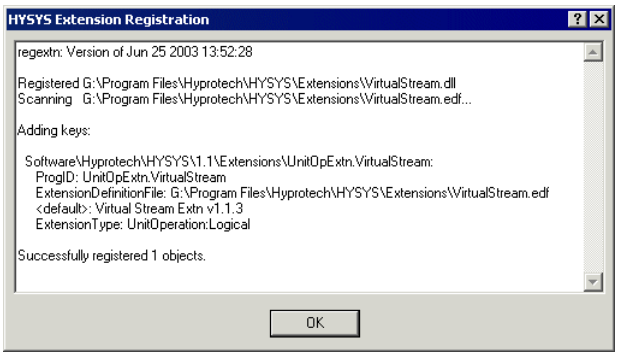
Figure 1



3. Click the **Register an Extension** button. The Select an Extension to be Registered view appears.

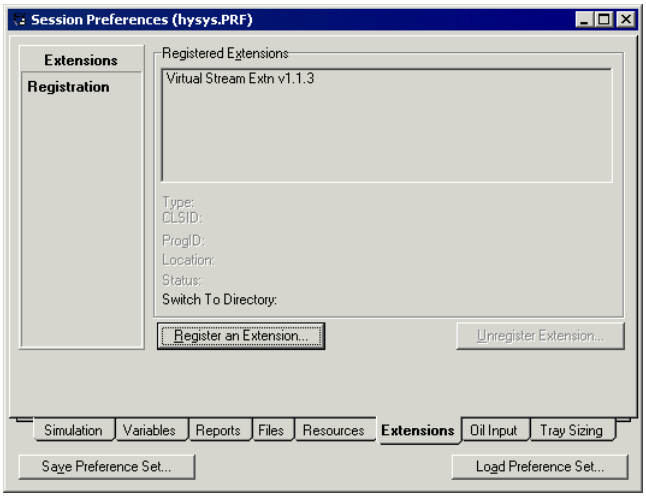
4. Navigate to where your extension file is located and double-click it to register it with the system.

Figure 2



Once an extension is registered, it will appear on the **Extensions** tab, and you will be able to use it in your simulation.

Figure 3



5. Close the Session Preferences view.

There is no need to restart your computer, although HYSYS may need to be restarted.

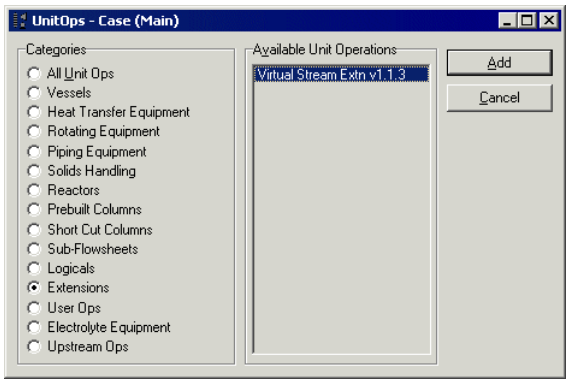
Adding Extensions to Your Simulations

Extensions are added just like any ordinary unit operation in HYSYS. The only difference is that they do not have an icon on the Object Palette.

The simplest way to add an extension to your simulation is to follow the steps below:

1. Press the <**F12**> hot key to bring up a menu of unit operations.
2. Select the **Extensions** radio button from the column on the left. The extensions that are registered with the system will appear in a column on the right.
3. Select the desired extensions and click the **Add** button.

Figure 4



If an extension that has just been registered does not show in the list of available extensions, then restart HYSYS.



Don't worry if you haven't built the Turbo Expander plant case. You can use the "ADV1_GettingStarted_Soln .hsc" file.

Workshop

In this workshop, you will add the Virtual Stream extension to various streams in the Turbo Expander case built in the Getting Started module.

The files for this extension (VirtualStream.dll, VirtualStream.edf and Virtual Stream User Guide.doc) have been supplied to you on the course disk. In addition to this extension, there are several others available on the AspenTech support website (support.aspentech.com).

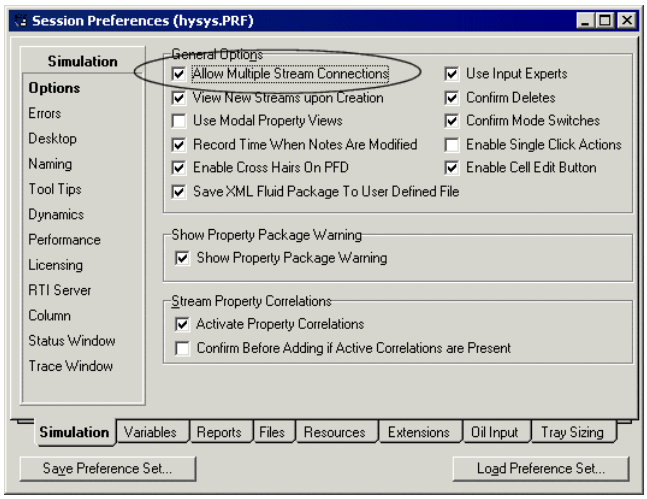
1. Copy these files to a location on the computer's hard disk.

It is not important where on the hard disk you copy these files, but remember the location as you will need to find these files in order to register them with the system.

2. Once the files have been copied to the hard disk, move to the **Extensions** tab of the Preferences view. (Accessed through the **Tools** menu in the main menu bar.)
3. Click the **Register an Extension** button, and use the file explorer to locate the **VirtualStream.dll** file. Opening this file will register it with the system and allow you to use it in the simulation.
4. Return to the PFD of the simulation and add the extension using the procedure described previously.
5. On the extension's **Connections** tab, use the drop-down list to select the Reference stream as **Feed Gas**, and for the Target Stream create a new stream called **Feed Gas VS**.

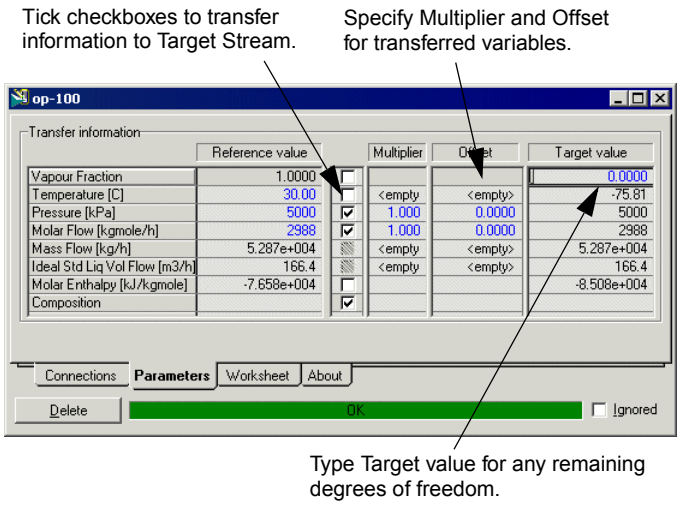
If the Feed Gas stream doesn't appear in the Reference stream drop-down then the **Allow Multiple Stream Connections** option must be set in the preferences (**Tools-Preferences** menu).

Figure 5



6. On the **Parameters** tab, configure the extension to transfer the Pressure, Molar Flow and Composition, with a specified Target Vapour Fraction, as below.

Figure 6



This means that stream Feed Gas VS automatically maintains the same Pressure, Molar flow, and composition as the Feed Gas stream, but will always have a vapour fraction of 0. In this way, the temperature of Feed Gas VS is always the bubble point of stream Feed Gas.

Challenge

Connect to the web and investigate the additional extensions available within the Sample Macros and Extensions section of the Knowledgebase on the AspenTech support web page at **http:\\support.aspentech.com**. In order to access the Knowledgebase you will need to have previously registered and obtained a login ID.

These can be downloaded and registered in exactly the same way as the Virtual Stream extension.

Alternatively you may have some others already stored on your company network or your computer.

Advanced Columns

© 2004 AspenTech - All Rights Reserved.
EA 1000.32.02
03 Advanced Columns

Workshop

Most users are familiar with the prebuilt columns that are available in the main HYSYS Environment. This module will introduce the concept of custom columns. HYSYS allows users to build columns without using the prebuilt configurations. This is useful for simulating columns that do not fit into the usual configurations.

Building custom columns allows users to replace reboilers with heat exchangers; the heat exchangers can then be rated and sized. Likewise, thermosyphon reboilers can be used in place of generic reboilers.

HYSYS also includes a Column Sizing utility that can size and rate column tray sections.

Learning Objectives

After completing this module, you will be able to:

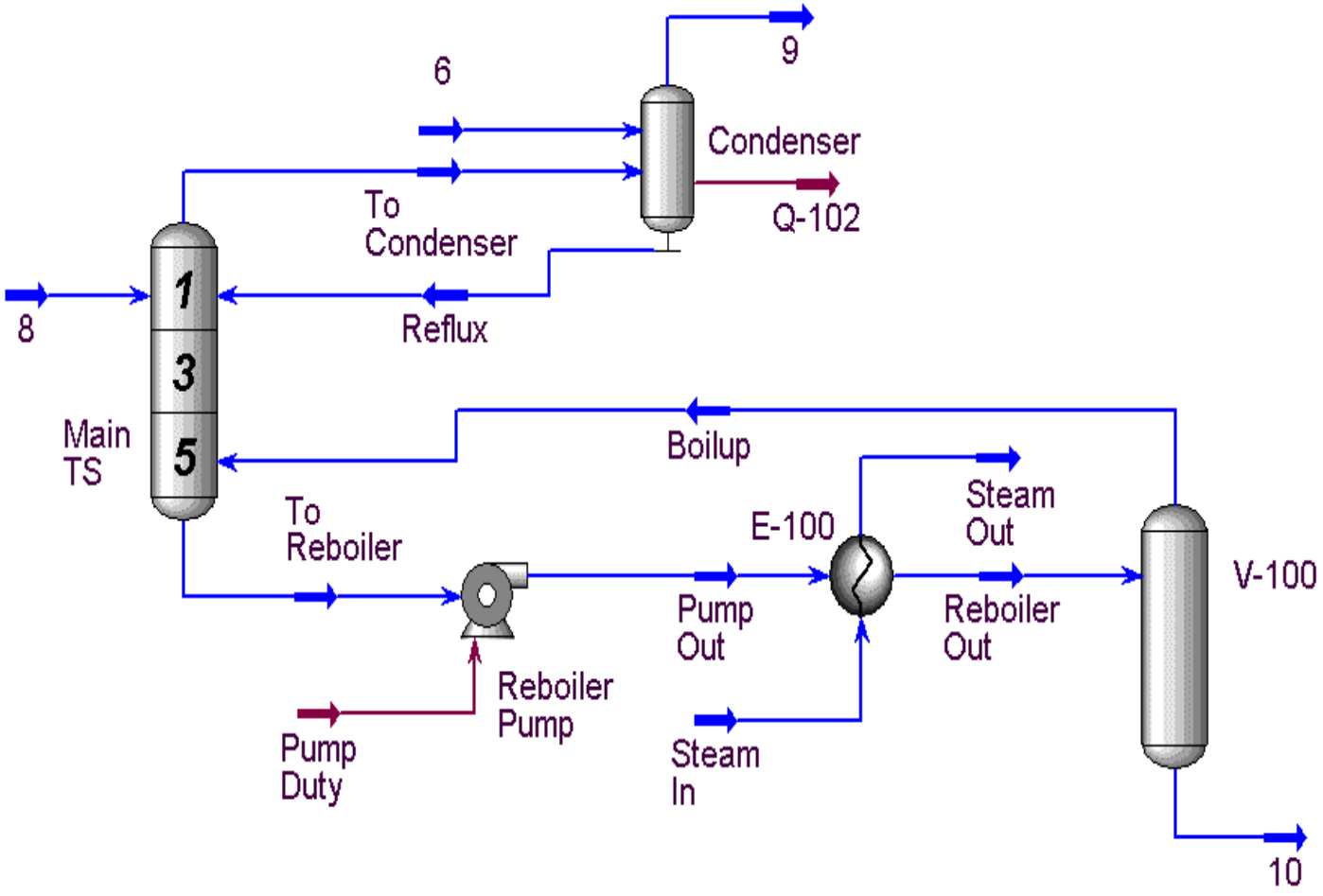
- Build custom columns in HYSYS
- Replace generic reboilers with sizable heat exchangers
- Perform Tray Sizing and Rating calculations

Prerequisites

Before beginning this module, you should be able to:

- Navigate the Main Simulation
- Add unit operations to the PFD
- Add, and converge, a generic prebuilt column

Column Overview



Custom Columns

The most common way of adding a column to a simulation is to use the prebuilt columns that HYSYS offers. There are four prebuilt columns available:

- **Absorber** - the simplest of all towers, no reboiler or condenser.
- **Refluxed Absorber** - an absorber tower with a condenser on the top stage that allows for a refluxing stream in the column.
- **Reboiled Absorber** - an absorber tower with a reboiler at its base, and no condenser at the top.
- **Distillation Column** - an absorber tower with both a condenser and a reboiler.

Customising a Column gives the user greater control over the simulation. For example, replacing the reboiler with a Heat Exchanger allows the user to use steam as a heating medium and size the exchanger based on the required duty.

Custom Columns can be built in two ways, either by modifying a prebuilt column, or by constructing a new column from the beginning. In this module, a prebuilt LPG recovery column will be modified.

The Column Sub-Flowsheet

Whenever a column is added to a HYSYS simulation, a **Column Sub-flowsheet** is created. The sub-flowsheet is essentially another layer in the HYSYS simulation. It is located under the Main layer, and can be seen by selecting **Tools-PFDs** in the main menu bar, or by pressing the hot-key **Ctrl P**. The nature of the layering scheme can be seen in the displayed list as the Column's PFD is indented under the Main PFD. Selecting the desired PFD and clicking the **View** button will open that particular layer for viewing.



In HYSYS Version 3.1+ it is not necessary to enter the Column Environment to edit it. Changes can be made by opening the PFD and editing it, although it is still recommended to enter the Column Environment.

Alternatively the PFD of a particular column can be seen by right clicking on the column PFD icon and choosing **Open PFD**.

In order to edit the column, i.e. to add and modify operations, it is best to enter the Column Environment. The Column Environment is accessed through the Column Property View. To bring up the Column

Property View, double click the column icon on the PFD. With the Column Property View as the active view, click the Column Environment button. This will make HYSYS enter the Column Environment. The operations within the column can be deleted, modified, replaced, and controlled just like ordinary unit operations in the Main Environment.



Enter Parent Environment icon

When in the Column Environment, returning to the Main Environment can be accomplished by clicking the Enter Parent Simulation Environment icon located on the Main Menu Bar.

Adding operations to the column is very similar to adding operations in the Main Environment. The choice of operations is reduced, but the method of installation is identical.



Don't worry if you haven't built the Turbo Expander plant case. The file "ADV1_GettingStarted_Soln .hsc" contains this case.

Building the Simulation

This module will continue with the column built in the Getting Started module. This column is an LPG Recovery unit for a Turbo Expander plant.

This column was constructed as a generic distillation column. The condenser is a "Total Reflux" type; this means that there is no liquid product from the condenser, rather all of the condensed liquid is sent back into the column to provide a refluxing liquid stream.

The reboiler on this column is also of the generic type. An energy stream is supplied and the liquid product from the last stage is boiled up. The vapours return to the column and the liquid leaves the column as a liquid product.

Replacing the Reboiler

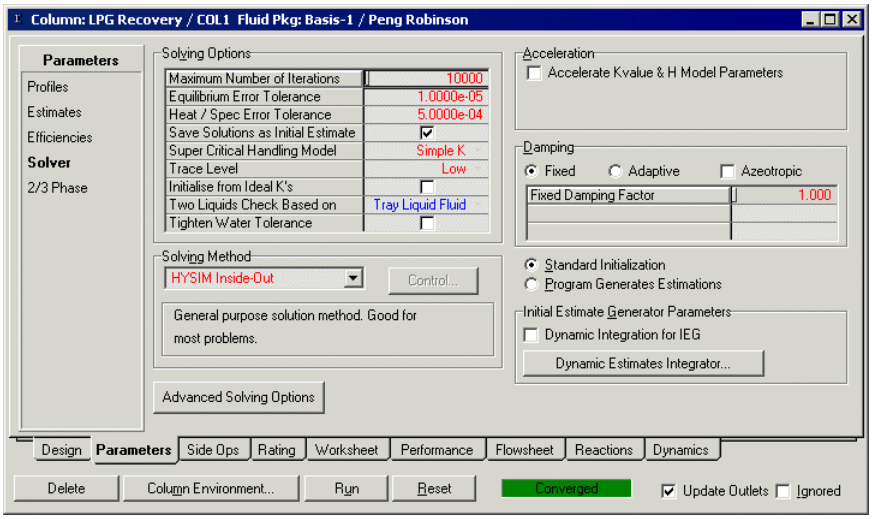


Modified HTSIM Inside-Out is a general-purpose solver that allows Heat Exchangers and other operations in the Column Environment.

The generic reboiler will be replaced with a shell and tube heat exchanger. This will allow the user to supply steam to the column as the energy stream, and size, or rate, the heat exchanger.

1. Before modifying the column, it is necessary to change the Column Solving Method to **Modified HYSIM Inside-Out**. The Solving method is accessed through the **Solver** page (on the **Parameters** tab) of the Column Property View.

Figure 1



2. In the Basis Environment, add **Water** to the list of components.
3. If necessary click the **Run** button to recalculate the column.
4. Enter the Column Environment and delete the existing reboiler and energy stream from the PFD.

Because the generic reboiler that had no pressure drop is being replaced with a heat exchanger that will have a pressure drop, a pump must be added to the simulation to “push” the fluid through the exchanger.

5. Add a **Pump** to the Column Environment with the following parameters:

In This Cell...	Enter...
Name	Reboiler Pump
Inlet	To Reboiler
Outlet	Pump Out
Energy	Pump Duty
Delta P (on Design... Parameters page)	75 kPa (10.9 psia)



Heat Exchanger icon

6. After the newly created pump, add a **Heat Exchanger** to the Column's PFD, with the following parameters:

In This Cell...	Enter
Name	E-100
Heat Exchanger Model	Calculated by Column (default)
Tube Side Inlet	Steam In
Tube Side Outlet	Steam Out
Shell Side Inlet	Pump Out
Shell Side Outlet	Reboiler Out
Tube Side Pressure Drop	50 kPa (7.25 psia)
Shell Side Pressure Drop	75 kPa (10.9 psia)

7. Define the **Steam In** stream as saturated pure steam at **200°C** (392°F). The mass flow rate is 2500 kg/h (5511 lb/hr).

Normally when the reboiler is deleted from the column Sub-flowsheet the pressure that was specified at the bottom will be lost, and hence must be added back on the **Parameters** page in **Profiles** tab.

However in this case the pressure at the bottom of the column is set by the pressure in the bottoms liquid product stream (10), which is linked to the Turbo Expander outlet pressure.



Be sure to pick the Separator operation from the object palette, rather than one of the Condensers which have similar icons.

8. Add a **Separator** after the Heat Exchanger, with the vapour product returning to the bottom stage of the column, and the liquid product leaving the Column Environment as stream 10. (See the Column Overview at the beginning of the module for the column PFD).

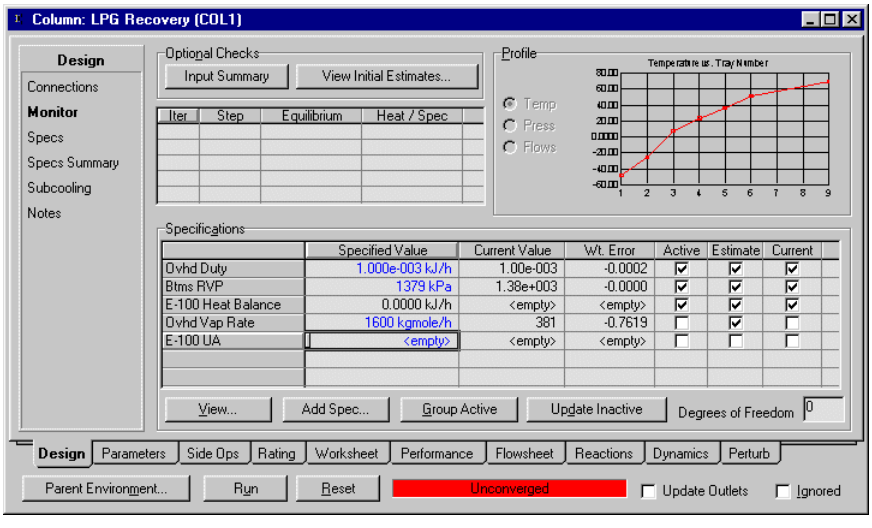
9. Because the Reboiler was deleted, HYSYS removed the Reboiler liquid RVP spec. Add this spec back:

RVP Spec...	
Type	Column Vapour Pressure Spec
Name	Btms RVP
Stage	V-100
Type	Reid Vapour Pressure
Phase	Liquid
Spec Value	200 psia

Since the Heat Exchanger was installed in the Column Environment it's specifications appear on the **Monitor** page along with the specifications of the column. Heat Exchanger specifications can be activated and deactivated just like Column specifications; they can also be added in exactly the same manner as Column specifications.

10. Return to the Main Flowsheet and ensure that the three active specifications for the column are: Ohvd Duty, Btms RVP, and E-100 Heat Balance.

Figure 2



11. Run the column; after it has converged, answer these questions:



What is the UA of the Heat Exchanger?

What is the LMTD for this exchanger?

What is the vapour fraction of the outlet steam?

Save your case!

Connecting Streams Between Flowsheets

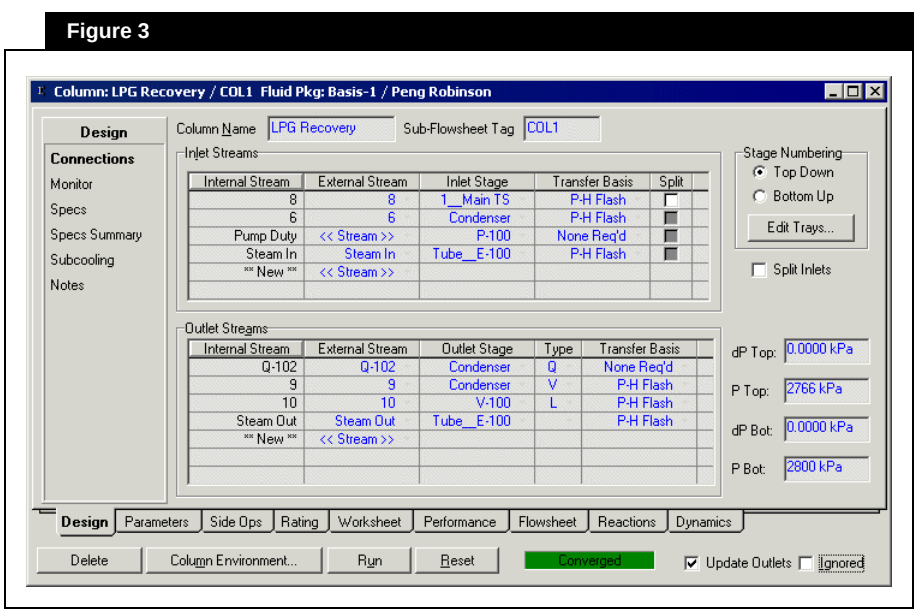
1. Look at the Main PFD. Notice the streams **Steam In** and **Steam Out** are not visible here. This is because they were created in the Column Sub-flowsheet, and have not been connected to the Main Flowsheet.
2. Streams are connected between the Main and Sub-flowsheet on the **Connections** page on the **Design** tab. Notice that there are two types of streams listed here, **Internal** and **External**. Internal streams are those on the Column Sub-flowsheet. External streams are the corresponding streams on the main Flowsheet. Currently **Steam In** and **Steam Out** do not have any linked external streams.

3. Internal streams are connected to the Main Flowsheet by typing a name in the **External Stream** column. Add the appropriate names to the **External Streams** column, and check to make sure that these streams appear in the Main PFD.



As this is now a Custom Column, the Connections page of the Design tab no longer shows a Distillation Column.

Figure 3



The **Steam In** flow rate was defined in the Column Environment, therefore, it is not a specifiable parameter in the Main Environment. Some users prefer to have all the feed streams visible and editable in the Main Environment.

4. Delete the flow rate of **Steam In** in the Column Environment, and enter the same value as a flow rate for **Steam In** in the Main Environment. The column will automatically resolve.
5. Delete the other **Steam In** specifications on the column sub-flowsheet (Vapour Fraction, Temperature and Composition) and transfer them to the main flowsheet.

Save your case!

Exercise - Simulating the Reboiler on the Main Flowsheet Using Internal Streams

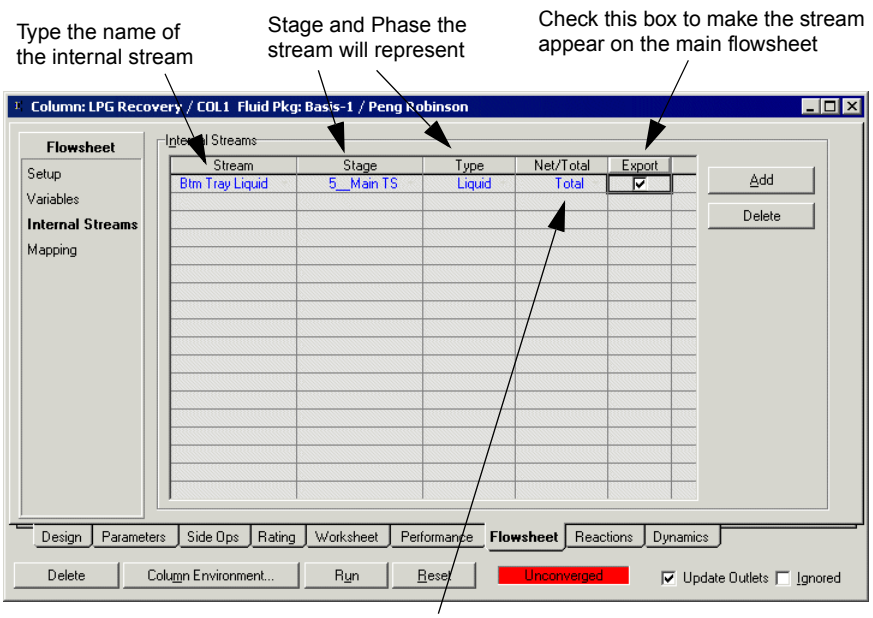
Another method of simulating the reboiler with a Heat Exchanger is to use the Internal Stream feature of the column to make a copy of the reboiler on the main flowsheet. Here this will be added to the case where a rigorous heat exchanger has already been added to the column sub-flowsheet. However this technique works just as well with columns using the standard Reboiler operation.

This way of modelling the reboiler is just as accurate as adding the heat exchanger to the column sub-flowsheet although it does not look quite as good.

An 'Internal Stream' is a special HYSYS stream that represents a fluid inside the column, for example the vapour leaving the top tray, or the liquid leaving the bottom tray. Internal Streams show with a cyan colour on the column sub-flowsheet.

1. Go to the **Flowsheet-Internal Streams** page of the column, press the **Add** button and configure the table as below:

Figure 4



2. Re-run the column to calculate the internal stream.
3. Locate the stream on the main flowsheet, check that its properties are the same as those for the liquid leaving the column.
4. Add a pump, heat exchanger and steam streams as earlier in the module.



Instead of manually adding the streams and operations you can copy & paste them from the column sub-flowsheet by using the right-click copy/paste options.

Pump Name	Reboiler Pump
Inlet	Btm Tray Liquid
Outlet	Pump Out
Energy	Pump Duty
Delta P	75 kPa
Exchanger Name	Copy of Reboiler
Tube Side Inlet	Steam In 2
Tube Side Outlet	Steam Out 2
Shell Side Inlet	Pump Out

Pump Name	Reboiler Pump
Shell Side Outlet	Reboiler Out
Heat Exchanger Model	Exchanger Design (Weighted)
Tube Side Pressure Drop	50 kPa
Shell Side Pressure Drop	75 kPa
Steam In 2 conditions	Saturated pure steam at 200°C. Mass flow 2500 kg/h.

5. Use a Set operation to specify the Reboiler Out stream temperature to be the same as the column bottom product stream.

Save your case!

Column Sizing

HYSYS contains a Tray Sizing utility that greatly simplifies the mechanical design of a distillation column. A user can size full towers, or sections of towers, by specifying information related to the trays, tower internals, the downcomers, and the weirs.

The most common use of the Tray Sizing utility is to identify a tray section, and then make HYSYS size the tower into sections based on your input, then if desired, perform a rating analysis on the column.

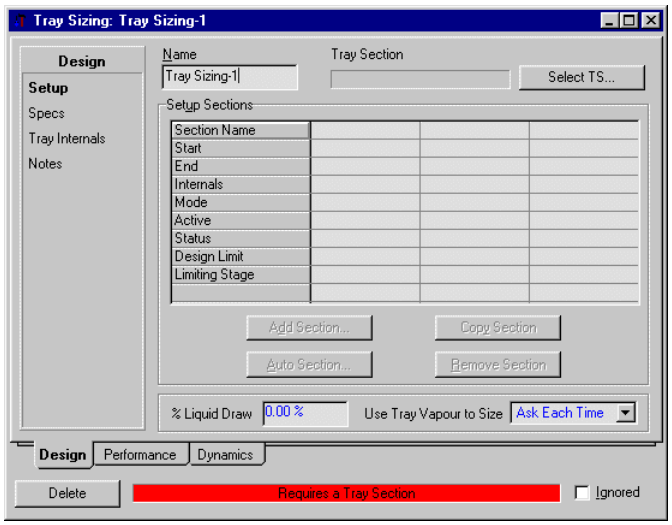
While HYSYS is able to size and rate tray sections, the values that it provides are only rough estimates and should be treated as such.

Column Sizing in Design Mode

1. Select **Tools/Utilities** from the Main Menu bar, or press the hot key **CTRL U**.

2. Select **Tray Sizing** from the list of available utilities and click the **Add Utility** button.

Figure 5



3. Click the **Select TS...** button, and select LPG Recovery as the Flowsheet and Main TS as the Object.

HYSYS allows users to select Tray Sections instead of entire columns so that users may size Side Strippers independently from the Main Column.

4. Click the **Add Section...** button. Accept all of the default values that are presented.

HYSYS will calculate the dimensions of the column using preset values for the column internals and for the various parameters.

On the Performance tab a summary of the calculations are presented. A brief explanation of the terms follows:

- **Number of Flow Paths**. The number of times liquid crosses the tray, most trays are single-pass or have NFP of 1.
- **Maximum Downcomer Backup**. Represents the maximum amount of liquid hold-up in the downcomer that can be tolerated by the column before flooding occurs.
- **Maximum Weir Loading**. Measures the amount of liquid flowing over the weir.
- **Pressure Drops**. Estimates the total pressure drop over the section and the maximum pressure drop per tray.



What is the maximum pressure drop per tray in the Distillation Column?

Over which tray does this pressure drop occur?

What is the diameter of the trays inside the column?

What is the total section height?

Column Sizing in Rating Mode

In rating mode, HYSYS allows you to perform rating calculations based on a specified tower diameter and fixed tray configuration.

Exercise

Head office is desperate to build a distillation column, but the fabrication mill is working overtime and there are very lengthy delays for special orders. The mill has the following trays in stock:

Diameter, m (ft.)	NFP (Passes)
0.75 (2.5)	1
1.0 (3.3)	1
1.25 (4.1)	1

Obtain the flow parameters and pressure drops for a column that uses the trays as given above. The following requirements must be met in order for the column to be constructed.

- Maximum% Flood = 85
- Maximum weir loading = 80 m^3/h-m (860 ft^3/h-ft)
- Maximum downcomer backup = 50%

For each case, follow these steps:

1. On the **Design** tab, click on the **Specs** page and set the Mode to Rating.
2. Set the diameter and number of flow paths, and move to the **Performance** tab.
3. Leave all the other specs at the default values.

4. Complete this table with the information provided by HYSYS.

Case Number	1	2	3
Diameter, m (ft.)	0.75 (2.5)	1.0 (3.3)	1.25 (4.1)
NFP	1	1	1
Weir Load			
Flood			
DC Back Up			
Total Delta P			



Compare the table above with the specifications on the previous page; which set of trays will best meet the restrictions? Remember that smaller trays will be less expensive.

Save your case!

Challenge

The Export Pressures button on the Tray Sizing Utility Performance tab allows the calculated pressure drops to be exported to the column pressure profile.

Use this feature to supply the LPG Recovery Column with the rigorously calculated pressure profile.

Hint: Currently the condenser, reboiler, top and bottom tray pressures appear as calculated (black) numbers. You will need to make them specified (blue) before the Export Pressures function will work.

Templates and Sub-Flowsheets

© 2004 AspenTech - All Rights Reserved.

Sub-Flowsheets

HYSYS has a multi-flowsheet architecture. This allows a large process to be split up into smaller sections or Sub-Flowsheets. Each Sub-Flowsheet, has its own streams and operations, PFD and workbook, and can be independently linked to a Fluid Package.

Templates

A Template is a special sort of HYSYS case, which has been set up ready for easy insertion as a Sub Flowsheet into other HYSYS cases. Template files have the file extension *.tpl.

Workshop

A typical use for templates is to construct a library of ready-made plant sections ready to be installed into other models. In this module a refrigeration loop template will be constructed and then installed into the Turbo Expander plant model.

Learning Objectives

By completing this module, you will learn how to:

- Build a template and/or a sub-flowsheet
- Install a template into a case
- Move objects between Flowsheets
- Efficiently use templates and sub-flowsheets in your simulations

Creating a Template

There are three ways to create a template:

- Convert a whole flowsheet into a template
- Create a new template
- Convert part of a flowsheet into a template.

Convert a Whole Flowsheet into a Template

1. Access the cases Main flowsheet's Property view (**Simulation - Main Properties** or press **Ctrl M**) and select the **Convert to Template** button. You will be asked to confirm changing the case to a template, and whether you want to save the simulation (as an ordinary *.hsc case file) first before converting it to a template.
2. Set the **Template Tag**, **Transfer Basis** (if a material stream) and other optional template information if required. These settings are covered later in this module.
3. When you save the simulation, it will be saved as a template.

Create a New Template

1. From the **File** menu, select **New**, then **Template**.
2. Follow the standard procedure for building your simulation.
3. Access the cases Main flowsheet's Property view (**Simulation - Main Properties** or **CTRL M**) and set the **Template Tag**, **Transfer Basis** (if a material stream) and other optional template information if required.
4. When you save the simulation, it will be saved as a template.

Convert Part of a Flowsheet into a Template

There are two possible approaches. The easiest is as follows:

1. On the PFD, select all unit ops and streams you wish to "capture".
2. Right-click on any of the selected objects and select **Cut/Paste Objects** and then **Copy Objects to File (Export)**. Save the group of objects to a *.hfl file.
3. From the **File** menu, select **Open**, then **Cut/Copy/Paste**, then navigate to the *.hfl file you just saved.
4. Convert the new case that is created to a **Template** as in the “Convert a Whole Flowsheet into a Template” section above.

Alternatively a group of objects can be copied and pasted into a new Template, using a combination of the two methods above. The disadvantage of this approach is that some fluid package assignments may need to be manually adjusted.

Example - Refrigeration Loop



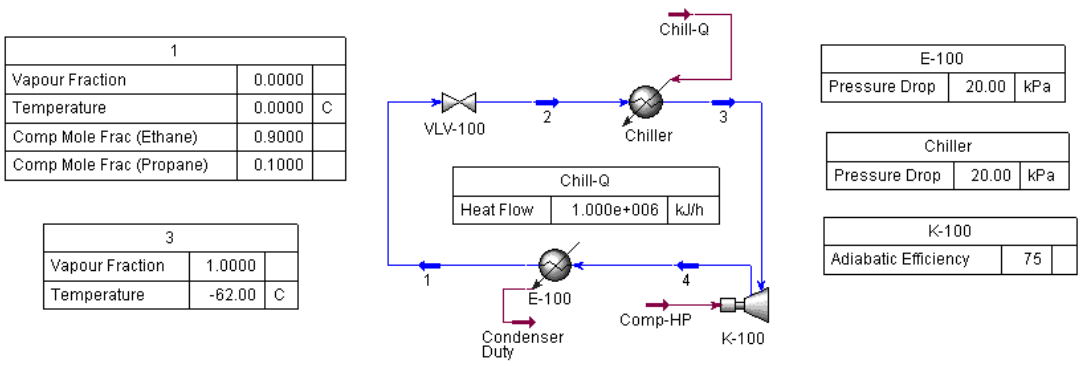
Don’t worry if you haven’t built the Turbo Expander plant case. The file “ADV1_GettingStarted_Soln .hsc” contains this case.

For this example, a refrigeration loop Process Template will be constructed. This template will then be installed as a sub-flowsheet into the Turbo Expander plant simulation built so far in this course.

The required power loads will be will then be determined and displayed on the main flowsheet.

1. Build the following simulation as a new case. Use **PR** as the property package.

Figure 1



2. Initially set Chill-Q to 1E6 kJ/hr, make sure the case solves and looks acceptable.
3. Ensure that the refrigerant flow rate is positive. If it is not, consider why this might be. Check your PFD carefully and make the appropriate changes.
4. Save the case as **Mixed Refrig.hsc**.

Now the simulation needs to be converted to a template, by following the instructions above. However before you do this, you must make sure the template is "ready to run".

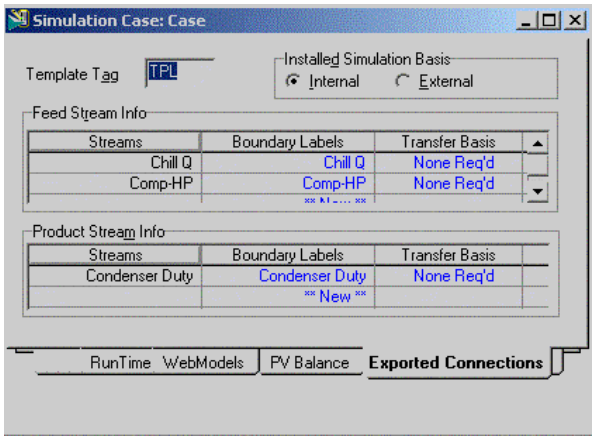
If it is converted and saved as it is now, then when it is installed into a case, HYSYS will generate a consistency error. This is because there are no unknown variables in the Refrig unit, so when a chiller duty is set from the main flowsheet the model will be overspecified.To overcome this the specified Chill-Q value needs to be removed, thus leaving the loop unsolved. Now when the chiller duty is set by the main flowsheet the loop will solve correctly.

5. Remove the Chiller duty, then convert the case to a template following the instructions given on page 4, use the name **Mixed refrig.tpl**.

Template Properties

The properties for the template can be set on the Main Properties window (Simulation - Main Properties or CTRL M). The first few tabs of this window are the same as for any simulation case. However, the two final tabs are exclusive to Templates. These are Exported Connections and Exported Variables. These two tabs only appear when the case has been converted into a template.

Figure 2



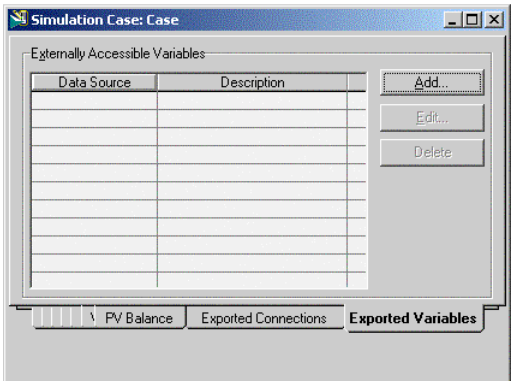
The Exported Connections tab lists all the feed and product boundary streams in the flowsheet, and allows a Transfer Basis to be set, this will be used when the stream is linked between flowsheets. Connections and Transfer Basis are covered in more detail shortly.

The **Template Tag** is used to identify the flowsheet within the case.

The **Installed Simulation Basis** option controls what Fluid Package the flowsheet will use when it is imported into another case.

- Internal - Use the Fluid Package it has now
- External - Use the Fluid Package of the parent flowsheet

Figure 3



Key variables within the flowsheet can be configured here and then easily monitored on the sub-flowsheet's property view when it is imported into the parent flowsheet.

It is not necessary to make any changes on these two tabs, all settings can be configured later when the template is imported as a sub flowsheet into another case.

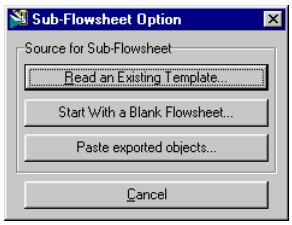
Installing a Template in a Simulation



Flowsheet icon

6. Reactivate the flowsheet from the previous module. To install click on the **Flowsheet** icon on the object palette and select **Read an Existing Template**. Select the template you have just saved.

Figure 4



Once the template is installed as a new sub-flowsheet any subsequent changes made only effect this instance, they do not effect the template from which the sub-flowsheet was derived.

Connections Tab

On the Connections tab, you can enter all the Feed and Product connections between the sub-flowsheet and the main flowsheet. Feed connections are material or energy streams into the sub-flowsheet, products are out of the sub-flowsheet.



Internal streams refer to streams in the sub-flowsheet. External streams are in the main flowsheet.

7. Click on the **External Stream** box along side Chill-Q and select **Q-100** from the pull down menu.
8. For the other two streams there are no existing streams in the main flowsheet so give them new stream names as in Figure 6.
9. Rename the sub-flowsheet to **Mixed Refrig Unit**.

Figure 5

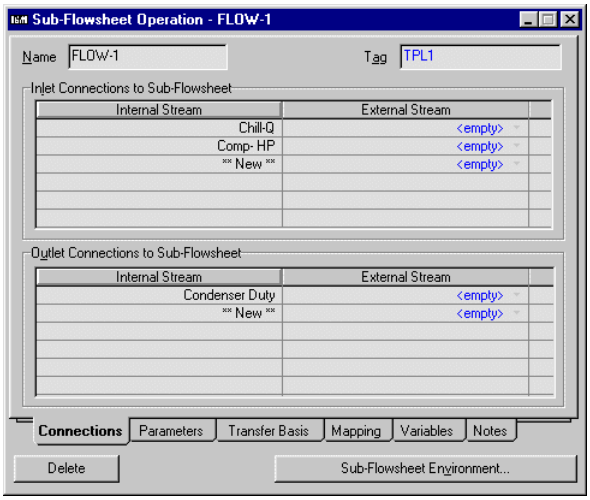
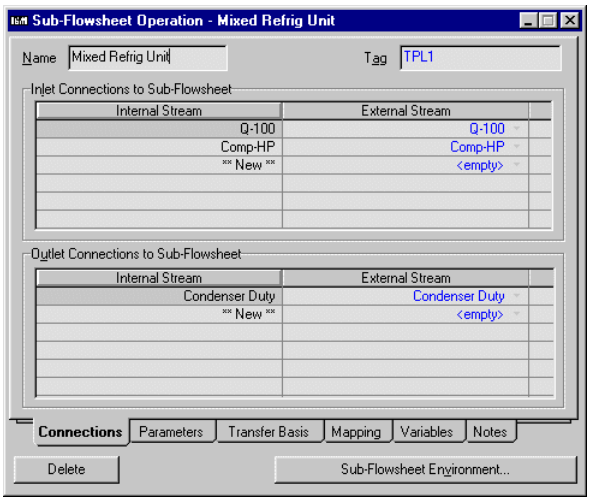


Figure 6



You will notice the Chill-Q stream on the sub-flowsheet is automatically renamed Q-100 to match the main flowsheet stream name.

Template Tag

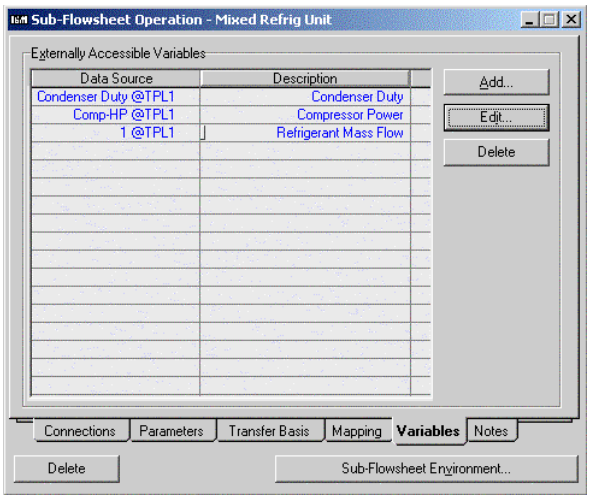
Tags are short names used to identify the sub-flowsheet associated with a stream or operation when viewed outside the sub-flowsheet. The default Tag name for sub-flowsheet operations is TPL1 (e.g.: strm6@ TPL1).

When more than one sub-flowsheet operation is installed the default tag increases, TPL2, TPL3 etc. You may give sub-flowsheets proper names (e.g.: refrig1).

Exporting Sub-Flowsheet Variables

10. Use the **Variables** tab to export the following variables:
    - Condenser Duty, Heat Flow
    - Comp-HP, Power
    - Stream 1, Mass Flow
11. Edit the variable descriptions to show what the variables represent.

Figure 7



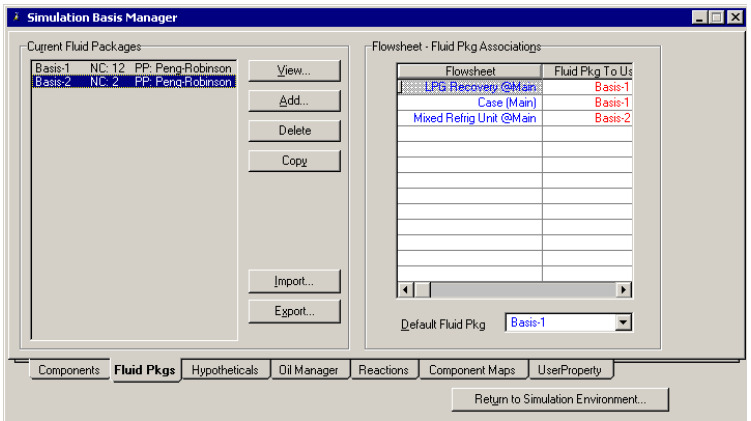
Notice that the values of these variables are now displayed on the Parameters tab.

Reviewing the Simulation Basis

12. Enter the Simulation Basis.

Note that since the default Internal Basis option was used when configuring the template, the refrigeration loop sub-flowsheet is using a different fluid package to the main flowsheet. If required the fluid package used by each sub-flowsheet can be changed in the right table.

Figure 8



Transfer Basis

HYSYS has the option to model a sub-flowsheet using a different fluid package to the main flowsheet. This allows, for example, the modelling of a Gas Sweetening process as a sub-flowsheet using PRSour within a main flowsheet using PR.

As each fluid package could calculate different properties HYSYS must do a flash for any streams passing between flowsheets. The Transfer Basis sets what kind of flash is done.

Flash Type	Description
P-H Flash	The Pressure and Enthalpy of the material stream are passed between flowsheets. A new temperature and vapour fraction will be calculated. Since the Enthalpy basis may be different for each property package this option is only recommended when the same fluid package is in use for both streams.
T-P Flash	The Pressure and Temperature of the Material stream are passed between flowsheets. A new Vapour Fraction will be calculated.
VF-T Flash	The Vapour Fraction and Temperature of the Material stream are passed between flowsheets. A new Pressure will be calculated.
VF-P Flash	The Vapour Fraction and Pressure of the material stream are passed between flowsheets. A new temperature will be calculated.
None Required	No calculation is required for an Energy stream. The heat flow will simply be passed between flowsheets.

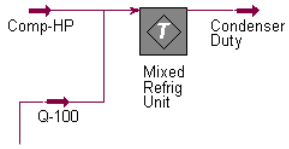
In this case no transfer basis is required since only energy streams pass across the flowsheet boundary.

Finish this section

13. Tidy up the PFD, and add a PFD table for the sub flowsheet to show the exported variables. (PFD tables are added by right clicking on the object and choosing **Show Table**.).

Figure 9

Mixed Refrig Unit		
Exported Variables (Condenser Duty)	5.335e+006	kJ/h
Exported Variables (Compressor Power)	628.5	kW
Exported Variables (Refrigerant Mass Flow)	1.123e+004	kg/h



Save your case!

Challenge

In this case the chiller is simulated using a cooler in the main flowsheet linked with an energy stream to a heater in the sub-flowsheet. It is possible to replace the heater and cooler with a single heat exchanger. One of the features of the HYSYS heat exchanger is the option to have the streams on each side of the exchanger in different flowsheets.

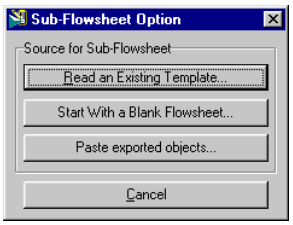
Replace the heater and cooler with a single heat exchanger in the main flowsheet. What advantages does modelling the chiller in this way have?

If you try this challenge, make sure to save your case with a different file name and revert back to the separate heater and cooler case for subsequent modules.

Creating and Manipulating Sub-flowsheets

Sub-flowsheets can be created without first creating a template by choosing one of the other two options when adding a flowsheet.

Figure 10

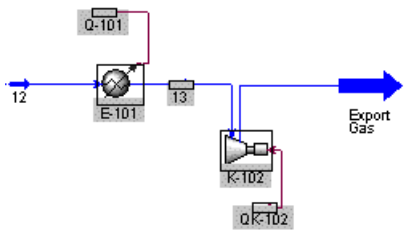


The paste exported objects buttons allows a sub flowsheet to be created that contains a group of objects that have previously been saved as a *.hfl file (by using the PFD right click Cut/Copy Objects … Copy Objects to File function).

A sub-flowsheet can also be created to contain a group of objects that already exist on the main flowsheet.

1. On the Turbo Expander plant PFD, select Cooler **E-101** and compressor **K-102**, their **energy streams**, and stream **13**.

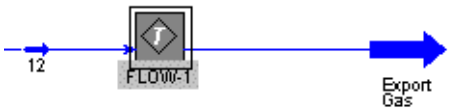
Figure 11



2. Right-click on any of these objects and select **Cut/Paste Objects** and then **Combine Into Sub-Flowsheet**.

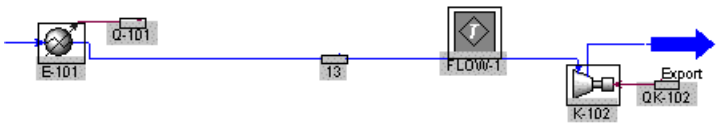
This group of objects are then combined into a sub-flowsheet. HYSYS automatically sets up all the stream connections

Figure 12



3. Right-click on the sub-flowsheet icon and choose **Cut/Paste Objects** then **Move Contents To Owner Flowsheet**.

Figure 13



HYSYS returns everything back to the main flowsheet level.

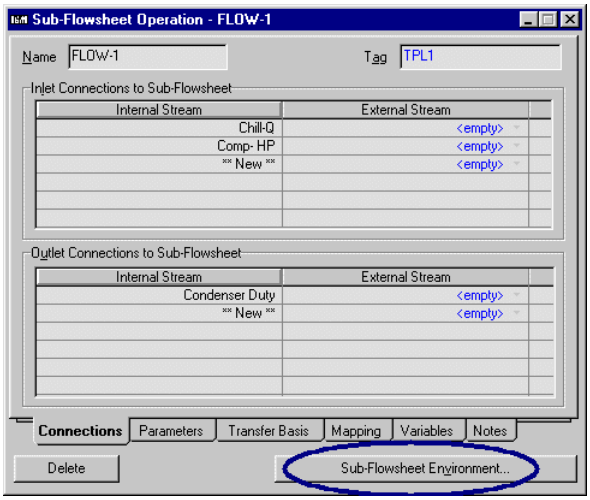
4. Select the sub-flowsheet icon and delete it as it is now empty.

Viewing and Editing the Sub-Flowsheet

Pressing the **Sub-Flowsheet Environment** button on the sub-flowsheet operation window causes HYSYS to enter the sub-flowsheet environment. (This is equivalent to pressing the Column Environment

button on the column to enter the column sub-flowsheet environment.)

Figure 14



The Environment label at the top right corner of the HYSYS window indicates which environment is currently active. To return to the main flowsheet, press the **Enter Parent Environment** button.

Whilst in the sub-flowsheet environment, the HYSYS solver will only solve the streams/operations within the sub-flowsheet. To see the effect of any changes on the whole case it is necessary to return to the top-level flowsheet.

It is also possible to open the sub-flowsheet PFD and make changes whilst remaining in the main flowsheet environment. Hence any changes made in the sub-flowsheet immediately affect the whole case.

To open the PFD for the sub-flowsheet:

1. Right-click on the **PFD** icon of the sub-flowsheet.
2. Choose **Open PFD**. Alternatively use the **Tools-PFDs** menu.

In HYSYS versions prior to 3.1, it was necessary to enter the sub-flowsheet environment to make any topology changes to the sub-flowsheet (e.g., change stream connections, add/delete objects, etc.), however this restriction has now been removed.

Use of Templates and Sub-Flowsheets

Consider the following possibilities:

- A case can contain multiple layers of sub-flowsheets
- Your company could develop a library of templates for everyone to access. These become company standards, and can be more easily maintained and approved
- Using library templates saves time in modelling and improves QA
- Complicated simulations are much easier to handle if split into sub-flowsheets
- Large PFDs are easier to read if you use sub-flowsheets
- Several engineers can work on the same overall simulation by using templates

Note that links are not dynamic. If a template is modified, it needs to be reloaded into the simulation case for the changes to be incorporated.

Spreadsheets and Case Studies

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
05 Spreadsheets and Case Studies

Workshop

The HYSYS Spreadsheet is a powerful tool that allows the user to apply Spreadsheet functionality to flowsheet modelling. The Spreadsheet has complete access to all process variables; this allows the Spreadsheet to be virtually unlimited in its applicability and function. In this module, the Spreadsheet will be used to calculate a simplified profit for the operating plant.

Learning Objectives

After completion of this module, you will be able to:

- Import and export variables to and from the Spreadsheet
- Add complex formulas to the Spreadsheet
- Use the HYSYS Spreadsheet in a wide variety of applications
- Use the casestudy utility to evaluate your flowsheets

The HYSYS Spreadsheet

With complete access to all process variables, the **Spreadsheet** is a very powerful tool in the HYSYS environment. The power of the Spreadsheet can be fully realized by the addition of formulas, functions, logical operators, and basic programming statements.

The Spreadsheet's ability to import and export variables means that seamless transfer of data between the Simulation Environment and the Spreadsheet is a simple matter. Any changes in the Simulation Environment are immediately reflected in the Spreadsheet, and vice-versa.

The Spreadsheet has several common applications. For example, the Spreadsheet can be used to:

- Collect together key inputs and results between flowsheet objects.
- Relate the pressure drop in a Heat Exchanger to the flow.
- Perform mathematical operations using variables from the simulation.

Importing and Exporting Variables



Note that it is not possible to import into, and export from the same cell, instead use two cells one for the import and one for the export, and link them together with a simple '=A1' type formula.

Any variable in the case can be imported into the Spreadsheet. The contents of any Spreadsheet cell can be exported to any specifiable (blue) variable in the case. There are three ways of importing values into the Spreadsheet.

- **Drag and Drop**. Position the cursor over the desired item; then click and hold the right mouse button. Move the cursor over to the Spreadsheet. Once over the Spreadsheet, the cursor's appearance will change to a "bull's eye" type. Release the right mouse button when the "bull's eye" cursor is over the desired cell. The specific information about the imported variable will appear in the Current Cell group.
- **Variable Browsing**. A variable may also be imported into the Spreadsheet by placing the cursor on an empty cell in the Spreadsheet and clicking (and releasing) the right mouse button. Choose **Import Variable** from the list that appears, and select the variable using the Variable Navigator.

- **Connections Page**. On the Connections page, click the Add Import button and select the desired variable using the Variable Navigator. After selecting the variable, choose the desired cell from the drop-down list.



The value in any spreadsheet cell can be exported, except if it is an imported value.

Exporting variables from the Spreadsheet into the Simulation environment is also a simple procedure. The methods for doing this are very similar.

- **Drag and Drop**. Position the cursor over the Spreadsheet cell that is to be exported. Click and hold the right mouse button; the cursor should now change to the "bulls' eye" type. Move the "bull's eye" cursor over to the desired cell. Release the right mouse button, the transfer should be completed.
- **Variable Browsing**. A variable may be exported from the Spreadsheet into the Simulation environment by placing the cursor on the exportable cell in the Spreadsheet and clicking (and releasing) the right mouse button. Choose Export Formula Result from the list that appears, and select the desired location for the variable using the Variable Navigator.
- **Connections Page**. On the Connections page, click the Add Export button and select the desired variable using the Variable Navigator. After selecting the variable, choose the desired cell from the drop down list.



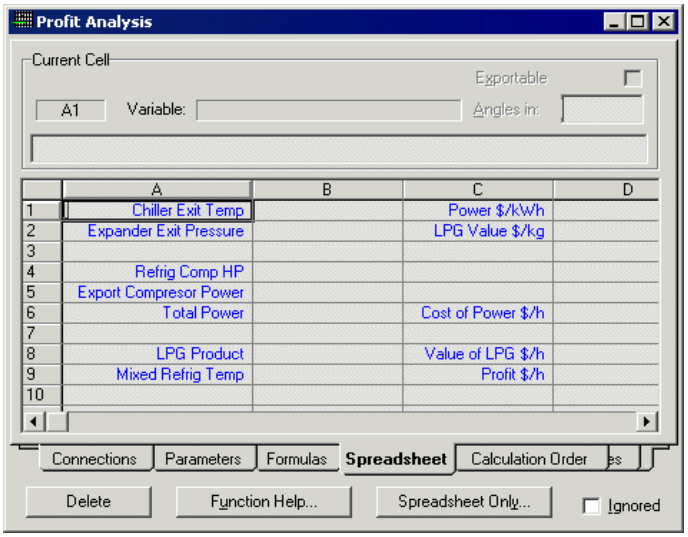
Don't worry if you haven't built the Turbo Expander plant case. Use the file ADV4_Templates_Soln.hsc.

Building the Spreadsheet

In this module a Spreadsheet to calculate a simple profit margin will be added to the Turbo Expander plant.

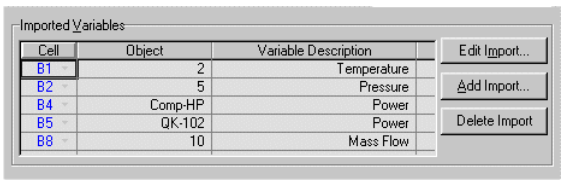
1. Add a Spreadsheet to your model by double-clicking on the **Spreadsheet** icon on the Object Palette. Rename the spreadsheet **Profit Analysis**.
2. On the **Spreadsheet** tab add the following text labels.

Figure 1



3. Set up the required imports.

Figure 2



Try each of the methods described on page 3.

4. Set the **Cost of Power** (cell D1) to be **0.05 $/kWh** and the **LPG** value to be **0.2 $/kg**.
5. Enter the formulae below.

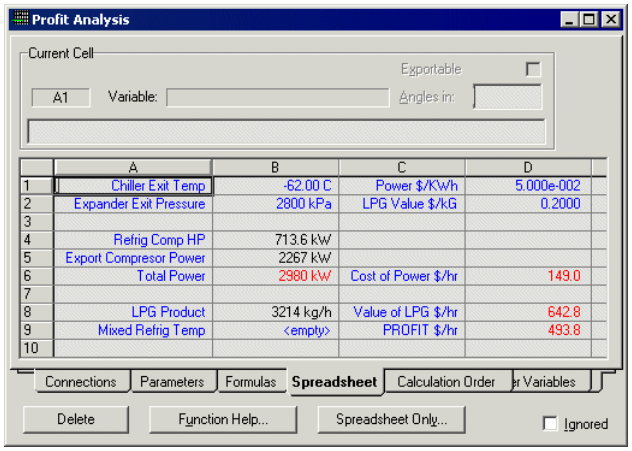
In this Cell...	Enter...
B6	=B4+B5
D6	=D1*B6
D8	=D2*B8
D9	=D8-D6

Notice that HYSYS assigns variable types of Heat flow to cell D6 and Mass flow to cell D8. This is because these are the variable types of the cells involved in the calculation.

6. Using the **Variable Type** drop-down list above the spreadsheet, change the types of these cells to unitless.

The spreadsheet should now look like this:

Figure 3





Remember in HYSYS process variables appear as blue numbers, calculated ones as black, and in spreadsheets any calculated numbers are shown in red.

The only cell remaining to be completed is B9. This is going to be used to control the temperature of the refrigerant in the Mixed Refrig Unit.

7. Add a formula in cell B9 such that it is **5°C** cooler than the Chiller exit temperature



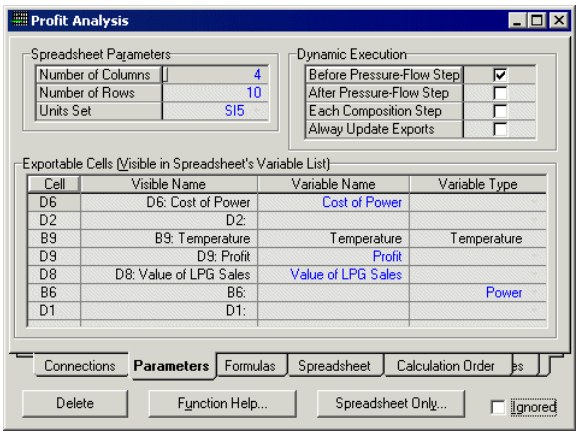
To delete the temperature there is no need to enter the Refrigeration sub-flowsheet environment, simply right click the sub-flowsheet icon and press 'Open PFD'

8. In the **Mixed Refrig Unit** sub-flowsheet, delete the temperature in stream 3.
9. Export the temperature from **B9** to stream **3**.

In order to make it easier to access and use the spreadsheet cells in another unit op (e.g. an Adjust) or in the Databook, cells can be named. This is done either by selecting the cell on the Spreadsheet tab and typing a name in the Variable field above the spreadsheet, or on the Parameters tab.

10. Name cells D6, D8, and D9 as in the following figure.

Figure 4





With a process temp of -62°C and a turbo exit pressure of 28 bar we have a profit of $493.8 /h.

Change the chiller exit temp to -60°C and the expander exit pressure to 20 bar.

What is your new profit?

_____

Save your case!

Use of Spreadsheets

The spreadsheet can be a very useful tool in HYSYS to:

- Pull together important parameters in the simulation into a single unit op. Use it to try "what ifs" by changing process variables and seeing if your target variables change, and in the right direction.
- Perform additional calculations that are not possible in HYSYS directly, such as the profit calculation you have just done.
- Combine data from process streams, energy streams or utility streams and use multiple spreadsheets to calculate your total cooling water requirements or power requirements.
- Careful use of spreadsheets can save you having to open several windows in HYSYS in order to both input information, or examine results.
- You can see that a spreadsheet can be used to set various parameters in the flowsheet as a result of a calculation on another variable. So in the flowsheet here the three sets controlling the pressures of streams 8, 9 and 10 could be replaced with a spreadsheet doing the same thing.

Note that when copying and pasting, spreadsheets links are not always maintained. To ensure all links are maintained convert the relevant part of the case to a template.

Challenge

As you can see the simulation can be "driven" from the spreadsheet, by changing the temperature of stream 2 and the pressure of stream 5 and looking at the new profit.

Do you think you could use the spreadsheet to optimise the cooler exit temperature and turbo expander exit pressure to give the maximum profit available? You could but it would take a long time using trial and error.

The Case Study

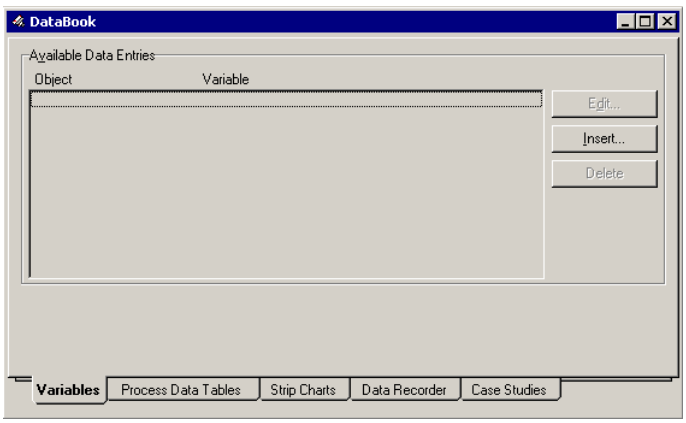
The Case Study tool allows repeated runs with varying input parameters to be automated. In the next section you will set up a case study to vary the Cooler exit temperature and Expander exit pressure between defined limits. For each case various results are recorded for analysis later.

1. Open the DataBook from **Tools-Databook**, or by pressing **CTRL D**.

With the DataBook, HYSYS provides a location from which a systematic approach to data analysis can be taken. The DataBook allows you to monitor key process variables in Steady State and in Dynamics mode. Variables for all DataBook features are selected in a single location. You can then activate variables from the main list for each application.

There is only one DataBook in each HYSYS case, containing variables from all Flowsheets. All of the following features are defined and accessed through this single DataBook:

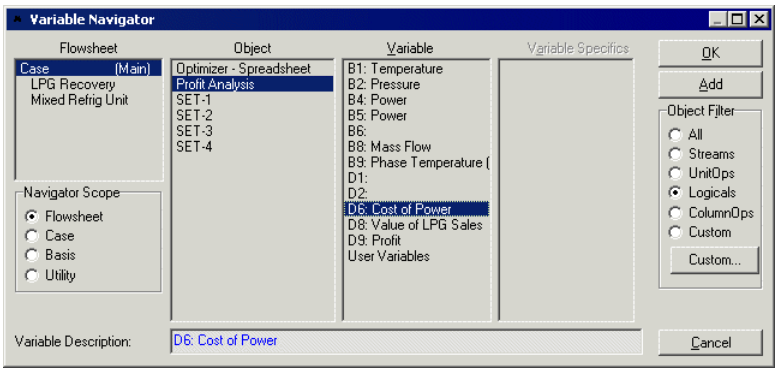
Figure 5



The first step is to configure all the variables of interest, both variables to be varied in the Case Study and the results. HYSYS will do a series of simulation runs and record the results for each case. If you omit a variable from the list you will not be able to look at it once the runs are complete. All data except for that declared is lost. Consider what you might want to observe before you commit to running the Case Study.

2. Click on **Insert** and add the first variable as shown:

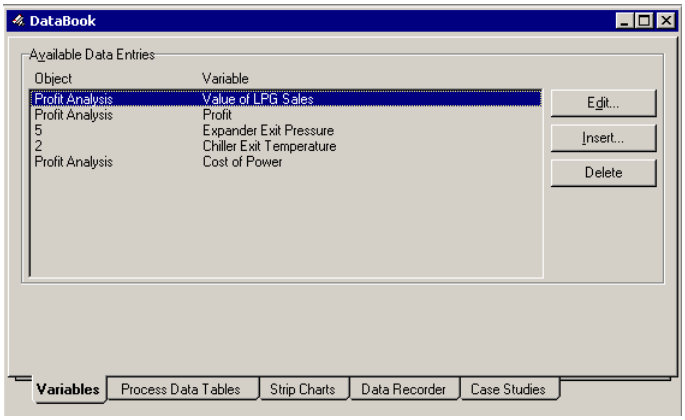
Figure 6



Note that the variable descriptions shown for spreadsheet cells correspond to the Visible Name of each of the cells. Since these are blue parameters they can be edited here. Removing the cell reference will tidy up the Case Study and make the final graphs look more presentable. It is best to add all of the required variables in one go using Add, press OK, and then return and edit their descriptions, rather than adding and editing each in turn.

3. Repeat the above until you have added the following variables. Remember to add all the variables first and then edit their descriptions.

Figure 7



Note, at any time before you actually run the case study you may add or delete variables from this list.



The Independent and Dependent variable checkboxes are only enabled when you add a case study. It is not necessary for all the variables to be ticked for each case study, a minimum of one dependent and one independent variable is required.

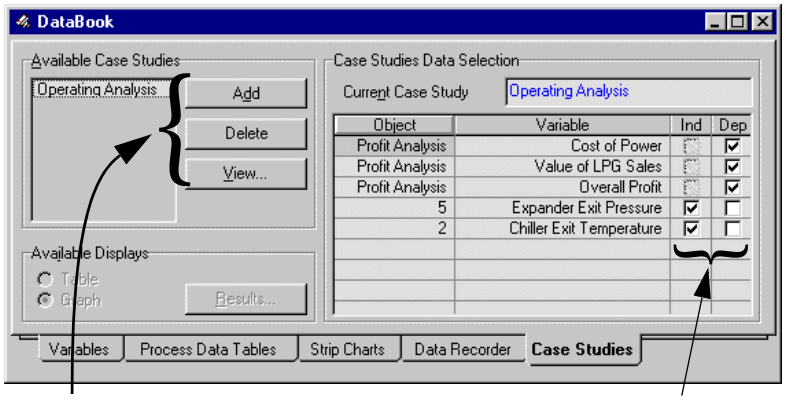
4. Go to the **Case Study** tab and click on **Add** to set up a new case study. Call it **Operating Analysis**.

All the previously configured variables are listed along with two columns **Ind** and **Dep**·

- **Independent** variables are the ones that will be varied by the case study. These must be specified variables (i.e. blue numbers).
- **Dependent** variables are the results to be monitored.

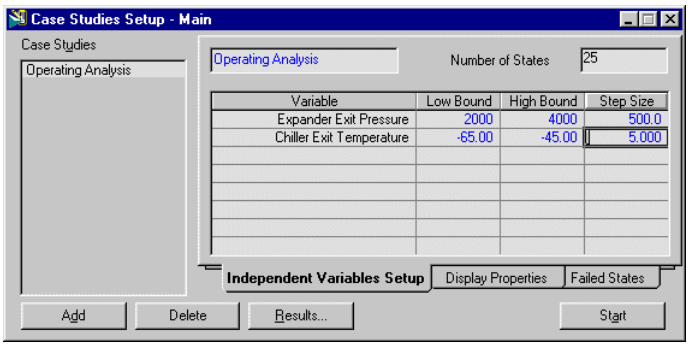
5. Select the independent and dependent variables.

Figure 8



6. Press **View** and configure the bounds as shown below. (Note the Pressure and Temperature units here are kPa and °C.)

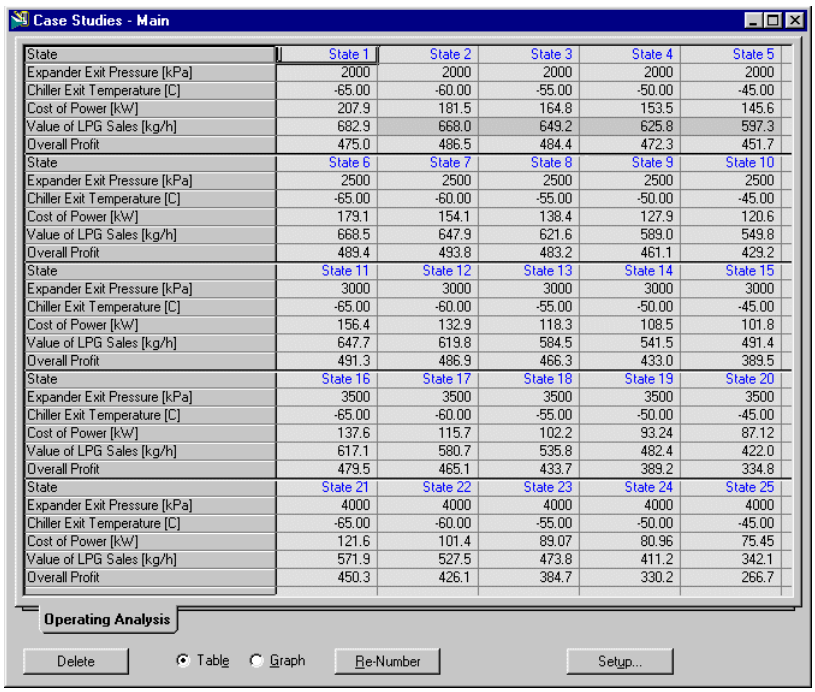
Figure 9



Five different temperatures with five different pressures, means a total of 25 states in all. For each of the states in the case study the whole case is solved, including any utilities. Since in this case the results from the tray sizing utilities are not needed in the case study, it makes sense to ignore the tray sizing utilities to speed up the case study.

7. Press **CTRL U** to open the list of Utilities and then view each tray sizing utility in turn and set it to **Ignore**.
8. Click **Start** to set the study running. The **Failed States** tab will show any combination of independent parameters that fail to solve. While the case study is running, or when it has finished, you can review the results.
9. Either press the **Results** button on the Case Studies Setup window or on the **Case Studies** tab of the Databook.

Figure 10



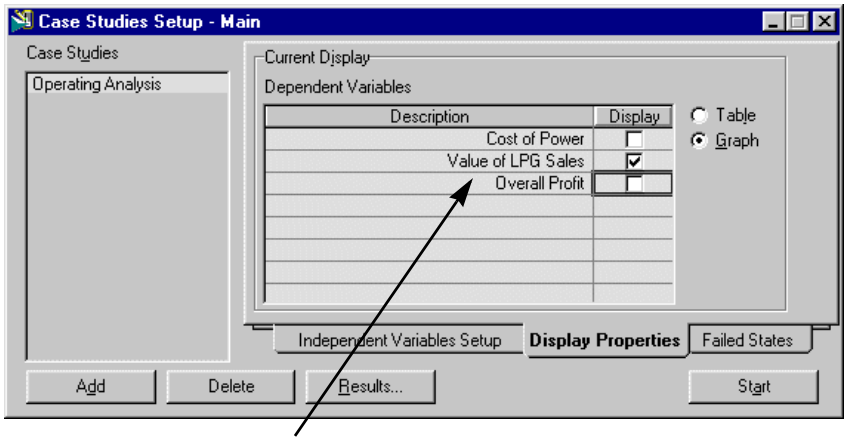
State	State 1	State 2	State 3	State 4	State 5
Expander Exit Pressure [kPa]	2000	2000	2000	2000	2000
Chiller Exit Temperature [C]	-65.00	-60.00	-55.00	-50.00	-45.00
Cost of Power [kW]	207.9	181.5	164.8	153.5	145.6
Value of LPG Sales [kg/h]	682.9	668.0	649.2	625.8	597.3
Overall Profit	475.0	486.5	484.4	472.3	451.7
State	State 6	State 7	State 8	State 9	State 10
Expander Exit Pressure [kPa]	2500	2500	2500	2500	2500
Chiller Exit Temperature [C]	-65.00	-60.00	-55.00	-50.00	-45.00
Cost of Power [kW]	179.1	154.1	138.4	127.9	120.6
Value of LPG Sales [kg/h]	668.5	647.9	621.6	589.0	549.8
Overall Profit	489.4	493.8	483.2	461.1	429.2
State	State 11	State 12	State 13	State 14	State 15
Expander Exit Pressure [kPa]	3000	3000	3000	3000	3000
Chiller Exit Temperature [C]	-65.00	-60.00	-55.00	-50.00	-45.00
Cost of Power [kW]	156.4	132.9	118.3	108.5	101.8
Value of LPG Sales [kg/h]	647.7	619.8	584.5	541.5	491.4
Overall Profit	491.3	486.9	466.3	433.0	389.5
State	State 16	State 17	State 18	State 19	State 20
Expander Exit Pressure [kPa]	3500	3500	3500	3500	3500
Chiller Exit Temperature [C]	-65.00	-60.00	-55.00	-50.00	-45.00
Cost of Power [kW]	137.6	115.7	102.2	93.24	87.12
Value of LPG Sales [kg/h]	617.1	580.7	535.8	482.4	422.0
Overall Profit	479.5	465.1	433.7	389.2	334.8
State	State 21	State 22	State 23	State 24	State 25
Expander Exit Pressure [kPa]	4000	4000	4000	4000	4000
Chiller Exit Temperature [C]	-65.00	-60.00	-55.00	-50.00	-45.00
Cost of Power [kW]	121.6	101.4	89.07	80.96	75.45
Value of LPG Sales [kg/h]	571.9	527.5	473.8	411.2	342.1
Overall Profit	450.3	426.1	384.7	330.2	266.7

The results can also be displayed graphically:

10. Select **Graph** on the **Case Studies** tab.
11. Select **Setup**.

12. Click the **Display Properties** tab.

Figure 11



13. Size and arrange the windows so that you switch the graph between the three dependent variables.

Figure 12

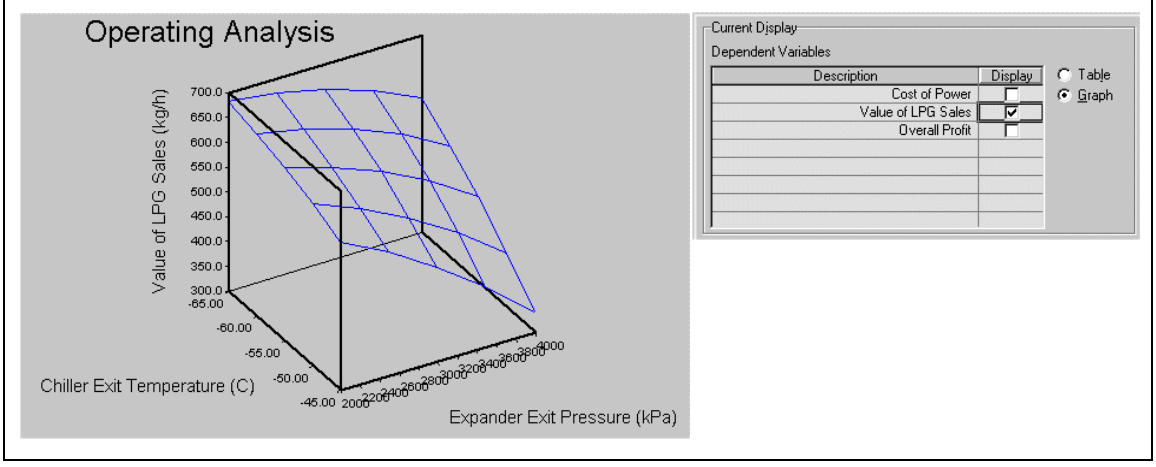
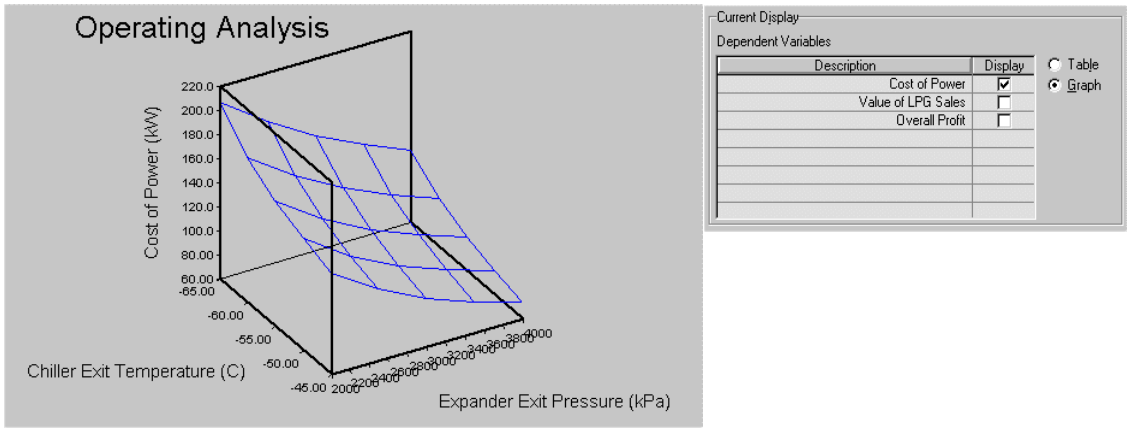
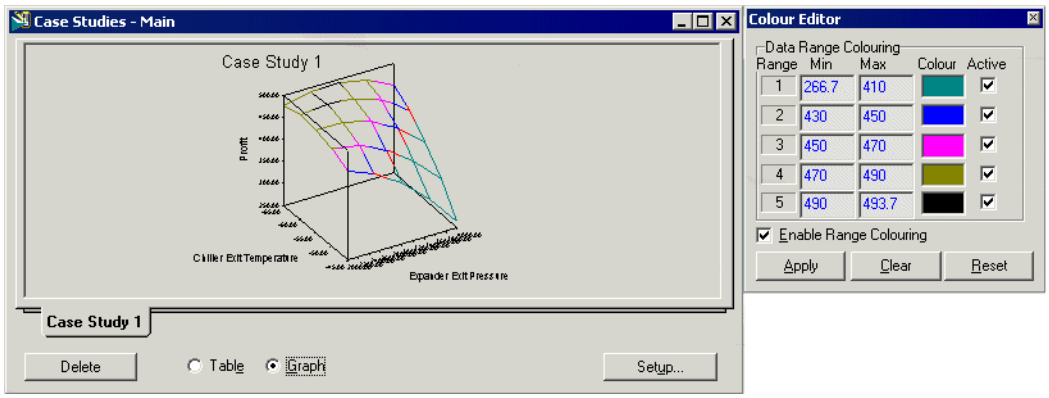


Figure 13



The main interest of course is the Overall Profit and the combination of Chiller exit temp and Expander exit pressure that will give us the maximum Operating Profit.

Figure 14



14. Right-click on the graph, and experiment with the tools available. Try removing **Hidden Lines**, **Rotation**, and **Plane Cutting**.
15. In order to view the graph with the colours shown previously, right-click on the graph and select **Colour Control**. Set the ranges as above. Note that the red colour appears because there is a discontinuity in the entered ranges between 410 and 430.

Save your case!



What can you see about the peak area of the operating surface? How many areas give you more than $470 / hr profit (Turquoise). Are they the same operational conditions?

What could this lead you on to study further?

Your tutor will go through this with you.

16. Before you leave this module, reset the chiller exit temperature to **-62 ºC** and the Turbo expander exit pressure to **28 bar**.

This will allow the next module to perform correctly.

Advanced Recycle Operations

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
06 Advanced Recycle Operations

Introduction

This module will introduce you to several advanced topics concerning the operation and convergence of the Recycle unit operation. The HYSYS Recycle logical operation is used to solve looped systems where downstream material is mixed back in upstream in the process. The Recycle is a logical operation; it does not transform the stream that passes through it.

The Recycle operation can be used several times in a given simulation. Every time a recycle operation is added, the total time needed for the simulation to successfully calculate and converge increases. The information presented in this module can be used to reduce this time and achieve greater success with the Recycle unit operation.

Workshop

The export gas compression section of the Turbo Expander plant is to be modified to deal with an additional hydrocarbon stream from elsewhere in the process.

To cope with this additional load it has been decided to replace the existing single export gas compressor with a two-stage compression system. Each compressor is to be equipped with an after cooler and knock out drum. Liquids from each separator are to be recycled back to the previous stage.

Learning Objectives

In this module, you will learn how to:

- Build simulations using Recycles.
- Position Recycle operations for optimum performance.
- Use the various numerical parameters to help reduce convergence time.
- Consider interactions between Adjusts and Recycles and how to control them with Calc Levels.
- Use Simultaneous mode when the flowsheet contains multiple, interacting Adjusts.

Prerequisites

Before beginning this module, you should have a reasonable understanding of the HYSYS program, and be able to add streams and operations, including the Set and Spreadsheet operations.

Structure of this Module

This module is split into four main sections:

- An information section discussing use and positioning of Recycle operations. The key points from this section are summarised on page 14.
- A series of exercises on Recycle positioning.
- A workshop where a new plant section that requires the use of Recycles is added to the Turbo Expander plant model.
- An exercise to illustrate the use of Simultaneous Adjusts.

Recycle Operation Information

Using the Recycle Unit Operation



Recycle icon

The HYSYS Recycle logical operation is used to solve looped systems where downstream material is mixed back in upstream in the process.

HYSYS employs a non-sequential solving method, which allows information to be propagated both upstream and downstream. This allows some looped systems to be solved explicitly (particularly heat recycles, and refrigeration loops). However for material recycles when downstream material is mixed back in upstream, a Recycle operation is needed.



Recycles are sometimes also know as 'Tears'.

The Recycle operation allows HYSYS to solve looped system iteratively. A set of conditions are assumed and used to solve the recycle loop. The assumed values are compared with the calculated values and updated. This is repeated until the values match within a specified tolerance. The Recycle operation now allows information to be transferred both forwards and backwards (i.e. the assumed value to be in either the outlet or inlet stream), although usually information is only transferred forwards (i.e. assumed value in outlet).



Forward and backward information transfer is discussed later in this module.

When the Recycle operation is first added, initial estimates need to be provided for all the assumed values. Typically this is done by allowing HYSYS to solve before closing the recycle loop. This is illustrated in the Workshop.

The Recycle Unit Operation and Dynamic Simulations

The Recycle operation has a role only in Steady State simulations. While operating in Dynamics, it is perfectly acceptable to return a product stream to an upstream operation without using a Recycle operation. If a Recycle operation is used, it will be ignored while operating in the Dynamic mode; the inlet and outlet streams will always be equal.

Positioning the Recycle Operations for Optimum Performance

It is possible to have numerous Recycle operations in a single simulation. When several Recycle operations are used together, the total calculation time can be reduced by carefully selecting the location of the Recycle blocks.

When a user is deciding on a tear (Recycle) location, the first choice is often in the actual recycling stream. This is an acceptable choice if only one Recycle operation is being used. If more than one block is being used, however, a better location may reduce the calculation time needed to solve the simulation.

Try to locate recycles:

- To define multiple streams (i.e. before Tees, after Mixers). See Exercise 2.
- In streams with fixed conditions (e.g. cooler and heater outlets). This means fewer variables need to be iterated on.
- To avoid conflicts with Adjust operations. This is illustrated later in the Workshop.
- In major flow streams. These are likely to be more stable.

Setting the Recycle Tolerances

After the Recycle operation has solved the inlet and outlet streams will match each other within certain tolerances. HYSYS allows the user to set these tolerances to match the requirements of their simulation.

The actual tolerance of the Recycle operation is calculated as the product of the absolute tolerance for the given property (fixed within HYSYS) and the relative tolerance (specified by the user). The absolute tolerance is dependant on the specific property in question, and is set by the HYSYS program. The following table gives the absolute tolerances for each property.



The Internal Vapour Fraction tolerance, when multiplied by the default recycle tolerance, is 0.1, which appears to be very loose. However, in most situations, if the other recycle variables have converged, the vapour fraction in the two streams will be identical. The loose Vapour Fraction tolerance is critical for close-boiling mixtures, which can vary widely in vapour fraction with minimal difference in other properties.

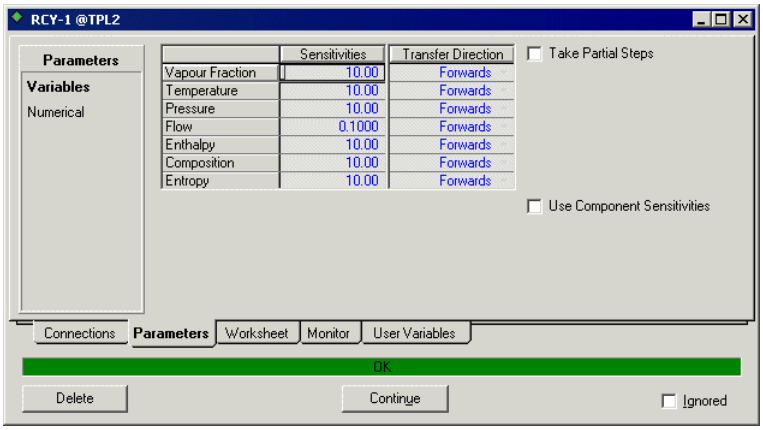
Property	Absolute Tolerance	Internal Unit
Vapour Fraction	0.01	
Temperature	0.01	°C
Pressure	0.01	kPa
Flow	0.001 (this is a relative error)	kgmole/s
Enthalpy	1	kJ/kgmole
Composition	0.0001	

Ticking the Use Component Sensitivities checkbox allows different tolerances to be used for each component.

While the absolute tolerances are set within the program, the user specifies the relative tolerances. The default relative tolerance for all properties is 10, which HYSYS inserts automatically. You are able to specify any value here; remember, however, that smaller tolerances will require more calculation time.

When connected to energy streams, the Recycle operation uses an absolute internal tolerance of 0.1 kW. (kW is the HYSYS internal unit for energy). The tolerance sensitivity multiplier used is Enthalpy.

Figure 1



Tolerances are calculated using HYSYS internal units - These units are essentially the SI System with pressures in kPa (as shown in the table above).

But how does HYSYS calculate the actual tolerance of the Recycle operation? To answer this question, take the example of Temperature. Multiplying the default relative tolerance of 10 by the set absolute tolerance of 0.01 gives a tolerance of 0.1. This means that the temperature of the Recycle's outlet stream must be within 0.1°C (0.18°F) of the temperature of the Recycle's inlet stream in order for the operation to be solved.

For flow rate the tolerance quoted in the table is relative. The absolute tolerance is calculated by multiplying the flow rate in internal units (kgmole/s) by the factor 0.001. For example with a flow rate of 100 kgmole/s and the standard multiplier of 10 the actual tolerance is calculated as follows:

$$\begin{aligned}\text{Actual Tolerance} &= \text{Relative Tolerance} \times \text{Absolute Tolerance} \\ &= \text{Relative Tolerance} \times 0.001 \times \text{Flow rate in kgmole/s} \\ &= 10 \times 0.001 \times 100 = 1 \text{ kgmole/s}\end{aligned}$$

Hence, the flow will be solved if it is within 99 - 101 kgmole/s.

If your simulation contains streams with very low (ppm) concentrations, you may want to set the concentration tolerance to a lower value. Otherwise the default tolerances work well in most applications.

Other Variable Parameters

The Transfer Direction column allows you to select the transfer direction of the variable. There are three selections:

- Not transferred
- Transfer forwards
- Transfer backwards

The *Not Transferred* option can be used if you only want to transfer certain stream variables. For example, if you only want to transfer P, T, composition and flow, the other variables could be set to *Not Transferred*.

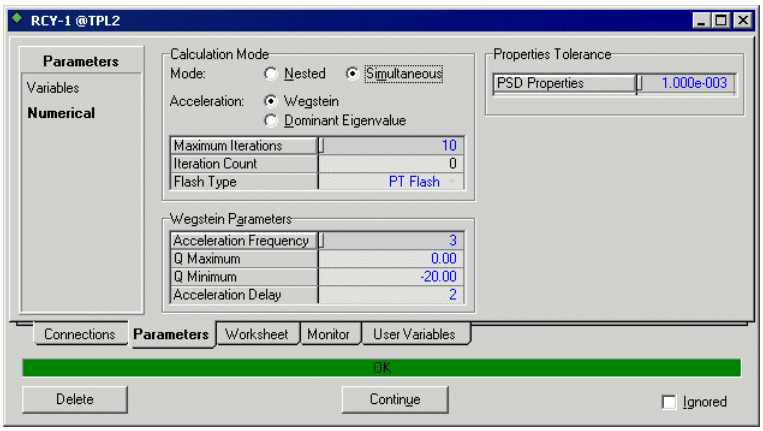
When the checkbox is deactivated, the Recycle operation waits until the inlet stream is completely solved before performing the next calculation step. The default setting for the checkbox is inactive.

Using the Acceleration Parameters

The Recycle operation can be set to use one of two types of mathematical algorithm in order to reach a converged solution faster. The two available acceleration methods are:

- Wegstein Acceleration
- Dominant Eigenvalue Acceleration

Figure 2



Using the Wegstein Acceleration

There are several numerical parameters that define the operation of the Wegstein Acceleration. These parameters will be defined here:

Parameter	Default Value	Definition
Acceleration Frequency	3	The number of iterations per number of accelerations. Using the default, acceleration is applied to every third iteration.
Q_{max}	0	Sets the maximum value for Q in Wegstein equation.
Q_{min}	-20	Sets the minimum value for Q in Wegstein equation.
Acceleration Delay	2	The number of iterations before the first acceleration is applied.

The Wegstein equation is given here. This equation is used to determine the values passed to the outlet stream for each accelerated iteration.

$$X_{N+1} = Q \times X_N + (1 - Q) \times Y_N \quad (1)$$

where: *X = the value in the outlet stream (assumed)*

Y = the value in the inlet stream (calculated)

N = the iteration number

Q = the acceleration factor

HYSYS chooses the value of Q that it will use depending on the amount of change that has occurred between successive iterations. A larger value of Q will be used when the change between successive iterations is large, and vice-versa.

Adjusting the Wegstein Acceleration



A positive Q will help dampen out any oscillations that may occur. The Q_{max} should be increased if, and only if, oscillations are affecting the convergence of the Recycle.

While Wegstein acceleration has been shown to reduce the number of iterations needed to converge a Recycle operation in most cases, there are a few cases in which the default Wegstein parameters will not help to converge the Recycle operation. In these cases it is necessary to reduce the amount of acceleration, or to ignore it all together.

Setting the Q_{min} value to a smaller negative number will reduce the amount of acceleration. The acceleration can be ignored completely by the Recycle operation if the Acceleration delay is set to a high enough value. Typically, Recycle operations will converge in less than 10 steps; therefore, setting the Acceleration delay to a value much larger than 10 means that acceleration will not occur.

Very rarely, the Recycle operation will oscillate as it converges on a solution. If you find this happening in your simulation, you can increase the value of Qmax to a small positive value. This will provide a damping effect that will, hopefully, reduce the oscillating behaviour.

Using Successive Substitution

In the rare cases in which Wegstein acceleration will not help to reduce the number of iterations in the Recycle operation, it may be necessary to set the operation to use Successive Substitution in order to reach convergence.

Successive substitution is when the recycle's outlet stream properties are rewritten with the inlet stream properties without any type of acceleration applied. This is considered the most stable solving method; however, it is also the slowest.

This can be accomplished using one of two methods; either set the Acceleration Delay term to a large value, e.g.100, or set both Qmax and Q_{min} to 0.

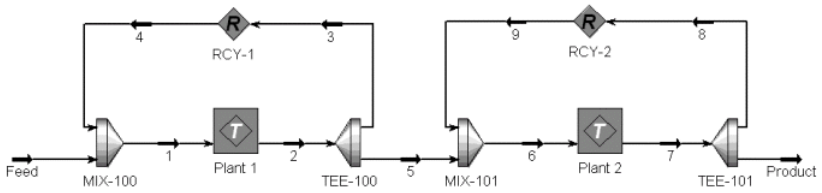
Using the Dominant Eigenvalue Acceleration Method

The Dominant Eigenvalue Acceleration method is less adjustable than the Wegstein method. It is recommended for Recycle operations in systems that are non-ideal and/or where strong interactions exist between the components. This method has the advantage of examining the interactions between the variables during the acceleration process.

Information on Multiple Recycles

When installing multiple Recycle operations, you have the choice between Nested or Simultaneous solution. They should be used as follows:

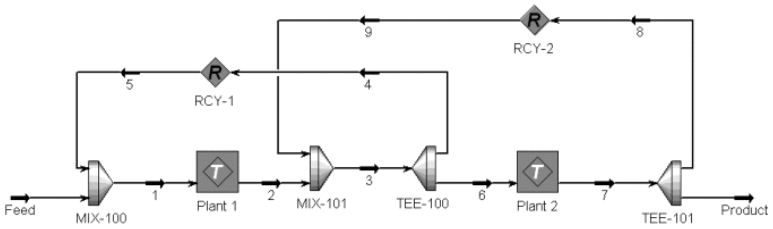
- **Nested** - Single Recycle or multiple, non-connected recycles





In the Simultaneous example, the number of recycle operations may be reduced to one if placed in the correct stream.

- **Simultaneous** - Interconnected, interacting recycles



When Recycle operations are selected as Simultaneous, they will not be calculated with the other unit operations. Instead, they are listed in a separate solver, and calculated only after the ordinary solver has finished calculating all other unit operations in the flowsheet.

Some Troubleshooting Tips

Typically, Recycle operations will converge in less than 10 steps. However, if the tolerances have been reduced, or the system is non-ideal, it may take more than the 10 iterations that HYSYS has set as the default limit. Once the limit has been reached, HYSYS will stop and ask the user if it should leave the operation unconverged, or continue for 10 more iterations.

If your Recycle operation has not converged in 10 iterations, it may be advantageous to stop the calculations and examine the flowsheet.

Monitoring the Recycle's Calculations

The progress of the calculations performed by the Recycle operation can be checked on the Monitor page of the operation's property view. This is useful if the Recycle is having problems converging. The results can be seen in tabular format, or in a plot format.

Choosing a Flash Type

It is possible to choose the type of flash that the Recycle operation will perform. The default choice is a PT flash.



Composition and Flow rate values are always passed through the Recycle operation, regardless of the flash type.

A PT flash means that pressure (P), temperature (T), and composition values are passed through the Recycle operation, and other variables (vapour fraction (V), enthalpy (H), and entropy (S)) are calculated in the other stream.

Choices of flash type include PH, PV, PS, and TV. While a PT flash will be sufficient for most applications, a PH flash is a better choice for very pure recycle streams. When dealing with very pure streams, a small change in T can make a big difference in H. Take a pure water stream at atmospheric pressure, for example, the enthalpy (H) of that stream will be very different if the temperature is 99.9°C (211.9°F) or 100.1°C (212.1°F).

Information Summary

Using the Recycle Operation

- The Recycle operation is used to solve looped systems where downstream material is mixed back in upstream.
- Initial estimates are needed for all assumed values.

Recycle Tolerances

- Calculated as the product of the relative tolerance (user specified) and the absolute tolerance (set in the program).
- Tolerances are calculated using HYSYS internal units.
- Recycle streams that have very low concentrations of important components, may require lower relative tolerances for the composition specification.

Using the Acceleration Parameters

- There are two types of acceleration available: Wegstein and Dominant Eigenvalue. Wegstein is the most common, and Dominant Eigenvalue is recommended for simulations where strong interactions exist between the components.
- Wegstein acceleration can be controlled using the four factors available: Acceleration Frequency, Q_{min}, Q_{max}, and Acceleration Delay.

Multiple Recycles

When using Multiple Recycles set the Calculation mode as follows:

- **Nested** - Single Recycle or multiple, non-connected recycles
- **Simultaneous**- Interconnected, interacting recycles

Flash Types

- The flash type can be changed.
- Use default PT flash choice for most cases
- PH flash is better for very pure recycle streams.

Recycle Positioning Exercises

Exercise 1

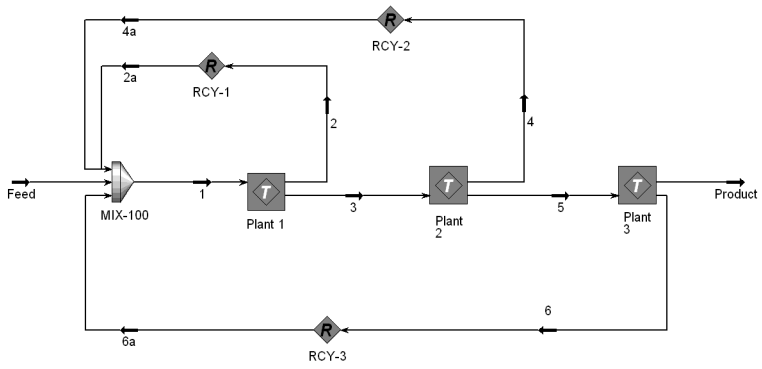
Examine the following PFD. This Flowsheet has three physical recycles and three HYSYS recycle operations.



What is the minimum number of recycle operations that are needed? _____

Where should the recycle operation(s) be positioned? _____

Figure 3 - Exercise 1



Exercise 2: Adding a Recycle

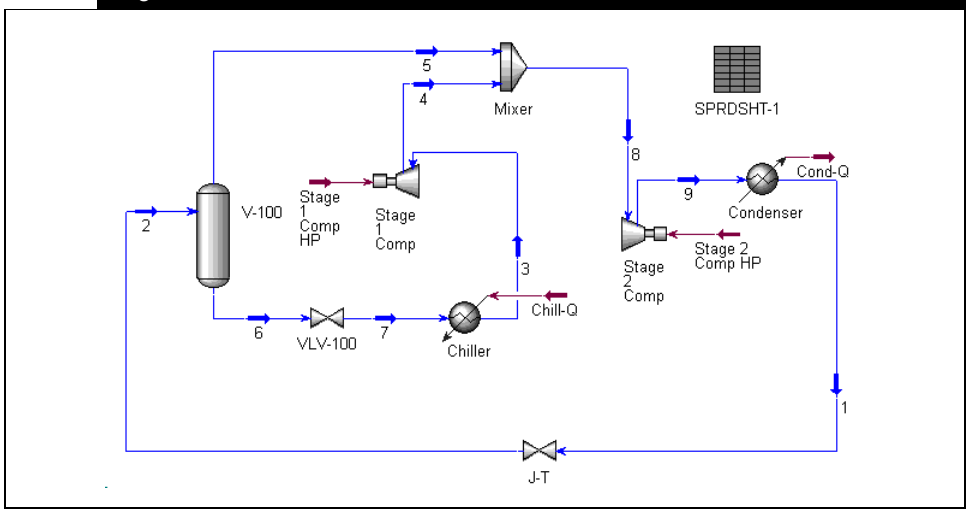
Assume the condenser outlet stream is fully defined (except flow rate), the chiller duty and outlet conditions are known, the pressure drops across the condenser and chiller are known, and the stage 1 compressor outlet pressure is known.



How many recycles are needed in this flowsheet, where should they be placed, and why?

_____

Figure 4 - Exercise 2



Exercise 3

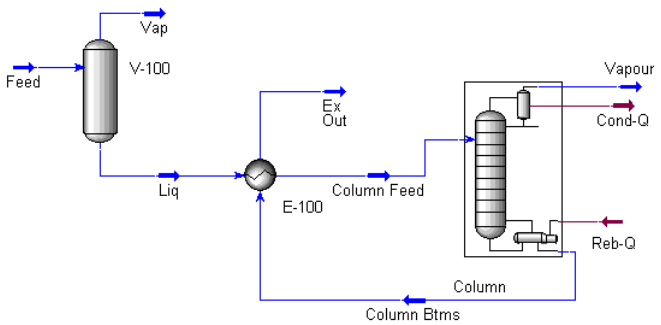
Assume that the Feed is fully defined, Shell and Tube Side pressure drops are known, as well as the Column Feed temperature.



How many recycles are needed in this flowsheet, where should they be placed, and why?

_____

Figure 5 - Exercise 3



Exercise 4

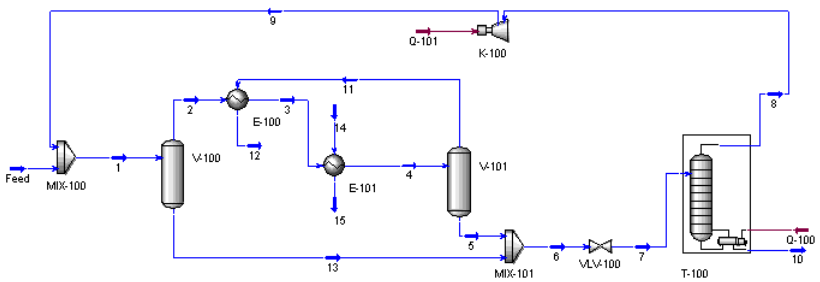
Assume the Feed is completely defined, shell and tube side pressure drops for E-100 and E-101, and the temperatures of streams 3 and 4 are known.



How many recycles are needed in this flowsheet, where should they be placed, and why?

_____

Figure 6 - Exercise 4



Exercise 5

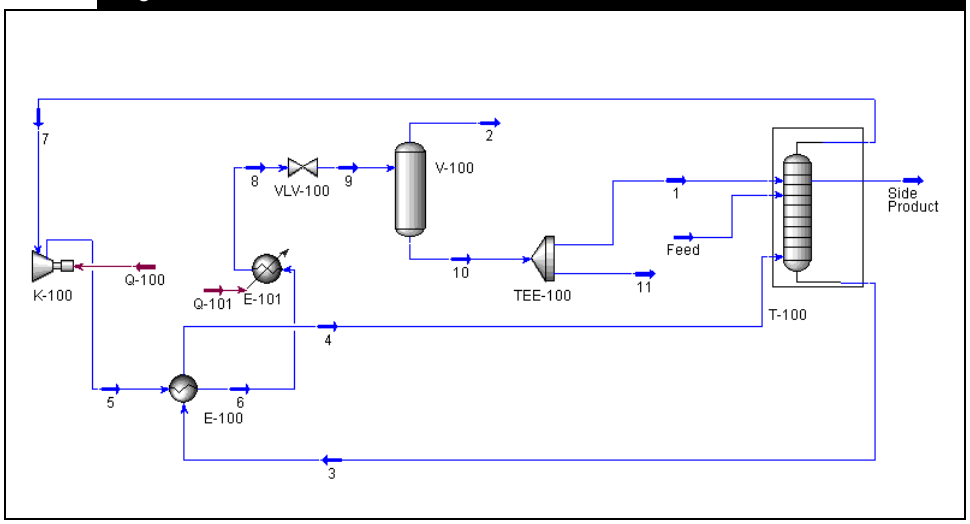
Assume the Feed is completely defined, and the shell and tube side pressure drop for E-100 are known.



How many recycles are needed in this flowsheet, where should they be placed, and why?

_____

Figure 7 - Exercise 5



Exercise 6

Examine the three stage compression plant PFD on the next page.

This flowsheet has five physical recycles and six HYSYS recycle operations.



What is the minimum number of recycle operations that are needed?
_____

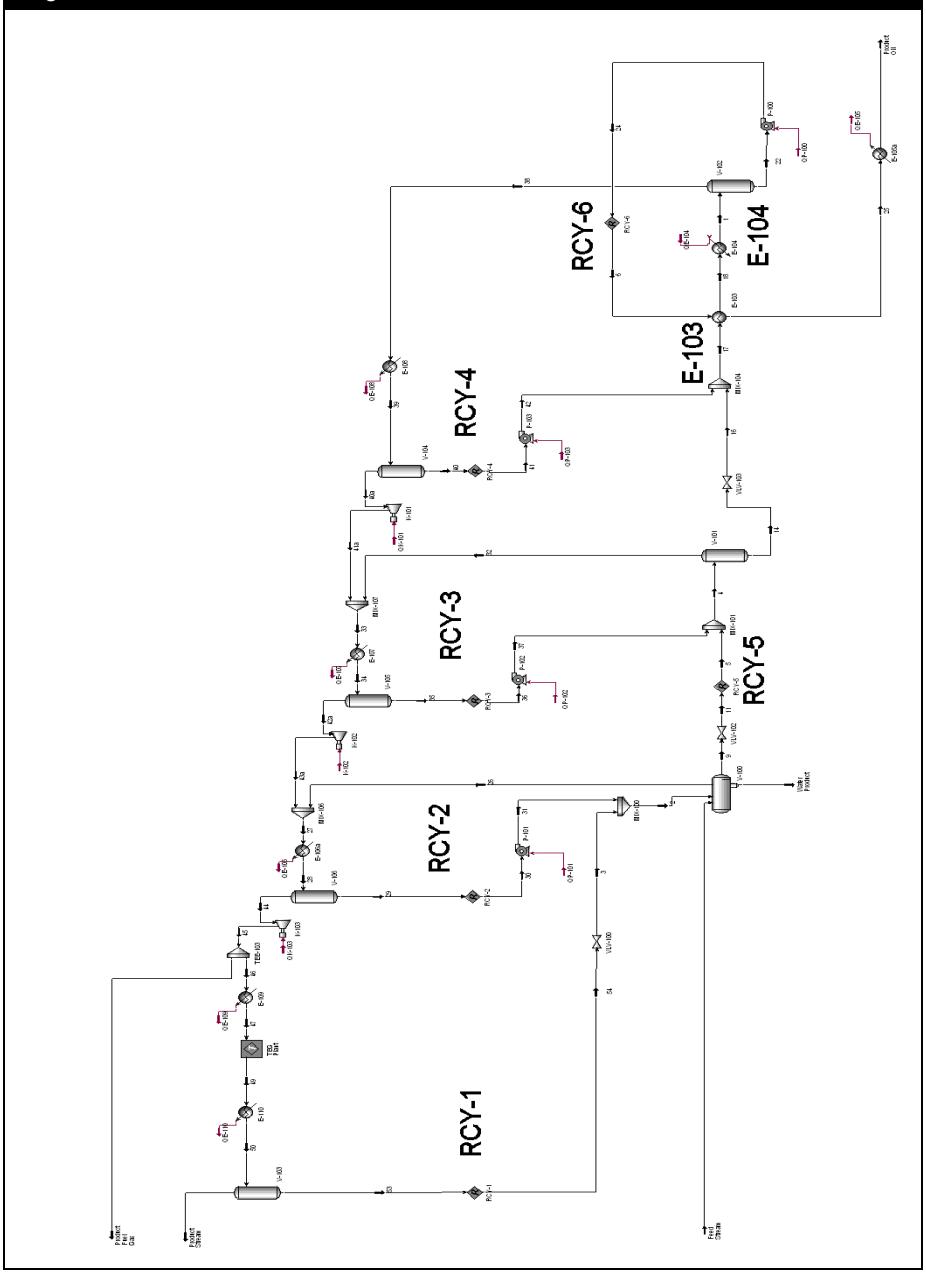
To answer this question consider all the information about recycle positioning in this module, and note the following:

- Every Exchanger in the PFD has pressure drops defined
- E103 has a UA specified
- E104 has an outlet temperature specified



Where should the recycle operation(s) be positioned?_____

Figure 8 - Exercise 6



Workshop

Building the Simulation



Don't worry if you haven't built the Turbo Expander plant case. The file "ADVI_GettingStarted_Soln.hsc" contains this case.

This module will continue with the case built in the Getting Started module. This is a Turbo Expander plant with an export gas compressor.

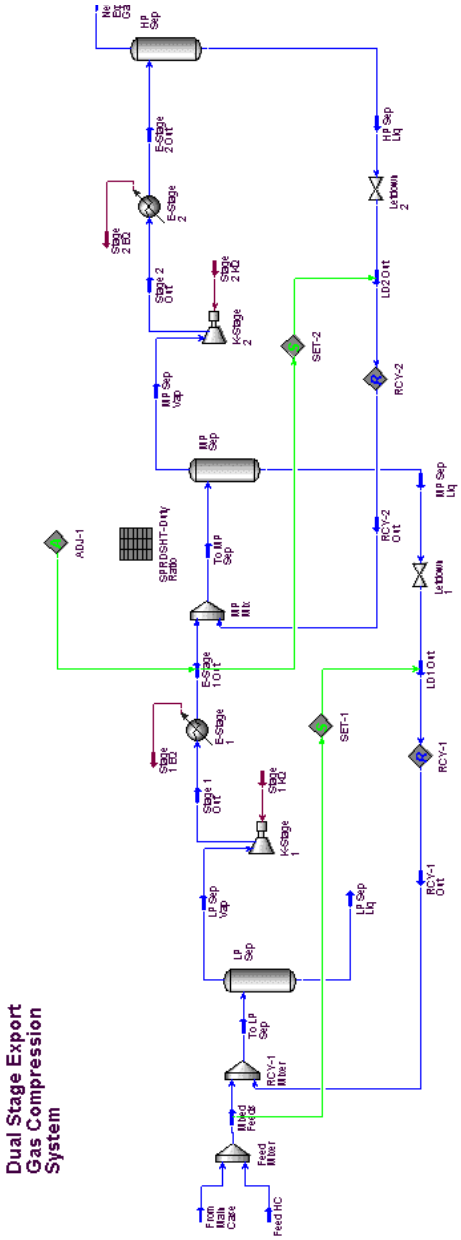
In this workshop the existing compressor will be replaced with a two-stage compression system. Each compressor is to be equipped with an after cooler and knock out drum. Liquids from each separator are to be recycled back to the previous stage.

The reason for this modification to the plant is that an additional hydrocarbon stream must be processed.

To simplify the main PFD the two stage export gas compression system will be constructed in a new sub flowsheet.

Process Overview

Figure 9

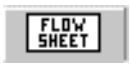


Add the New Feed Stream

Add the **Feed HC** stream with the following conditions and composition to the main flowsheet.

Name	Feed HC
Temperature	35°C (95°F)
Pressure	3000 kPa (435.1 psia)
Flow rate	6000 kgmole/hr (13228 lbmole/hr)
Component	**Feed Stream**
Nitrogen	0.005
CO2	0.015
Methane	0.32
Ethane	0.24
Propane	0.11
i-Butane	0.075
n-Butane	0.075
i-Pentane	0.065
n-Pentane	0.05
n-Hexane	0.045

Add a New Sub-Flowsheet



Object Palette Sub Flowsheet button

Add a new sub flowsheet and choose to **Start with a Blank Flowsheet**.

Make Feed Connections



The Allow Multiple Stream Connections option allows any stream to be connected anywhere.

Without it you can only make a feed connection for a stream that is not already connected as a feed stream elsewhere.

Rather than delete the existing export gas compressor, the new system will be connected in parallel on the HYSYS PFD to allow for comparison (i.e., the full export gas flow rate will go through both export gas compression options).

In order to allow this the "Multiple Stream Connections" feature of HYSYS must be enabled.

1. Go to the **Session Preferences** window (via the **Tools-Preferences** menu).
2. Tick the **Allow Multiple Stream Connections** option on the **Simulation** page in the **Options** tab.

Now stream 13 can be connected to the Sub flowsheet even though it is still the feed stream to the existing export gas compressor.

3. In the Sub-flowsheet operation window on the **Connections** tab, connect up streams **13** and **Feed HC** as feed streams to the sub-flowsheet by using the dropdown menu under external stream.
4. Review the **Transfer Basis** settings, here the default PH flash basis will be used.
5. Change the **Name** of the sub-flowsheet to **Export Gas Compression**.

Build the Flowsheet Without the Recycle Operations



If a stream name is changed in the sub-flowsheet, the name of the linked stream in the main PFD is not changed.

You can find some of the required stream and operation names by referring to the PFD in the Process overview.

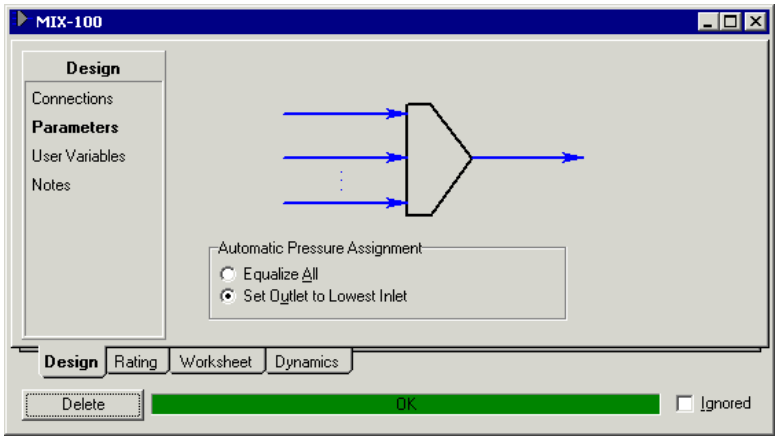
1. Ensure you are in the sub-flowsheet environment.
2. Rename stream **13** to the more meaningful **From Main Case**.
3. Mix the two feed streams. Name the outlet **Mixed Feeds**.

When mixing streams at different pressures the HYSYS Mixer operation offers a number of pressure assignment options. These can be found on the **Parameters** page of the **Design** tab.



The Equalize All options will set the pressure of any connected streams so they are all equal.

Figure 10



4. Ensure the **Automatic Pressure Assignment** option for the Feed Mixer is set to **Set Outlet to Lowest Inlet**.

Now the product stream from this first mixer is always at the lowest pressure of either of the two feed streams.

5. Add another mixer ready to take the returned liquid from the first compression stage. The Mixer will have two feeds: Mixed feeds and one from the recycle. For now, just set Mixed Feeds as the inlet. Call the output To LP Sep.

The mixed feed streams are then fed to a separator, compressor, and after cooler.



Since HYSYS knows the Cooler outlet pressure and pressure drop, it can back calculate the Compressor outlet pressure.

6. Install a separator. Call it LP Sep. The vapour stream is named LP Sep Vap and the liquid stream is named LP Sep Liq.
7. Install a compressor and a cooler. Use the following parameters:

Compressor	
Inlet	LP Sep Vap
Outlet	Stage 1 Out
Energy	Stage 1 KQ

Cooler	
Inlet	Stage 1 Out
Outlet	E-Stage 1 Out
Pressure Drop	25 kPa
Energy	Stage 1 Eq

Set the temperature of stream **E-Stage 1 Out** to **30°C** and the pressure to **5000 kPa**.

8. Install another mixer in preparation to take the second stage returned liquid. At the moment this will only have the first stage after cooler product as its feed stream. Call this mixer **MP Mix** and its product **To MP Sep**.

The second compression stage is an exact copy of the first. Hence here is a good place to make use of HYSYS' **Copy / Paste** functionality.

9. Select the entire first compression stage from the LP Separator, to the after cooler product stream (EStage 1 Out).
10. Right click somewhere on the PFD and choose **Cut/Paste Objects ...**. **Copy Selected Objects** from the pop up menu that appears.



Since the objects are being pasted immediately to the same flowsheet, the Clone function could have been used. This is equivalent to Copying then Pasting.

A question box may pop up if you did not also select all the attached streams for any of the operations you selected.

11. Unselect the objects, then right click on the PFD background and choose **Paste Objects** from the fly out menu.
12. HYSYS automatically renames the pasted objects so the stream names will need to be changed. Rename streams according to the flowsheet on page 20**.**
13. Install a final knock out drum and the product gas stream.
14. Modify the second compression stage so that the product gas will be at **70 bar**.

The liquid from the MP and HP Sep knock out drums is to be returned upstream, and hence must be flashed to the appropriate pressure. Since this may be altered in the design process it is sensible to link the returning liquid pressures to the compressor inlet pressures.



Object Palette Set icon

15. Install valves and valve outlet streams on the MP and HP Separator liquid products. Use a **Set** operation to make the valve outlet pressure the same as the appropriate compressor inlet pressure.

Think carefully about the source streams for these pressures. Ensure the source stream is upstream of any operations effected by the returning flashed liquid.

Save your case!

Installing the Recycles



The physical recycle is often a convenient place to put the Recycle operation initially. Although it is not always the best place!

The PFD is now ready to add recycle operations. Initially Recycles will be added in the physical recycle streams.

16. Install a Recycle on the outlet of the let down valve from the MP separator.
17. Add an outlet stream.

In this case (as in the majority of all situations) the default parameters are appropriate.



Object Palette Recycle icon

18. Review the settings on the **Parameters** tab.
19. Connect the recycle outlet into the RCY-1 Mixer.

The first recycle will then iterate to a solution.

20. Repeat this procedure for the second liquid return.

Save your case!

Analysing the Results

Examine the convergence process for the Recycles.

21. Open the Recycle property view and look at the **Tables** page of the **Monitor** tab.



How many iterations did each Recycle need to converge? _____

22. Look at the **Worksheet** tab for each Recycle. Complete the following table:



Notice that the pressures are exactly the same on both sides of each recycle. Since these are specified by the set, there is no need to for HYSYS to iterate.

Recycle	RCY-1	RCY-2
Inlet VF		
Outlet VF		
Inlet Temperature		
Outlet Temperature		
Inlet Pressure	3000 kPa (435.1 psia)	5000 kPa (725.2 psia)
Outlet Pressure	3000 kPa (435.1 psia)	5000 kPa (725.2 psia)
Inlet Molar Flow		
Outlet Molar Flow		
Inlet Molar Enthalpy		
Outlet Molar Enthalpy		



Do any of the tolerances need to be tightened?

_____

23. Make any tolerance adjustments you feel are necessary.

Make the Product Stream Appear on the Main Flowsheet

Return to the main flowsheet and connect up an External stream for the product HP gas stream.

Save your case!

Exercise



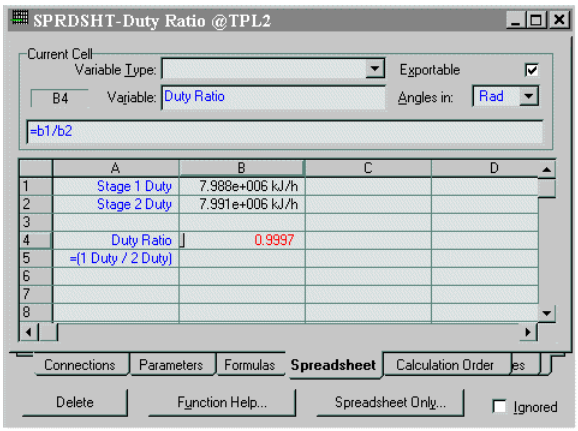
The details of setting up the spreadsheet are not covered here. If you have any problems with this section, ask the instructor.

Part of the design process for this new multi-stage compression plant is to choose an inter-stage pressure to balance the load between the two compressors.

In this exercise you will calculate the duty ratio for the two compressors using a HYSYS Spreadsheet, and then use an Adjust to change the inter-stage pressure such that the load is balanced across the two compressors.

Add a **Spreadsheet** that calculates the Compressor Duty ratio.

Figure 11



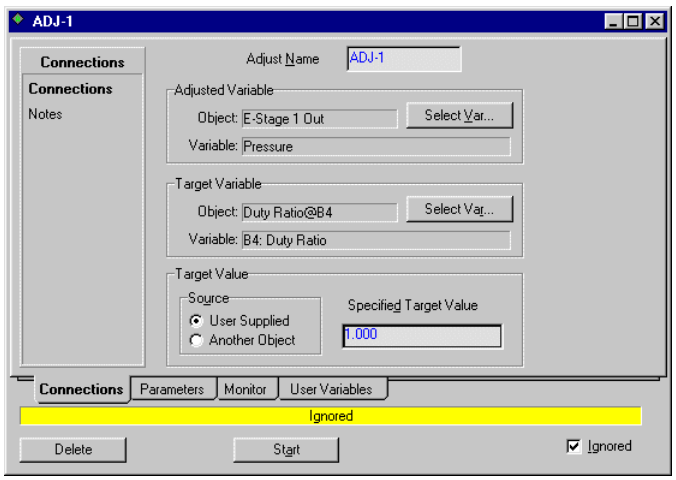
Adding the Adjust


Adjust button

The Adjust operation is another Logical Operation. It will vary the value of one stream variable (the Adjusted variable) to meet a required value or specification (the Target variable) in another stream or operation.

24. Add an **Adjust** operation.
25. For the moment check the **Ignored** box to prevent the Adjust from solving before it's calculation level has been correctly set.
26. Set the Adjust to vary the first stage outlet pressure (Adjusted Variable), until the calculated duty ratio (Target Variable) is 1.

Figure 12



27. On the **Parameters** tab set the following values:

In This Cell...	Enter...
Method	Secant
Tolerance	1 x 10^-3
Step Size	50 kPa (7.25 psia)
Minimum	3000 kPa (435.11 psia)
Maximum	7000 kPa (1015.3 psia)
Maximum Iterations	50

A Brief Introduction to Calc Levels

Calc(ulation) Levels are used to determine the order in which streams and unit operations are solved in HYSYS.

HYSYS uses a non-sequential solution method (i.e. it can solve both backwards and forwards). Each time the solver is triggered it constructs a list of objects (streams and operations) to solve. This list is arranged in order of the object Calc Level. The lower the calc level of an object the higher it will appear in the list, and hence the earlier it will be solved.

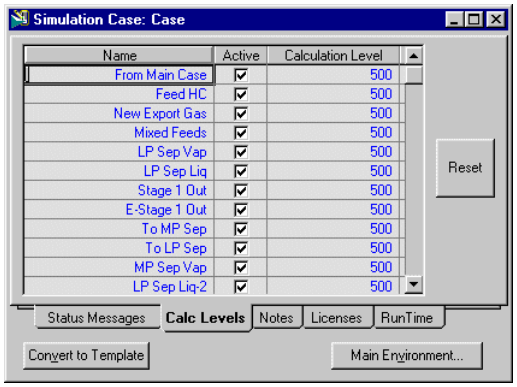
When upstream or downstream objects are affected by new calculations they are added to the solver list. The solver continues until all items are solved.

The default calc levels are listed in the table below:

Item	Calc Level
Material and Energy Streams	500
Ordinary Unit Operations (e.g. Pumps, Heaters, Coolers...)	500
Columns and Sub-flowsheets	2500
Simple Logical Operations (Set and Balance)	500
Complex Logical Operations (Adjust and Recycle)	3500

The Calc Levels of all the objects in a flowsheet can be viewed and changed by going to the **Simulation / Main Properties** menu option and choosing the **Calc Levels** tab.

Figure 13



Hence the default calculation levels for Adjusts and Recycles mean they solve after the rest of the flowsheet, but are at the same level of priority in the solver list.

If the target of an Adjust is inside a recycle loop then sometimes the Adjust and Recycle can conflict. In this situation it is best to set the calc level of the Adjust slightly higher so that it solves after the recycle has solved.

28. Change the Calc Level of the Adjust to 4000 so that it solves after the recycles.
29. Unignore the Adjust, it should now solve.



What inter-stage pressure balances the compressor loads?

Exploring with the Simulation

Since the two Recycles in this simulation are interconnected it would make sense to change their **Calculation Mode** (under **Recycle Parameter** tab, **Numerical** page) to **Simultaneous** rather than **Nested**.

Deliberately displace the simulation from the load balance point (e.g. by changing the inter stage pressure to 40 bar). Compare the solution time between the two calculation modes.



Which Calculation Mode makes the simulation solve fastest?

Try tightening all the tolerances to 0.1, What happens now?

The Author found that for a displacement to 40 bar the **Nested** recycles took around 13 seconds to solve, whereas the **Simultaneous** recycles solved in just 4 seconds. In this case the step size is probably too small since the Adjust requires many small steps to work up to the solution.

Exercise - Backward Pressure Transfer

The Backward transfer feature of the Recycle operation allows the two Sets to be removed. Using the following steps, remove both the Sets in the multi-stage compression sub flowsheet.

1. Ensure the upstream pressure is propagated into the recycle outlet stream by changing the mixer pressure assignment mode to 'Equalize All' (Design / Parameters page).
2. Change the Transfer Direction on the Recycle (Parameters / Variables page).
3. Delete the Set.
4. Move the assumed value from the Recycle outlet stream to the inlet stream.

Challenge

You complain to your boss, Grayson Streed, that your simulation takes too long to converge on your computer because of the multiple recycles and use this opportunity to petition for a new high-speed computer. Grayson has a look at your simulation and denies your request on the basis that you can reduce your convergence time by eliminating one of the recycles in the simulation. Is your request justified or is Grayson correct?

Hint: visualize the two recycle loops in the case, and try to see if there are any overlaps. If you find that Grayson was right, eliminate one of your recycles. Think carefully about the best way to proceed. Remember that you'll need to provide an initial estimate of the recycle outlet stream. What is the best way to obtain this estimate?



With one recycle eliminated how long does the simulation take to recover from a disturbance now?_____

Exercise - Using Simultaneous Adjusts

It has been decided to split the export gas flow between the multi-stage export gas compression system and the existing single stage export gas compressor. There is a fixed product gas flow requirement of 100 MMSCFD from the multi-stage compression plant. The requirement to set the interstage pressure so that the compressor duty is balanced still holds.

This will require the following modifications to the case:

- Remove the current multiple stream connections on stream 13 and add a Tee to split the flow between the two export gas compression systems.
- Add an Adjust to vary the feed gas flow to the multi-stage compression system.

The compression sub-flowsheet will therefore have two Adjust operations. Since changing either adjusted variable (feed gas flow or interstage pressure) will effect both target variables (compressor load balance and product gas flow), if the Adjusts are left with the standard

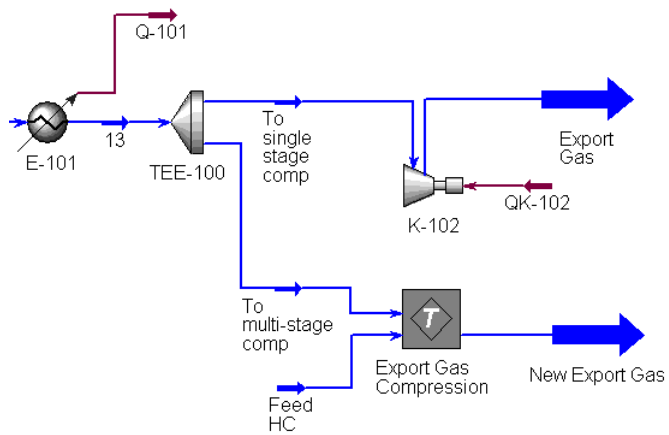
solving method they may interfere with each other while they are solving. This is because each Adjust considers only its adjusted and target values, and does not cooperate with any of the other adjusts.

To prevent this interference the Adjusts can be set to solve simultaneously. This uses a different solution algorithm, which makes the Adjusts solve cooperatively at the end of each flowsheet calculation step.

Use the following steps to adapt your existing model.

1. Break the connections between the main flowsheet stream 13 and the main flowsheet export gas compressor (K-102), and the multi-stage compression sub-flowsheet.
2. Insert a **Tee**, reconnect the streams as below. Remember the Feed gas from the main flowsheet (labelled To multi-stage comp) should be connected to the stream currently called From Main Case on the sub-flowsheet.

Figure 14





MMSCFD is one of the available units for the Molar Flow variable type, it uses the volume of an ideal gas at standard conditions to do the conversion.

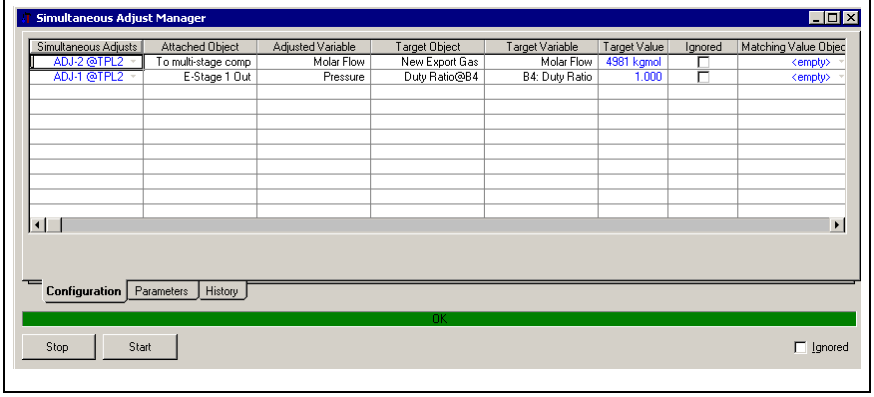
3. Enter an initial flow of **2500** kgmole/h in the sub-flowsheet To multi-stage comp stream.
4. Add an **Adjust** to the sub-flowsheet to vary this flow rate with a target value of **100** MMSCFD in the New Export Gas stream.
5. Set both Adjusts in the sub-flowsheet to use the **Simultaneous Solution** method. (This is accomplished using a checkbox on the Parameters tab.)

6. Start the Adjusts solving

What feed flow rate is required to give an export gas flow of 100 MMSCFD?_____

The Simultaneous Adjust Manager (SAM) allows all the simultaneous adjusts in the case to be controlled in one place. The SAM can be accessed via a button on each Adjust or from the Simulation menu.

Figure 15



The **Configuration** tab shows the set-up of each of the Simultaneous Adjusts, you can view the individual Adjust windows by clicking on their names. The **History** tab shows iteration-by-iteration results for each Adjust.

Parameters Tab

The Parameters tab allows you to modify the tolerance, step size, and max and min values for each Adjust. It also displays the residual, number of iterations the SAM has taken and the iteration status. This tab also allows you to specify some of the calculation parameters as described in the table below:

Parameter	Description
Type of Jacobian Calculation	Allows you to select one of three Jacobian calculations: • **ResetJac**. Jacobian is fully calculated and values reset to initial values after each Jacobian calculation step. Most time consuming but most accurate. • **Continuous**. Values are not recalculated between Jacobian calculation steps. Quickest but allows for 'drift' in the Jacobian therefore not as accurate. • **Hybrid**. Hybrid of one of the above two methods.
Type of Convergence	Allows you to select one of three convergence types: • **Specified**. SAM is converged when all Adjusts are within the specified tolerances. • **Norm**. SAM is converged when the norm of the residuals (sum of squares) is less than a user specified value. • **Either**. SAM is converged with whichever of the above types occurs first.
Max Step Fraction	The number x step size is the maximum that the solver is allowed to move during a solve step.
Peturbation Factor	The number x range (Max - Min) or the number x 100 x step size (if no valid range). This is the maximum that the solver is allowed to move during a Jacobian step.
Max # of Iterations	Maximum number of iterations for the SAM.

Answer Key

Exercise 1

Examine the PFD above. This Flowsheet has three physical recycles and three HYSYS recycle operations. What is the minimum number of recycle operations that are needed?



One. Three are 3 separate loops and they overlap in stream 1

(**Hint**: Visualise the separate loops in the system. Which stream is included in all the loops?)

Where should the recycle operation(s) be positioned?

At the outlet of the mixer

Exercise 2

How many recycles are needed in this flowsheet, where should they be placed, and why?



There is no need for a Recycle operation in this refrigeration loop flowsheet due to the positioning of the specifications.

Exercise 3

How many recycles are needed in this flowsheet, where should they be placed, and why?

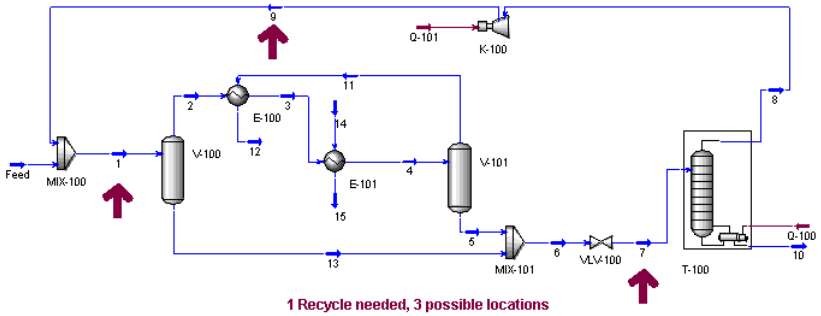


Again, there is no need for a Recycle operation. The column feed stream is fully defined even though Exchanger E-100 hasn't completely solved, hence the column can solve and then E-100 can solve.

Exercise 4



How many recycles are needed in this flowsheet, where should they be placed, and why?



1 Recycle needed, 3 possible locations

There is no need for a Recycle in the small loop containing the two exchangers and separator V-101 because the pressures and temperatures in streams 3 and 4 are known, hence stream 4 can flash fully before the exchangers are solved.

However, the column will not be able to solve unless its feed stream is fully defined, hence a Recycle is need somewhere in the main loop - for example in stream 9, 1 or 7.

Exercise 5



How many recycles are needed in this flowsheet, where should they be placed, and why?

The column requires both feed streams to be fully defined before it will solve, hence a Recycle is required in each of the two loops, for example a recycle in stream 4 and another in stream 1.

Exercise 6 Questions (page 19)



Examine the PFD of a three stage compression plant above. This Flowsheet has five physical recycles and six HYSYS recycle operations. What is the minimum number of recycle operations that are needed?

Three

To answer this question consider all the information about recycle positioning in this module, and note the following:

- All the Exchangers in the PFD have pressure drops defined
- E103 has a UA specified
- E104 has an outlet temperature specified.

Where should the recycle operation(s) be positioned?

See the PFD below for suggested positioning.

Recycles 1 and 2 can be combined at the outlet of the mixer.

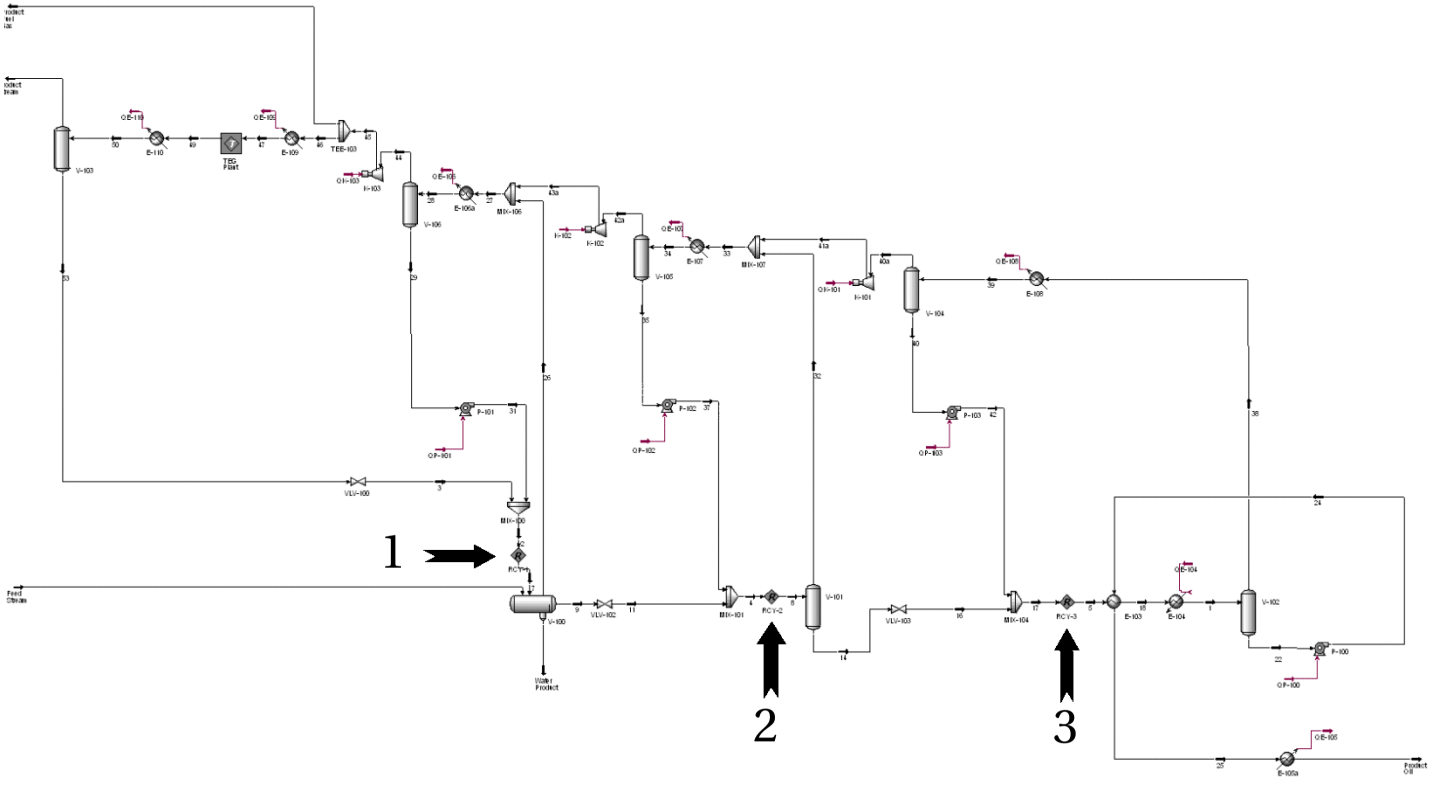
Recycle 5 is superfluous since the first feed separator can solve fully with the other Recycles in place.

Recycle 3 can be positioned anywhere in the loop containing the liquid return from the first stage of compression and the vapour produced from the second feed separator. Here it has been moved to one of the main streams as this is likely to be a more stable. (Larger flows and less fluctuation of flow rate as the case solves.)

Likewise Recycle 4 has been moved to one of the main streams.

Recycle 6 is also superfluous since both feed streams to E-103 are fully defined as E-104 has a specified outlet temperature.

Answer Key PFD



Troubleshooting

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
07 Troubleshooting

Workshop

In this module, you will be presented with cases derived from the HYSYS Steady State course. There have been errors introduced into the cases that prevent them from solving. It is left up to you to find these errors and solve the cases. Of course, the instructor will be willing to assist you in any way that he/she can.

In order to save time, all of the cases can be found on the supplied **Starter** disk. The solved cases are also provided on the **Solutions** disk.

Also included in this module are several troubleshooting tips that you can use both for this module and for troubleshooting your own cases.

Learning Objectives

After competing this module, you will be able to:

- Troubleshoot existing HYSYS cases
- Recognize common problem areas in a HYSYS case
- Understand the message HYSYS gives after a consistency error occurs

Prerequisites

Before beginning this module, you should be able to:

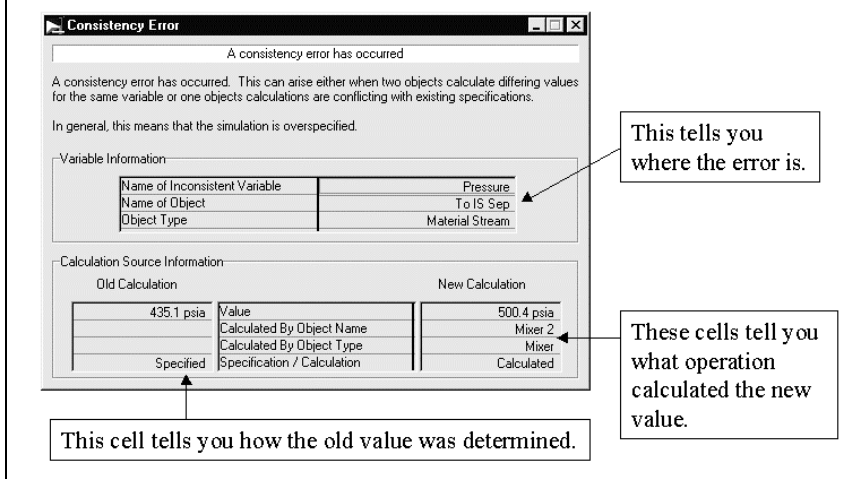
- Navigate the PFD and Workbook Environments
- Add and delete specifications for various unit operations

General Troubleshooting Tips

These tips are given here to help you complete this module, but they are generic so that they can be used when troubleshooting almost all HYSYS simulations. This list was compiled with the help of the Technical Support department and contains several of the problems that they encounter on a daily basis.

1. Always check that the solver is not in "Holding" mode. Whenever a consistency error is encountered in the simulation, the solver is placed in this mode. When the Holding mode is active, streams and operations that are not solved can appear to be solved, and vice versa; this can make the troubleshooting process quite difficult. When the solver is holding, "Holding..." will appear in the HYSYS status bar and the focus will be on the red "traffic light" in the tool bar. The solver can be returned to "Active" by clicking the green "traffic light."
2. Carefully examine all consistency error messages that HYSYS provides. They can often help you find the source of the error. All Consistency Errors will look something like this.

Figure 1



3. Always debug simulations in the direction of the process flow. For example, if the feed streams enter on the left and product streams exit on the right, debug from left to right. It is important that upstream operations be error-free before you attempt to debug downstream operations.
4. The HYSYS Workbook is a handy debugging tool. You can quickly determine which simulation variables are user specified (blue) and which are calculated (black). Remember that in order for the workbook to accurately represent actual conditions, the solver must not be in "Holding" mode.
5. Make sure that all required streams are fully specified. All column feed streams and, usually, all process feed streams are fully defined. Make sure all assumed values in Recycle operations are fully defined. In most cases these will be in the outlet stream, unless the recycle is set to transfer information backwards.
6. Use the HYSYS Status window and Trace window to their full potential when debugging HYSYS simulations. Carefully monitor all messages in both windows; pay special attention to messages in red or blue type.
7. Check that no operations or streams are hidden or ignored. Ignored operations will not be solved, and hidden operations can affect other operations in the simulation resulting in errors. To determine if a case has any hidden objects, and to unhide them, right click on the PFD background and choose Reveal Hidden Objects …
8. When dealing with Adjust operations there are several items to remember.
    - Make sure that the step size and tolerance values are reasonable.
    - Use maximum and minimum values to limit the operation.
    - The adjusted variable must be user-specified or a consistency error will result.

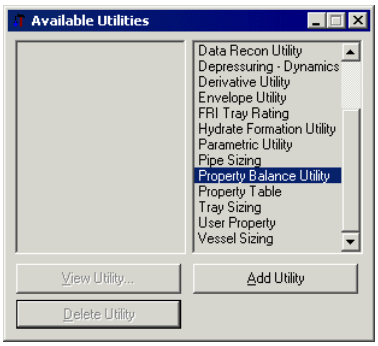
Using the Property Balance Utility to Check Overall Mass and Energy Balances

The property balance utility displays material and energy balances across the whole flowsheet or across selected operations. It can be useful for troubleshooting.

To add a property balance utility:

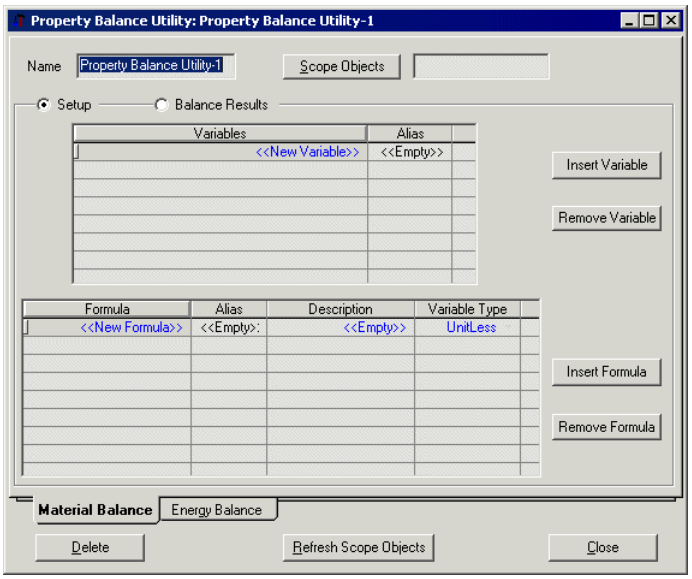
1. Open the **Tools-Utilities** menu, or press **CTRL U**. The Available Utilities view appears.
2. Select the **Property Balance Utility**.

Figure 2



3. Click the **Add Utility** button. The Property Balance Utility view appears.

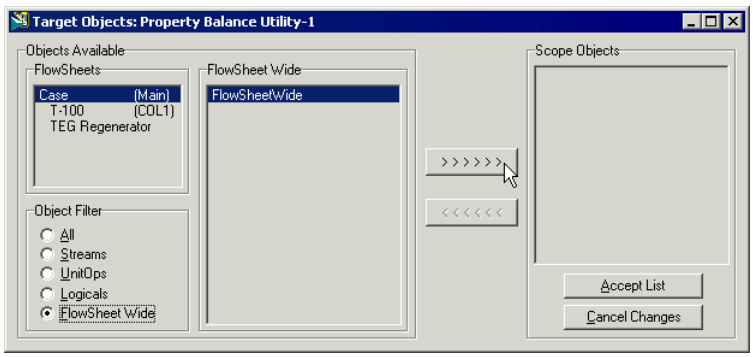
Figure 3



Next, you must choose the scope for the utility.

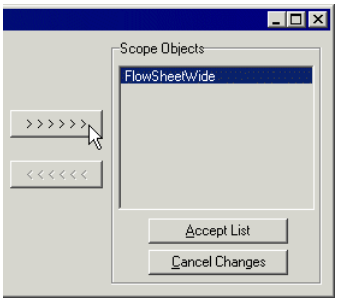
1. Click the **Scope Objects** button.
2. To select the whole flowsheet as the scope, select **Case**. (Ensure the Object Filter is set to the **FlowSheet Wide** radio button as shown.)

Figure 4



3. Select the required flowsheet, and press the >>>>>> button as shown.

Figure 5





Notice that **FlowSheetWide** appears in your Scope Objects list.

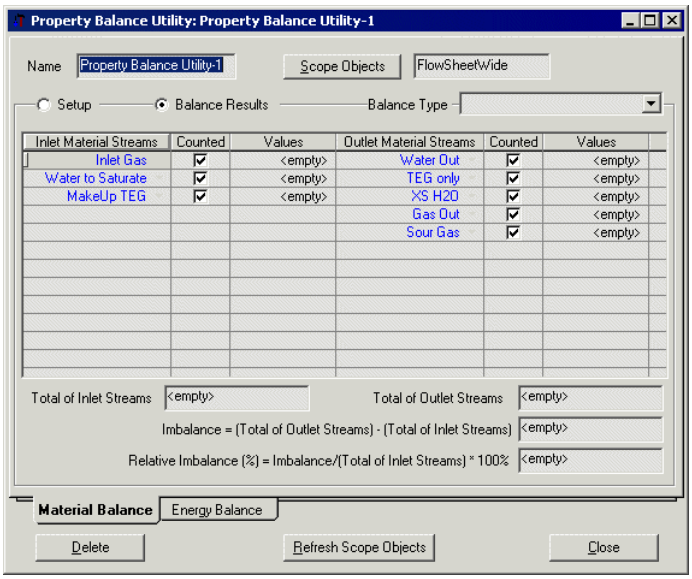


Alternatively the balance can be limited to selected operations.

4. Click the **Accept List** button.
5. Next choose variables to include in the material balance by using the **Insert Variable** button.

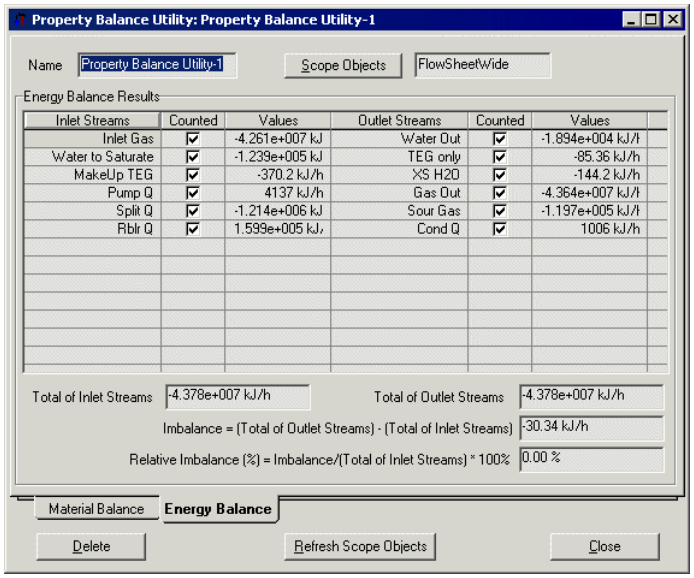
To view material balance results, click the **Material Balance** tab and select the **Balance Results** radio button.

Figure 6



To view energy balance results, click the **Energy Balance** tab.

Figure 7



Column Troubleshooting Tips

Columns are the key operations in many HYSYS simulations, and because their operation is more complex than most HYSYS operations, a separate section of this module is dedicated to tips that you can use to converge all types of column operations.

Degrees of Freedom

Degrees of freedom play an important role in the operation of the HYSYS solver, but their role is most obvious when working with column operations. The DOF can be checked on the **Monitor** page. They must be zero before the column solver will attempt to converge the column. The number of active specifications that the column requires depends on the configuration of the column and can be determined using this formula:

# of active specs = # of side exchangers + # of side draws + # of pumparounds + # of side strippers (1)

In the above formula, reboilers and condensers (any type) are counted as side exchangers.



What will be the required number of active specifications for a column operation with a condenser, three side strippers, three pumparounds, and no reboiler? _____

This is a tricky question because many people forget that each pump around and side stripper also has its own individual side draw. So the actual number of required active specifications is 13, not 7.

When you are defining the active specifications for your column operations, ensure that you are not entering conflicting specifications. For example, with a generic distillation column (a condenser and a reboiler) do not specify both the reboiler duty and overhead rate as active specifications. These values are linked and are really the same specification; so specifying both as active will probably not allow the column to solve.

It is also a good idea to spread the active specifications between the top of the column and the bottom. For example, do not specify the condenser temperature, overhead vapour rate, and reflux ratio as your three active specifications. These specifications all focus on the top of the column; it would be much better if the three specifications were reflux ratio, bottoms draw rate, and overhead vapour rate. This means that the bottom of the column will be partially specified as well.

Another common mistake is that the HYSYS user will specify the product flow rates as their active specifications. This is commonly done when attempting to model an existing column because product flow rates are often readily available. The problem here is that if all of the product flow rates are fixed, HYSYS has no flexibility in determining a solution. It is much better to specify the flow rates as estimates, and use other specifications as the active specs.

Temperature estimates are not required for most columns; however, if they are specified, you may find that the column will converge faster. If you use temperature estimates in your simulations, remember to enter values for the top stage and bottom stage temperatures only; however, if a condenser is used as stage 1, enter a stage 2 temperature also.

Often a steam feed is used to supply energy to the bottom stage in a column. If a steam feed is used, remember to attach a water draw at an appropriate location on the column to remove the excess water.

All feed streams to a column must be fully defined before the column can solve. Columns can not calculate the conditions of a feed stream based on product streams. Likewise, all product streams should not contain any user specified information. A product flowrate specification must be listed with the column's other specifications on the **Monitor** page, not specified as the flow rate for that stream in the worksheet.

The configuration of a column must be defined before the column can solve. This means that the following items must be fully defined:

- All feed streams and their respective feed locations
- Number of Ideal Stages
- The Tower Pressure - specify both a top stage pressure and a bottom stage pressure. If stage 1 is a condenser, specify a stage 2 pressure (a condenser pressure drop) also.
- The Type of Tower - Contactor, Refluxed Absorber, Reboiled Absorber, or Distillation.
- Location and number of side strippers, pumparounds, and side draws, if applicable.

Column Diagnostics

Once all of the required information is entered and the column solver is able to begin calculations, there is no guarantee that the given specifications will lead to a solved column. As many HYSYS users are aware, finding the specific reason for convergence failure can be a difficult and frustrating challenge.

The following five situations can occur if the column fails to converge. Each situation has possible causes, which may help you find the source of the problem.

Condition 1 - The Column fails almost immediately after start-up:

- A vapour-liquid mixture may not be possible at tower conditions. Check BP and DP of all feed streams at tower pressures and ensure that a V-L mixture is possible.
- The mass balance around the column is failing. Check that the product flow estimates (specifications) do not sum to a value that is greater than the feed flowrate.
- A component specification exists for a component that does not exist in the feed stream.
- Columns with no condenser must have a top stage liquid feed, and columns with no reboiler must have a bottom stage vapour feed.

Condition 2 - The Heat and Spec Error fails to converge:

- The column may be unable to meet the desired purity specifications. If this is the case, increase the number of stages.

Condition 3 - The Heat and Spec Error oscillates and fails to converge:

- If the components in the column have similar bubble points, allow looser component specs.
- This condition can also result from a build-up of water in the column, which can be solved by adding a side water draw. This is usually added to the condenser, but may be added at any stage.

Condition 4 - The Equilibrium Error fails to converge.

- Check that the top stage calculated temperature is not too cold. If it is, a side water draw may be required.
- Check the material balance around the column, make sure that your specifications are not preventing the column from solving.

Condition 5 - The Equilibrium Error oscillates and fails to converge:

- This occurs most often with non-ideal towers. In these cases, convergence may be reached by changing the damping factor to a number between 0.4 - 0.6. Another option is to set the damping factor at "Adaptive" rather than "Fixed". This will allow HYSYS to determine its own damping factor.

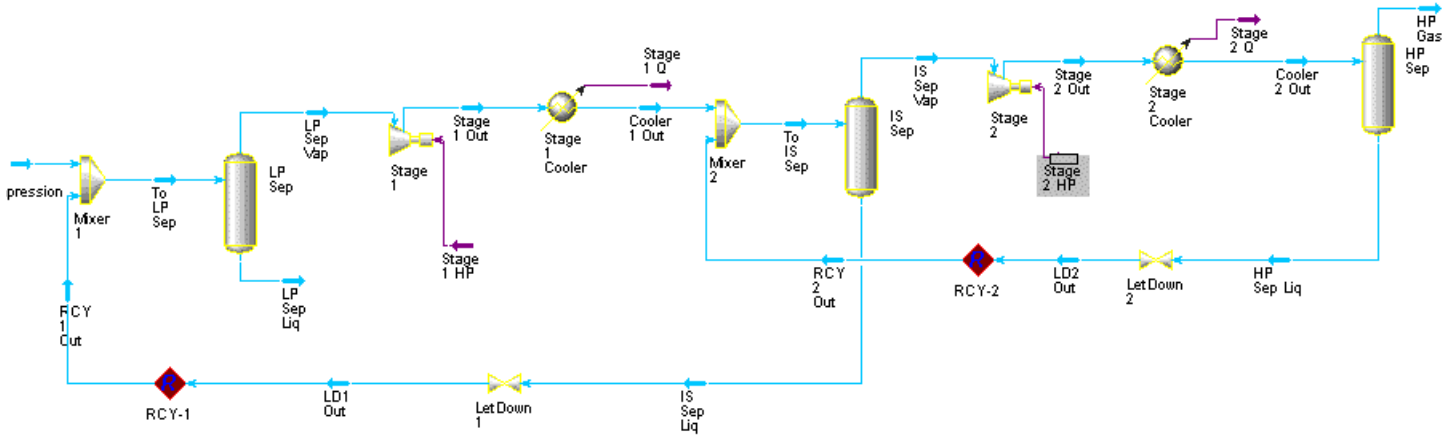
Working on Cases

This is your opportunity to apply the tips that were presented on the previous pages. The instructor will let you work through the problems individually; however, if you require assistance, please, ask the instructor for help.

Case 1

Open the HYSYS case called **Case 1.hsc** located on the **Starter** course disk. This case is a multi-stage compression plant with liquid recycled upstream.

Process Flow Diagram - Case 1





After opening the case, you may notice that it is in Holding mode. To resume the calculations, click the "Green Light" button in the Main Menu Bar.

Attempt to solve the case by adding and deleting specifications as needed. It is a good idea not to delete anything too quickly as you may need the value later on. It is also a good idea to begin at the very beginning of the simulation and work you way through to the end.

There are errors in three places in this simulation that must be removed before the simulation will solve.



What was the first thing that you changed? _____

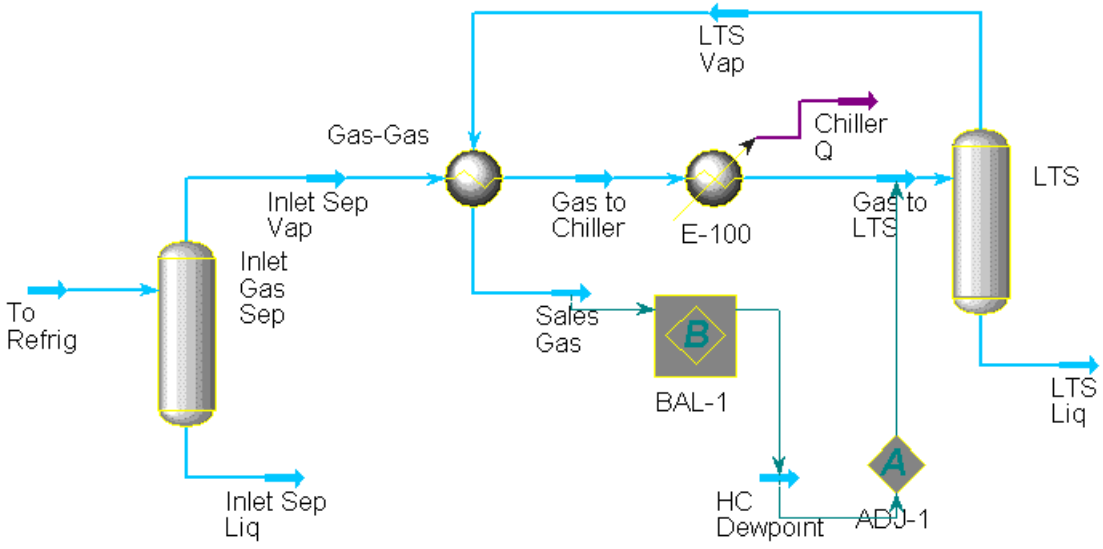
The second? _____

The third? _____

Case 2

Open the HYSYS case called **Case 2.hsc** located on the **Starter** course disk. This case is a simple gas plant where the separator temperature is set to meet a dew point temperature specification on the export gas.

Process Flow Diagram - Case 2



In this case, there are five errors distributed among three operations. Start at the beginning of the simulation; remove all of the errors and solve the simulation.

In order to solve this simulation, you need to think about the purpose of the Balance operation, and the solving behaviour of the Adjust operation. The balance operation can perform material and energy balances over it's connected streams. If the Mole balance type is chosen then component mole flows are balanced, the Mass balance type balances overall mass flows.



What was the first thing that you changed? _____

The second? _____

The third? _____

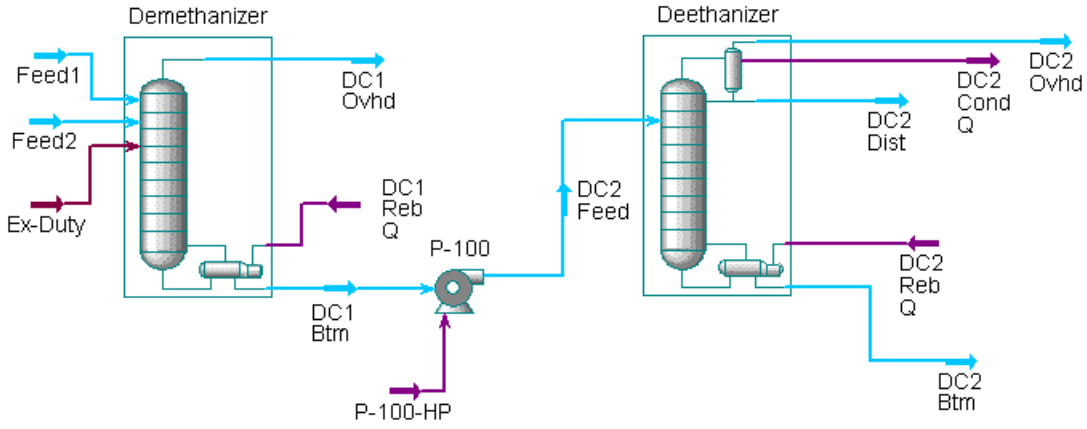
The fourth? _____

And the last thing that you changed? _____

Case 3

Open the HYSYS case called **Case 3.hsc** located on the **Starter** course disk. This case simulates demethaniser and deethaniser columns.

Process Flow Diagram - Case 3



In this case, there are four errors that prevent the two columns from converging. It is important to remember what a column needs in order to solve. Once again, start at the very beginning of the simulation and work your way through the case.



What was the first thing that you changed? _____

The second thing? _____

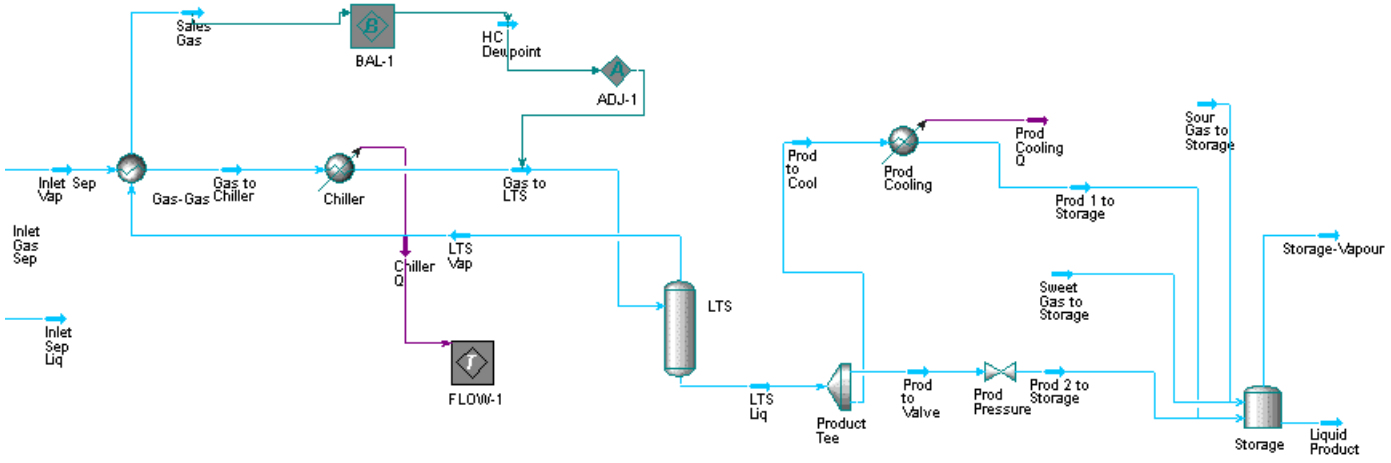
The third thing? _____

And the last thing that you changed? _____

Case 4

Open the HYSYS case called **Case 4.hsc** located on the **Starter** course disk.

Process Flow Diagram - Case 4



In this case, there are only two things that must be changed, finding the errors could prove challenging; things are not always as they appear. Remember to think twice before deleting anything; once it is gone, you might not be able to get it back. In this case, it will help to consider the purpose of every operation.



What operation contained both problems? _____

Why did you change in this operation?_____

Why does this operation need to be specified in this manner? _____

This guide has been prepared based upon questions frequently asked regarding the Dynamic Depressuring utility introduced in HYSYS 3.0. It should provide users with an explanation how to use the utility and correctly interpret the results. It is divided into three sections:

1.0 Overview
2.0 Adding and Configuring the Utility
3.0 Example Problem

1.0 Overview

Why are there two Depressuring utility options?

The original Depressuring utility in HYSYS was a pseudo-dynamic calculation based on a series of steady state calculations. The Dynamic Depressuring utility was introduced in HYSYS 3.0 to allow users to perform proper time-dependant calculations. A HYSYS Dynamics licence is **NOT** required to use this new utility.

What can this utility be used for?

The Depressuring utility can be used to simulate the depressurisation of gas, gas-liquid filled vessels, pipelines and systems with several connected vessels or piping volumes depressuring through a single valve. References to “vessel” in this guide can also refer to piping or combinations of the two.

What types of depressuring calculations can be performed?

There are two major types of depressuring calculations available:

- **Fire Mode** is used to model a vessel or pipe under fire conditions. This mode has three sub-types: Fire, Fire Wetted and Alternative Fire.

- **Adiabatic Mode** is used to model the blowdown of pressure vessels or piping with no external heat supplied.

A more in depth discussion of the different methods follows in Section 2.0.

2.0 Adding and Configuring the Utility

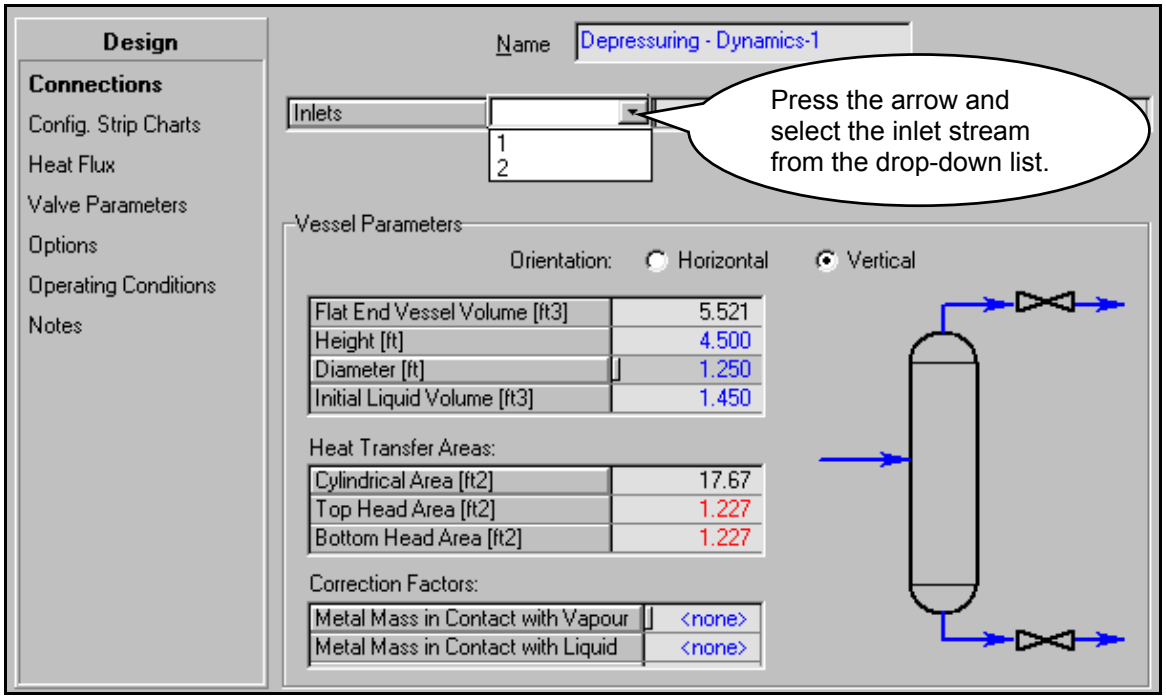
How to add the utility

A Depressuring utility can be added to the case by selecting "Tools" → "Utilities", highlighting "Depressuring - Dynamics" and pressing the "Add Utility" Button. You may note that the original Depressuring model is still shown on the "Available Utilities" menu, this option will be discontinued after version 3.0.1 and all existing models will be converted to the new Dynamic utility.

Connections

How to connect the utility to a stream

On the "Design" tab, "Connections" page, choose the stream that represents the fluid you want to use as the source for the depressuring. If you have a single vessel, for example, the stream would be the feed stream into the vessel. Attaching the stream to the utility is accomplished as shown in the view below.



Entering Vessel Parameters

Ideally, the vessel size will be known and this data can be entered into the appropriate fields on the form shown above. If the vessel size is unknown, then the vessel sizing utility in HYSYS can be used to estimate the required parameters.

The initial liquid volume is normally calculated at the normal liquid level (NLL). The heads of the vessel are not taken into account so the volume will be the liquid in the cylindrical portion only. If the feed stream is two-phase, the equilibrium composition of the liquid will be calculated. If an initial liquid volume is not specified, HYSYS will take a volume equal to the volumetric flow of the feed liquid over one hour. This may be disproportionate to the total vessel volume.

HYSYS does not take account of the heads in a vessel so volumes and areas are calculated as for a cylinder. The total vessel volume is calculated from the diameter and height (or length for a horizontal vessel). To account for piping or head volume contributions, a small amount can be added to the height or length of the vessel.

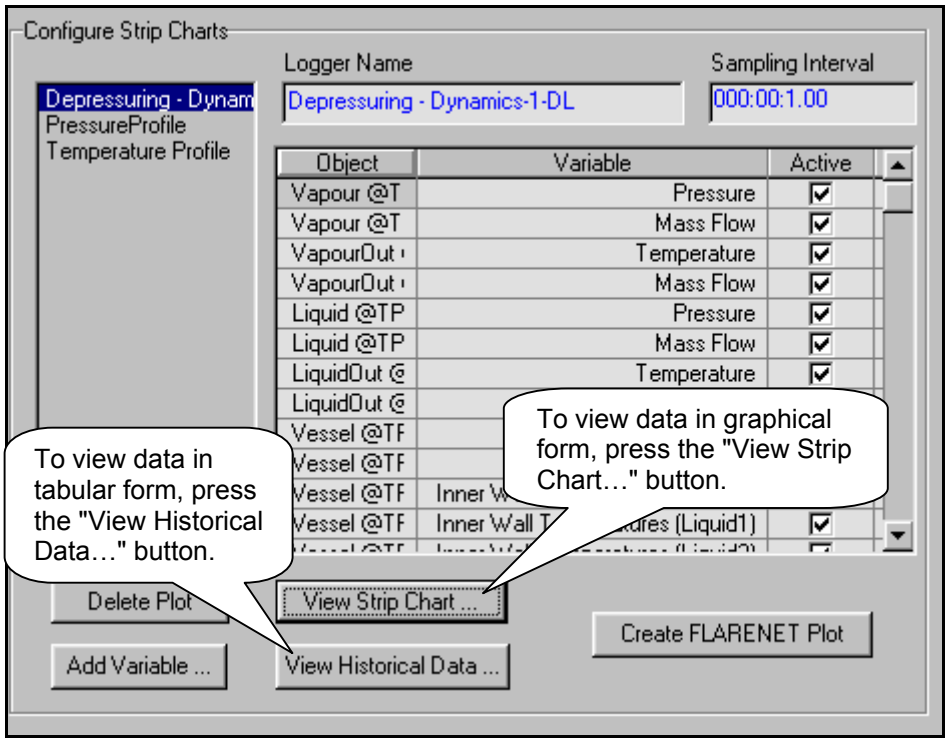
If the condition of the system at settle out are such that the vapour is superheated, HYSYS will **not** allow a liquid inventory. The settle out conditions for mixed sources and volumes are calculated on a constant enthalpy, volume and mass basis.

Correction Factors allow for adjustments to the amount of metal in contact with the top or bottom of the vessel. This can also be used to account for additional nozzles, piping, strapping or support steelwork in close contact with the vessel. HYSYS will use the heat content of this metal when performing the

calculations. This is analogous to adding, for example, ten percent to the vessel mass to account for fittings.

Configure Strip Charts

When the Depressuring utility is run, all data is stored using strip charts. Three default strip charts are added when the utility is added. It is possible to remove variables by deselecting the appropriate variable in the "Active" column. A variable can be added by pressing the "Add Variable" button and selecting it from the list of simulation variables. Any configuration to the strip charts should be done before the utility is run, otherwise any new variables will not be stored.



Heat Flux Parameters

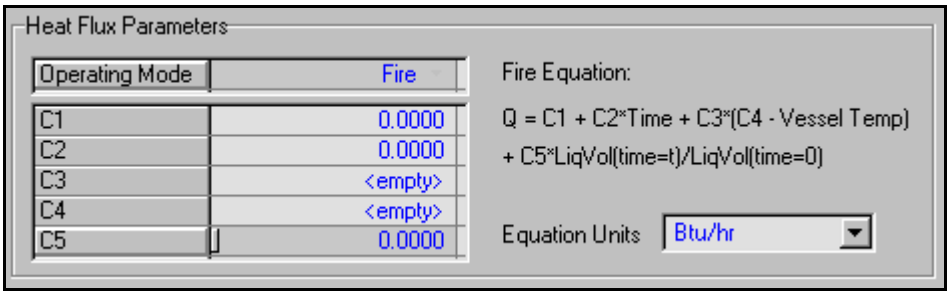
On this page, the type of depressuring to be performed is specified. The different modes and their respective equations are described here.

- **Fire Mode** can be used to simulate plant emergency conditions that would occur during a plant fire. Pressure, temperature and flow profiles are calculated for the application of an external heat source to a vessel, piping or combination of items. Heat flux into the fluid is user defined using the following equation:

$$Q = C_1 + C_2 \times time + C_3\left(C_4 - T_{VESSEL}\right) + C_5 \times \frac{LiquidVolume_{time=t}}{LiquidVolume_{time=0}}$$

The Fire equation can also be used to simulate the depressuring of sub-sea pipelines where heat transfer occurs between seawater and the pipeline. If C_3 was equal to UA, C_4 was equal to T_1 and C_1, C_2 and C_5 were equal to zero, the above equation would reduce to:

$$Q = UA(\Delta T)$$



- **Fire Wetted Mode** uses similar heat flux parameters to those used in Fire mode. Three coefficients: C_1, C_2 and C_3 must be specified. The equation used by HYSYS is an extension to the standard **API** equation for heat flux to a liquid containing vessel. A wetted area is required and used to calculate the heat transfer into the vessel.

  The following notes are based on extracts from Guide for Pressure-Relieving and Depressuring System, API Recommended Practice 521, Forth Edition, March 1997.

  The amount of heat absorbed by a vessel exposed to an open fire is affected by:

  a) The type of fuel feeding the fire
  b) The degree to which the vessel is enveloped by the flames (a function of size and shape)
  c) Any fireproofing on the vessel

  The following equations are based on conditions where there is prompt fire fighting and adequate drainage of flammable materials away from the vessel.

  API Equation (field units) $Q = 21000 \times F \times A^{0.82}$

  Q = total absorption to wetted surface (BTU/h)
  F = environmental factor
  A = total wetted surface (ft^2)

  API Equation (metric units) $Q = 43.116 \times F \times A^{0.82}$

  Q = total absorption to wetted surface (kJ/s
  F = environmental factor
  A = total wetted surface (m^2)

  Environmental Factor

  Table 5 on Page 17 of API 521 lists F factors for various types of vessels and insulation. For a bare vessel, F = 1. For earth-covered storage, F = 0.03. For below-grade storage, F = 0. For insulated vessels, users should consult the reference and select an F value based on the insulation conductance for fire exposure conditions.

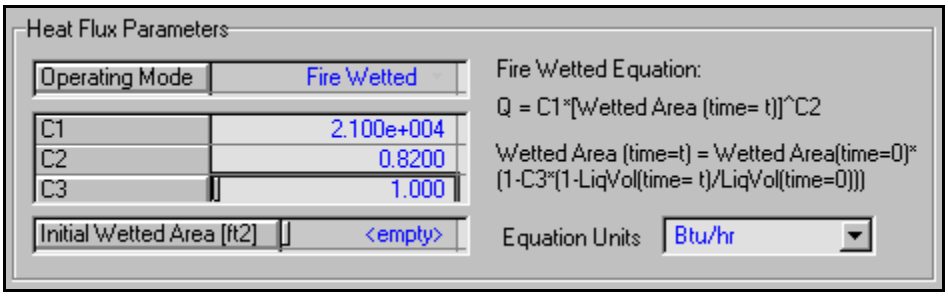
  Wetted Area

  The surface area wetted by the internal liquid content of the vessel is effective in generating vapour when the exterior of the vessel is exposed to fire. To determine vapour generation it is only necessary to take into account that portion of the vessel that is wetted by liquid up to 7.6m (25ft) above the source of the flame. This usually refers to ground level but it can be any level capable of sustaining a

pool fire. The following table indicates recommended volumes for partially filled vessels. Volumes above 7.6m are normally excluded as are vessel heads protected by support skirts.

Type of Vessel	Portion of Liquid Inventory
Liquid full (e.g.: treaters)	All (up to 7.6m)
Surge drums, knockout drums and process vessels	Normal operating liquid level (up to 7.6m)
Fractionating columns	Normal level in the bottom plus liquid hold up from all the trays dumped to the normal level in the column bottom. Total wetted surface only calculated up to 7.6m [1]
Working storage	Maximum inventory level (up to 7.6m)
Spheres and spheroids	Either the maximum horizontal diameter or 7.6m, whichever is greater

[1] Reboiler level is to be included if the reboiler is an integral part of the column.



The HYSYS equation is an extension of the standard API equation. Therefore, in field units, C1 will be 21000 multiplied by the environmental factor, F and C2 will 0.82. (In most cases, C1 will be equal to 21000).

$$Q = C_1 \times \left(WettedArea_{time=t}\right)^{C_2}$$

Wetted area at time t is defined by the following equation:

$$WettedArea_{time=t} = WettedArea_{time=0} \times \left[1 - C_3 \times \left(1 - \frac{LiquidVolume_{time=t}}{LiquidVolume_{time=0}}\right)\right]$$

The following table is an example showing how the C_3 term affects the wetted area calculation. An initial liquid volume of $6m^3$ and a wetted area of 500 m^2 were given.

Time (minutes)	Liquid Volume (m^3)	Volume Ratio	C_3				
			1	0.75	0.5	0.25	0
			Wetted Area (m^2)	Wetted Area (m^2)	Wetted Area (m^2)	Wetted Area (m^2)	Wetted Area (m^2)
0	6	1.0	500.0	500.0	500.0	500.0	500.0
5	4	0.7	333.3	375.0	416.7	458.3	500.0
10	3	0.5	250.0	312.5	375.0	437.5	500.0
15	2	0.3	166.7	250.0	333.3	416.7	500.0

Therefore if a C_3 value of 0 is used, the initial wetted area is used throughout the calculations. This could represent a worst case scenario. Alternatively, if a C_3 value of 1 was used, the volume would vary proportionally with the liquid volume. This would represent a vertical vessel.

HYSYS 3.0.1, Build 4602 KNOWN ISSUE

Depressuring Heat Flux Equation is incorrect if Field units are selected. If the fire wetted equation is used while field units are selected (i.e.: BTU/h), the heat flux equation used by the Depressuring utility will be incorrect. There is a problem with the conversion between SI and Field units. Instead of using the normal API coefficient of 21000, the value of C1 should be multiplied by 7 (i.e.: 147000). This will correct for the unit conversion problem. Because of this defect, the following equations should be:

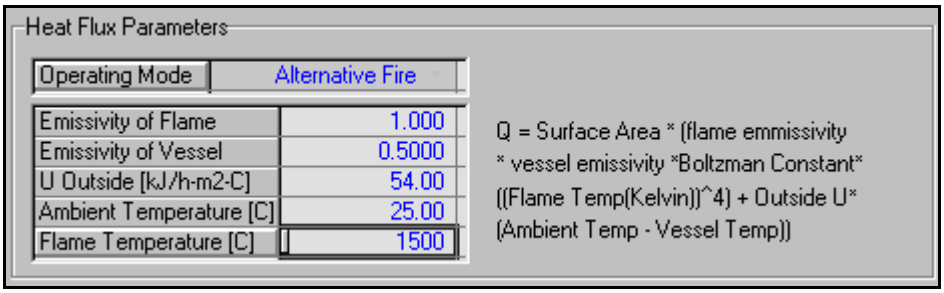
API Equation	**Equation Units**	**Area Units**
$Q = 147000 * F A^{0.82}$	BTU/h	ft2
$Q = 155201 * F A^{0.82}$	KJ/h	m2
$Q = 43.116 * F A^{0.82}$	KJ/s	m2

- **Alternative Fire Mode** uses the Boltzman constant to take into account radiation, forced convection, flame temperature and ambient temperature. The method may be considered as an alternative method to the API standard.

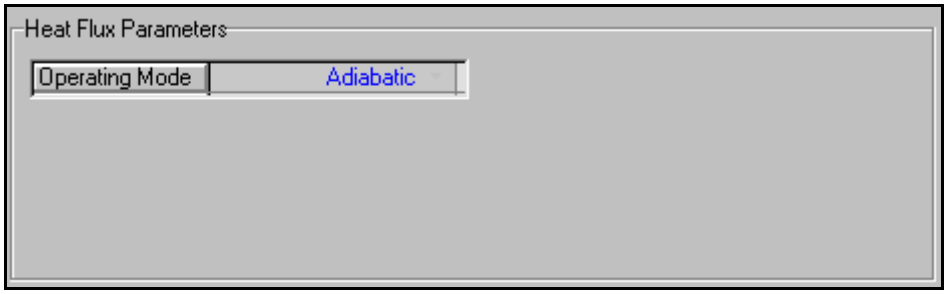
$$Q = A_{total} \times \left(\varepsilon_f \times \varepsilon_v \times k\left(\left(T_f + 273.15\right)^4 - \left(T_V + 273.15\right)^4\right) + outsideU \times \left(T_{amb} - T_V\right)\right)$$

where:

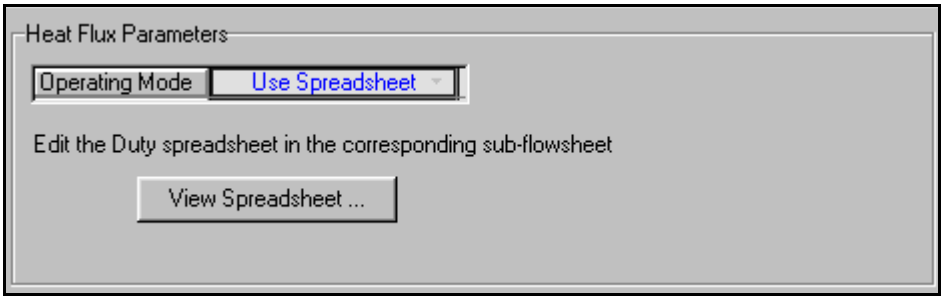
A_{total}	= total wetted surface area	
ε_f	= flame emissivity	generally ranges from 0.2 to 0.5 (for burning heavy HCs)
ε_v	= vessel emissivity	generally ranges from 0.5 to 1 (for polished metal)
k	= Boltzman constant	equals $5.67*10^{-8}$ $W/m^2 K^4$
T_f	= flame temperature	1500 K and upwards
T_v	= vessel temperature	
outside U	= convective heat transfer between vessel and air	
T_{amb}	= ambient air temp	



- **Adiabatic Mode** can be used to model the gas blowdown of pressure vessels or piping. No external heat is applied so no parameters need to be entered in this section. Heat flux between the vessel wall and the fluid is modelled as the fluid temperature drops due to the depressurisation. Typical use of this mode is the depressuring of compressor loops on emergency shutdown.



- **Use Spreadsheet** is an option that allows the user access to the spreadsheet used by the depressuring utility. Values can be altered in this spreadsheet and additional equations substituted for calculation of the heat flux. It is recommended that this option only be used by advanced users.

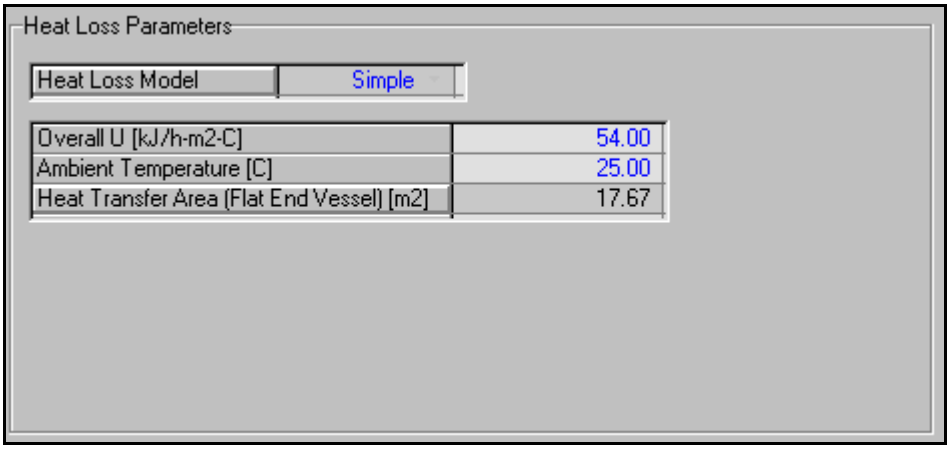


Heat Loss Parameters

There are three types of Heat Loss models available:

1. **None**: does not account for any heat loss
2. **Simple**: allows the user to either specify the heat loss directly or have it calculated from specified values
3. **Detailed**: allows the user to specify a more detailed set of heat loss parameters

Simple Model



- An overall U value can be specified in this section.
- Heat Transfer Area is the cylindrical area of the vessel with no allowance for head area. This value is calculated using the vessel dimensions specified on the "Connections" page.
- Using the Simple Heat Loss Model, heat loss from the vessel is calculated using the following formula:

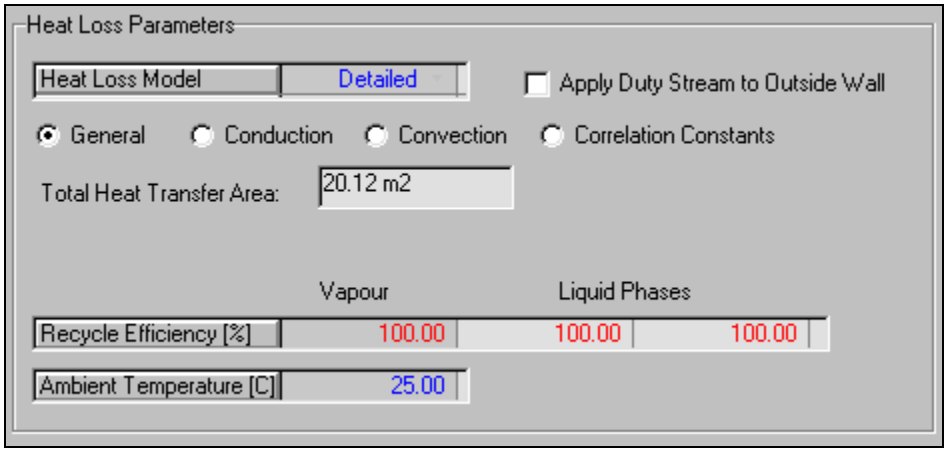
$$Q = UA\left(T_{fluid} - T_{ambient}\right)$$

Detailed Model

The duty can be applied to the vessel wall or directly to the fluid. The former would be used to model a fire and the latter to model a heater. There are four portions of the model to be set up. They are General, Conduction, Convection and Correlation Constants.

General

The General section allows the user to manipulate Recycle Efficiencies and the ambient temperature.

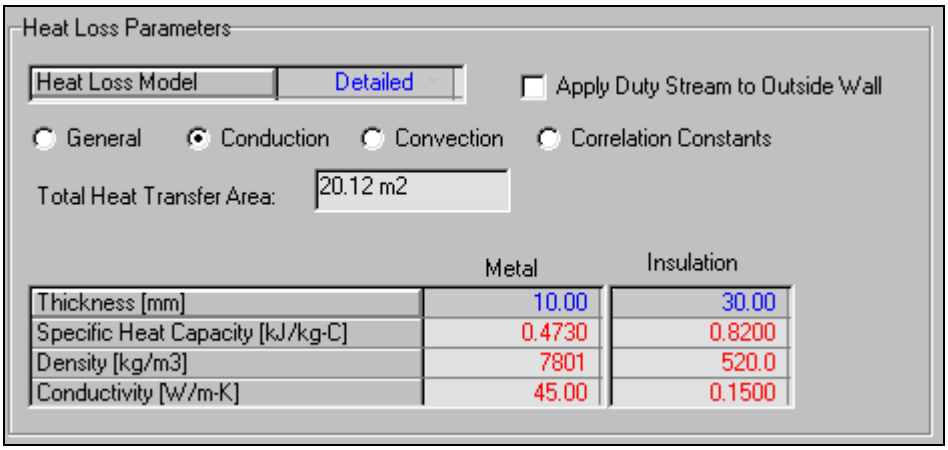


The default value for all three Recycle Efficiencies is 100%. This means that all material in the vessel has been flashed together and is in thermodynamic equilibrium. If the Recycle Efficiencies were to be reduced a portion of the material would by-pass the flash calculation and the vapour and liquid would no longer

instantaneously reach equilibrium. In this case, the phases may have different temperatures. Unfortunately, there is no single typical number suggested for these parameters. The best option would be to try various scenarios and observe the results.

Conduction

The Conduction parameters allow the user to manipulate the conductive properties of the wall and insulation.



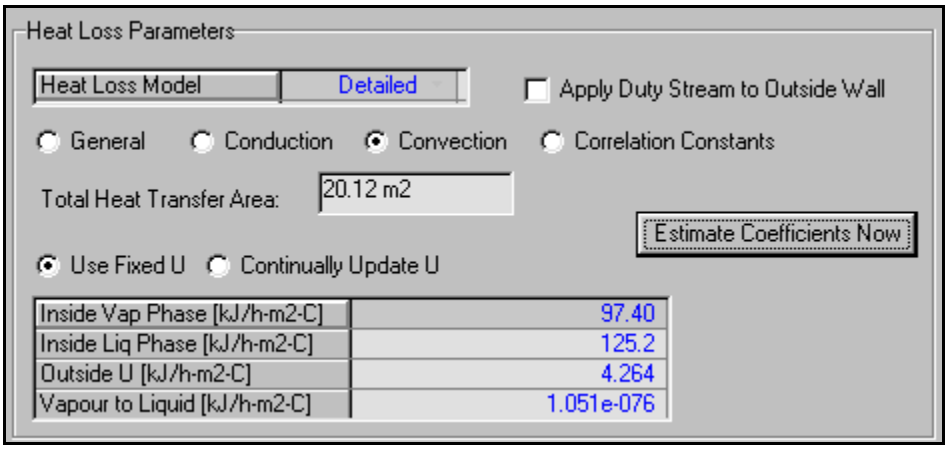
The metal wall thickness must always have a finite value (i.e.: it cannot be <empty>). To model a vessel without insulation, the insulation value thickness should be zero. Users are also required to enter the specific heat capacity of the material(s), the density of the material(s) and the conductivity of the material(s).

Some typical values for metals are:

Metal	**Density** kg/m^3	**Specific Heat** kJ/kg K	**Thermal Conductivity** W/m K
Mild steel	7860	0.420	63
Stainless steel	7930	0.510	150
Aluminium	2710	0.913	201
Titanium	4540	0.523	23
Copper	8930	0.385	385
Brass	8500	0.370	110

Convection

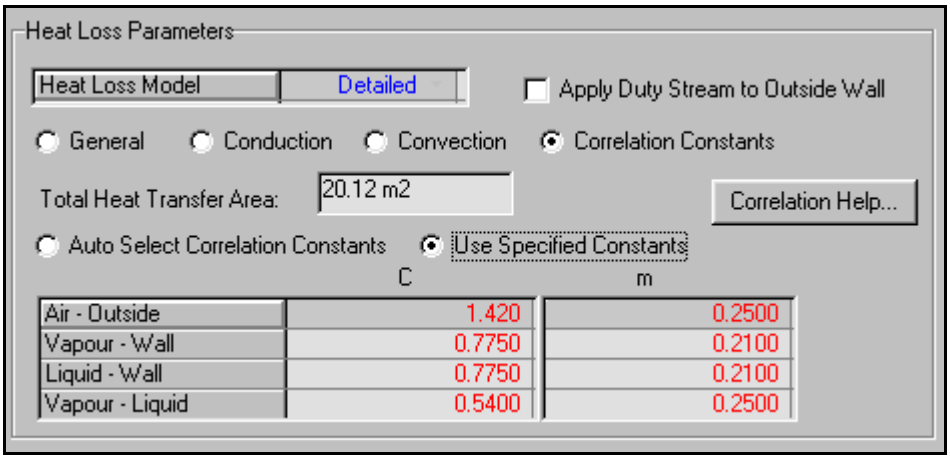
The Convection view allows users to manipulate the heat transfer coefficient for inside and outside the vessel as well as between vapour and liquid material inside the vessel.



To use a set of fixed U values, the "Use Fixed U" option should be selected. If the U values are unknown, the user can press the "Estimate Coefficients Now" button and have HYSYS determine the U values. In order to have HYSYS vary the U values throughout the depressuring scenario, select the "Continually Update U" value.

Correlation Coefficients

This feature gives users the opportunity to manipulate the coefficients used in the heat transfer correlation. By selecting "Use Specified Constants", the user may manually enter the constants used in the heat transfer correlations.



The equation which determines the outside heat transfer coefficient for air is:

$$h = C \times \left(\frac{\Delta T}{length} \right)^{m}$$

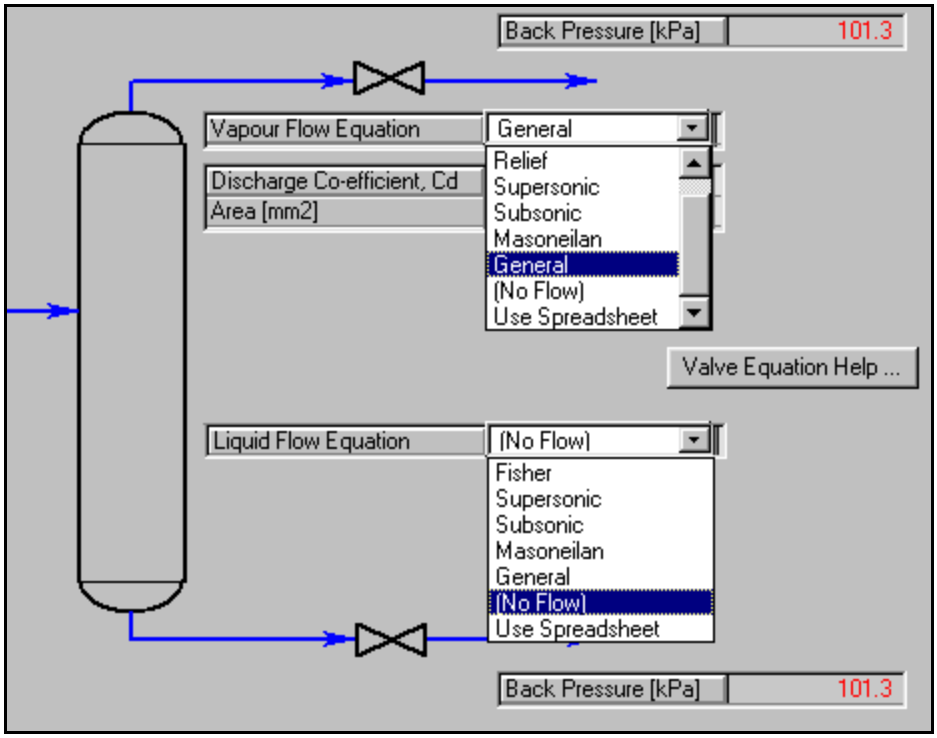
The equation used for the other three correlations is:

$$Nu = C \times (Gr \times Pr)^{m}$$

Where: Nu = Nusselt Number
Gr = Grashof Number
Pr = Prandtl Number

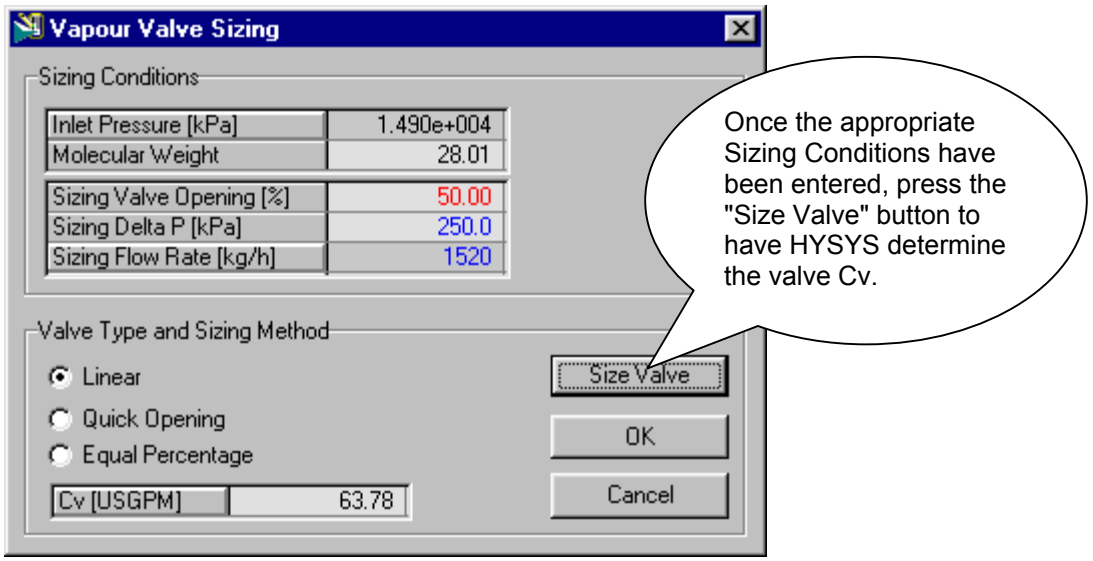
Valve Parameters

The Valve Parameters page allows users to select the type of valves to be used for both vapour and liquid service. In most cases, either the Fisher or the Relief valve should be used for valve sizing. Their equations are more advanced than some of the others and can automatically handle choked conditions. Furthermore, these two valve types support other options that can be accessed through the valve property view accessible through the Depressuring sub-flowsheet. The seven available valve types are described in the sections that follow.



Fisher

The Fisher option uses the standard valve option in HYSYS. It allows the user to specify both valve Cv and percent opening. By pressing the "Size Valve", the valve can be sized for a given flow rate.



Relief Valve

The relief valve option uses the standard HYSYS relief valve. The user can specify orifice area (or diameter), relief pressure and full open pressure. The user is required also to specify an orifice discharge coefficient. To have the relief valve open at all times, enter a full open pressure that is lower than the final expected vessel pressure and a set pressure that is only slightly lower than the full open pressure.

Vapour Flow Equation	Relief
Orifice Area [mm2]	2.500
Orifice Diameter [mm]	1.784
Discharge Coefficient	1.000
Set Pressure [kPa]	200.0
Full Open Pressure [kPa]	300.0

Supersonic

The supersonic valve equation can be used for modelling systems when no detailed information on the valve is available. The discharge coefficient (C_d) should be a value between 0 and 1. The area (A) should be a value between 0.7 and 1. P_1 refers to the upstream pressure and ρ_1 the density.

$$F = C_d \times A \times \left(P_1 \times \rho_1\right)^{0.5}$$

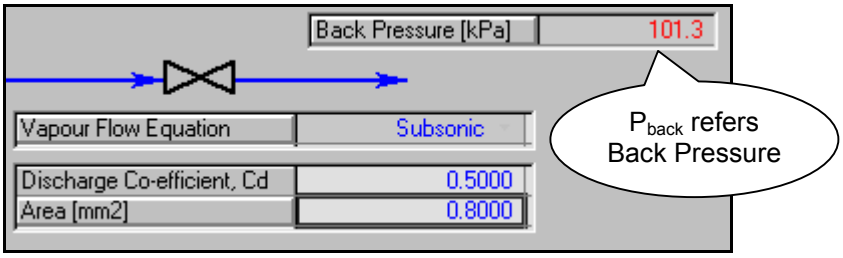
Vapour Flow Equation	Supersonic
Discharge Co-efficient, Cd	0.5000
Area [mm2]	0.8000

Subsonic

The subsonic valve equation can also be used for modelling systems when no detailed information on the valve is available but the flow is sub-critical. This can occur when the upstream pressure is less than

twice the backpressure. The discharge coefficient (C_d) should be a value between 0 and 1. The area (A) should be a value between 0.7 and 1. P_1 refers to the upstream pressure and ρ_1 the density.

$$F = C_d \times A \times \left(\frac{(P_1 + P_{back}) \times (P_1 - P_{back})}{P_1} \rho_1 \right)^{0.5}$$



It is possible to have the depressuring scenario cycle between pressure build-up and relief. To perform this analysis, ensure a reasonable pressure differential and increase the number of pressure steps.

Masoneilan

This equation was taken from the Masoneilan catalogue. It can be used for general depressuring valves to flare. When this option is selected, the user must specify C_v and C_f. The remaining parameters in the equation are set by the Depressuring utility.

$$F = C_1 \times C_v \times C_f \times Y_f \times (P_1 \times \rho_1)^{0.5}$$

where:

C_1	=	1.6663 (SI Units)
	=	38.86 (Field Units)
C_v	=	valve coefficient (often known from vendor data)
C_f	=	critical flow factor
Y_f	=	$y - 0.148y^3$
y	=	expansion factor
P_1	=	upstream pressure
ρ_1	=	upstream density

General

The General valve equation is based on the equation used to calculate critical flow through a nozzle as shown in *Perry's Chemical Engineers' Handbook* [1]. It should be used when the valve throat area is known. Note that this equation makes certain limiting assumptions concerning the characteristics of the orifice.

$$F = C_d \times A_v \times K_{term} \times (g_c \times P_1 \times \rho_1 \times k)^{0.5}$$

where:

C_d	=	discharge coefficient
A_v	=	throat cross sectional area
K_{term}	=	$\left(\frac{2}{k+1} \right)^{\frac{k+1}{2(k+1)}}$

k = ratio of specific heats (Cp/Cv)
P_1 = upstream pressure
ρ_1 = upstream density

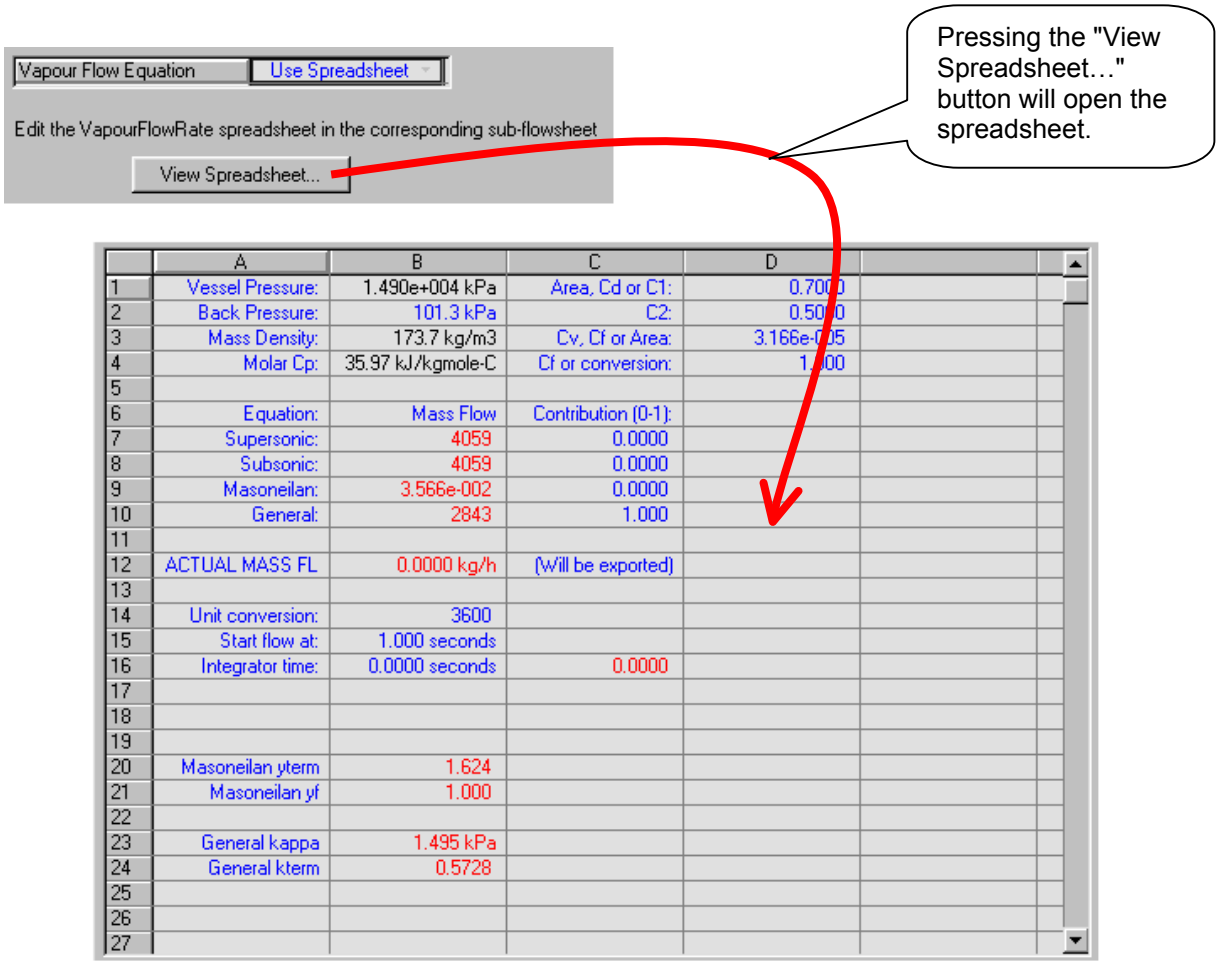
[1] Page 5-14, Equation 5.20 (6th Edition) & Page 10-15, Equation 10.26 (7th Edition)

No Flow

This option indicates that there is no flow through the valve.

Use Spreadsheet

Recommended only for advanced users, this option allows the user to customise a valve equation by editing the valve spreadsheet found inside the Depressuring sub-flowsheet.



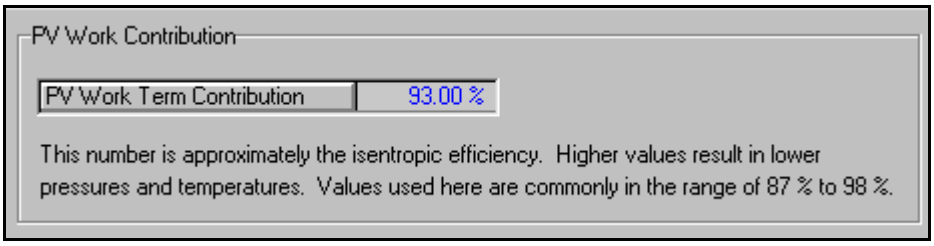
	A	B	C	D
1	Vessel Pressure:	1.490e+004 kPa	Area, Cd or C1:	0.700
2	Back Pressure:	101.3 kPa	C2:	0.500
3	Mass Density:	173.7 kg/m3	Cv, Cf or Area:	3.166e-005
4	Molar Cp:	35.97 kJ/kgmole-C	Cf or conversion:	1.000
5				
6	Equation:	Mass Flow	Contribution (0-1):	
7	Supersonic:	4059	0.0000	
8	Subsonic:	4059	0.0000	
9	Masoneilan:	3.566e-002	0.0000	
10	General:	2843	1.000	
11				
12	ACTUAL MASS FL	0.0000 kg/h	(Will be exported)	
13				
14	Unit conversion:	3600		
15	Start flow at:	1.000 seconds		
16	Integrator time:	0.0000 seconds	0.0000	
17				
18				
19				
20	Masoneilan yterm	1.624		
21	Masoneilan yf	1.000		
22				
23	General kappa	1.495 kPa		
24	General kterm	0.5728		
25				
26				
27				

Discharge Coefficient

When the relief, supersonic, subsonic or general valve is selected, the user is required to specify a discharge coefficient. This correction factor accounts for the vena contracta effect. Values ranging from 0.6 to 0.7 are typically used. In order to disregard this effect, set the discharge coefficient equal to 1.

Options

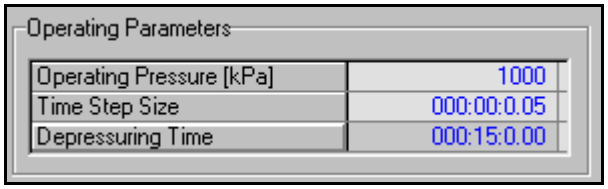
"PV Work Term Contribution" refers to the isentropic efficiency of the process. A reversible process should have a value of 100% and an isenthalpic process should have a value of 0%. For gas-filled systems, values range from 87% to 98%. For liquid filled systems the number ranges from 40% to 70%. A higher isentropic efficiency results in a lower final temperature.



Operating Conditions

Operating Parameters

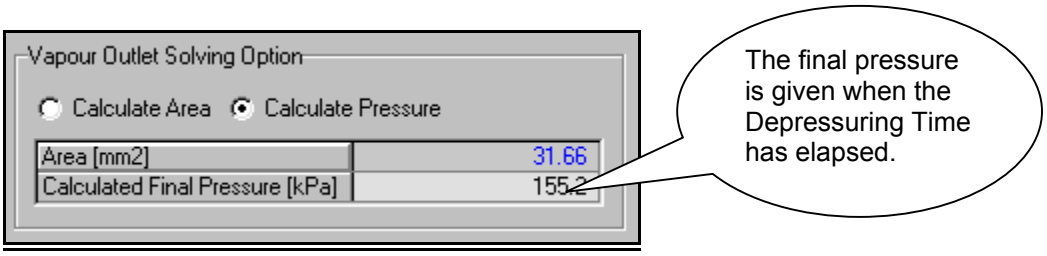
Operating pressure refers to the initial vessel pressure. By default, this value is the pressure of the inlet stream. The time step size refers to the integration step size. It may be a good idea to reduce the step size if the flow rate is significantly larger than the volume or if the vessel depressurises in a relatively short amount of time (~3s).



Vapour Outlet Solving Option

Either the Dynamic Depressuring utility can solve for the final pressure or the C_v/Area required to achieve a specified final pressure.

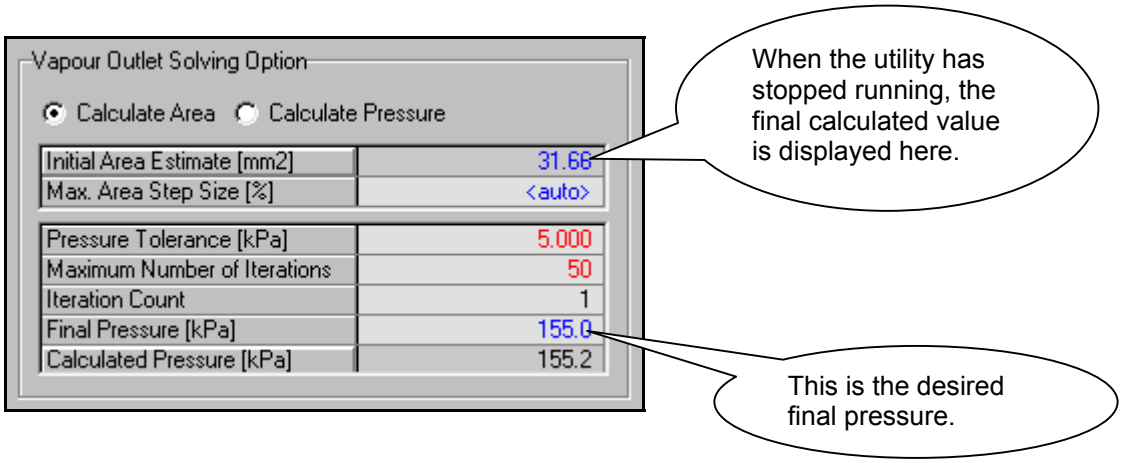
The "Calculate Pressure" option uses the specified area/Cv to determine the final pressure.



"Calculate Area" is available for Relief, Supersonic, Subsonic and General valves. "Calculate Cv" is available for Fisher and Masoneilan valves. The two options differ only in the type of value calculated.

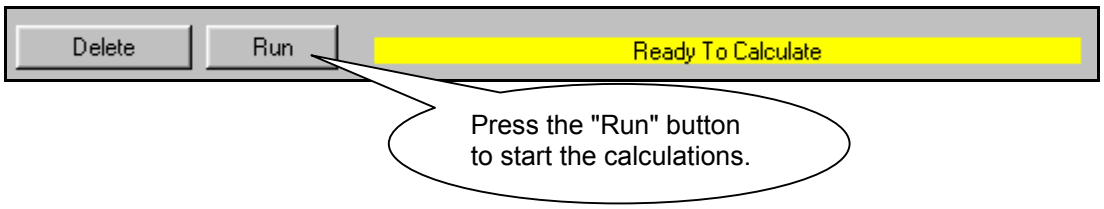
Based on API, it is normal to depressure to 50% of the staring pressure or to 100 psig. Before the calculations start, the user must specify an initial Cv or area. If the depressuring time is reached before

the final pressure is achieved, then the calculations stop and a new Cv or area is calculated using the final pressure. The calculations are repeated until the final pressure is reached in the given amount of depressuring time. The user may specify a maximum number of iterations and a pressure tolerance to improve convergence. If the user wishes to stop the calculations at any time, the <CTRL> <BREAK> keys can be used.

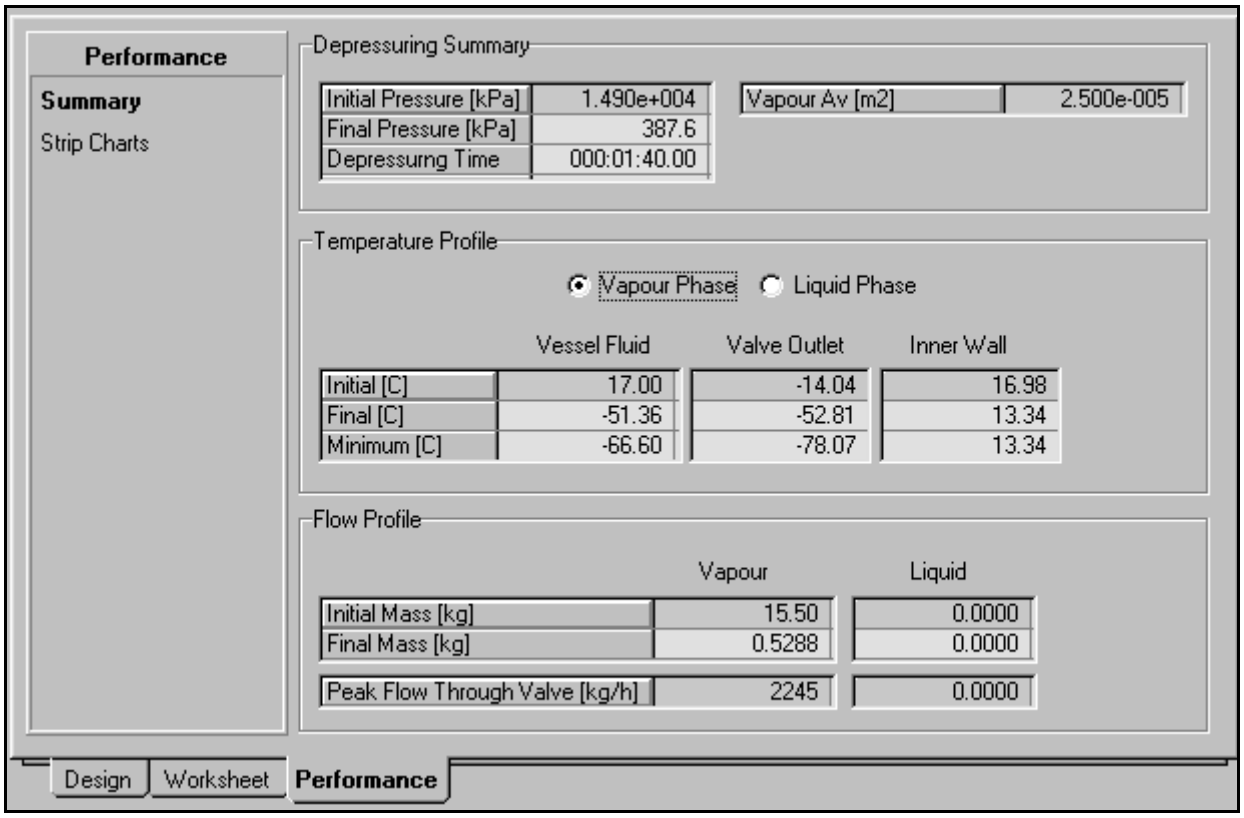


Performance

Once all the required information has been submitted, a yellow bar that reads "Ready To Calculate" will appear at the button of the Depressuring view.



Once the utility has run, users can go to the "Performance" → "Summary" page to view the results.



1.0 Example Problem

Simple Fire Depressuring

In the exercise, the required valve size for depressuring a vertical vessel to 50% of its operating pressure in a Fire Wetted case will be calculated.

Select the Peng-Robinson equation of state, add the required components and then add a stream with the following properties and molar flows:

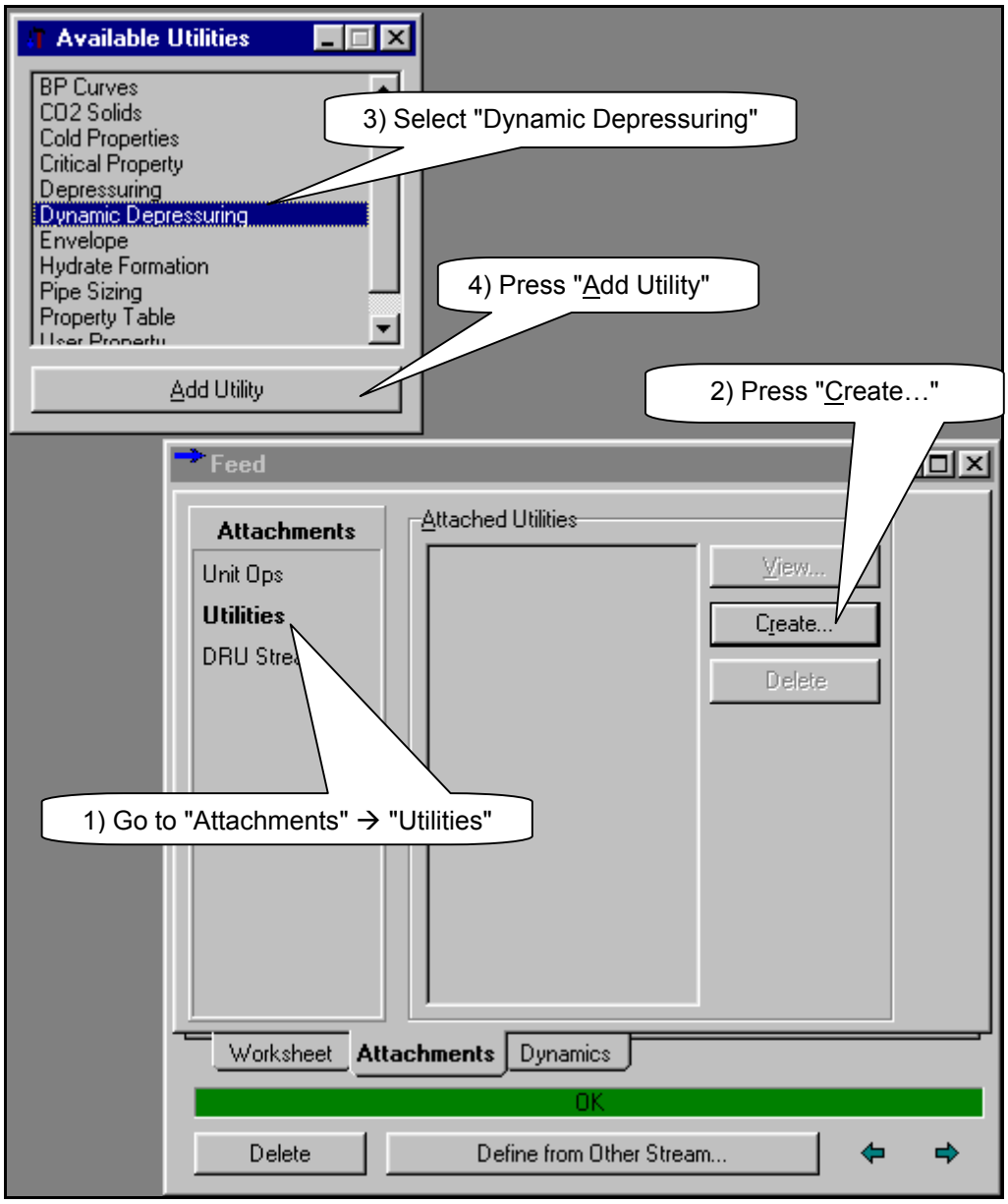
Stream Name	**Feed**	
Temperature	108 C	(226.4 F)
Pressure	1000 kPa	(145.04 psia)

Component	**Molar Flow**	
Methane	30.0 kmol/h	(66.138 lbmol/h)
Ethane	30.0 kmol/h	(66.138 lbmol/h)
Propane	30.0 kmol/h	(66.138 lbmol/h)
i-Butane	30.0 kmol/h	(66.138 lbmol/h)
n-Butane	30.0 kmol/h	(66.138 lbmol/h)
i-Pentane	30.0 kmol/h	(66.138 lbmol/h)
n-Pentane	325.0 kmol/h	(716.495 lbmol/h)
n-Hexane	30.0 kmol/h	(66.138 lbmol/h)

Stream Name	Feed
Vapour / Phase Fraction	0.70252
Temperature [C]	108.00
Pressure [bar]	10.000
Molar Flow [kgmole/h]	535.00

	Molar Flows
Methane	30.000
Ethane	30.000
Propane	30.000
i-Butane	30.000
n-Butane	30.000
i-Pentane	30.000
n-Pentane	325.00
n-Hexane	30.000

To attach the Dynamic Depressuring utility to the stream, open the stream property view, go to "Attachments" → "Utilities" and press "Create…". Select "Dynamic Depressuring" from the list of available utilities. Press the "Add Utility" button.

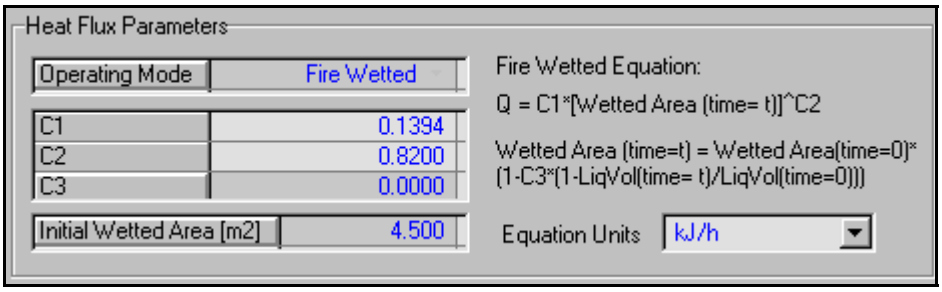


Enter the following vessel information on the "Design" → "Connections" page:

Variable Name	**SI Units**	**Field Units**
Height	4.50 m	14.76 ft
Diameter	1.25 m	4.101 ft
Initial Liquid Volume	1.45 m^3	51.21 ft^3

Enter the following information on the "Heat Flux Parameters" section of the "Heat Flux" page:

Variable Name	Value
Operating Mode	Fire Wetted
Equation Units	kJ/h
C1	0.1394
C2	0.8200
C3	0.0000
Initial Wetted Area	4.5 m^2 (48.44 ft^2)



Enter the following information on the "Valve Parameters" page:

Variable Name	Value
Vapour Flow Equation	Fisher
Cv	10 USGPM
% Opening	70%

On the "Options" page, enter a PV Work Term of **90%**. On the "Operating Conditions" page, select "**Calculate Cv**" and enter a final pressure of **500 kPa** (72.52 psia).

Once you have submitted the required information, press the "Run" button to execute the calculations. Explore the strip charts, analyse the results and answer the following questions:

What size valve was required to achieve the depressurisation? _____
What is the peak flow through the valve? _____ kg/h

Using the default values provided, try the "Simple" heat loss model.
What Cv is calculated? _____
What is the peak flow? _____ kg/h

Using the default values provided, try the "Detailed" heat loss model.
What Cv is calculated? _____
What is peak flow? _____ kg/h

Compressor and Pump Curves

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
09 Compressor and Pump Curves

Workshop

In this module, compressor and pump curves will be used to model the behaviour of simulated compressors and pumps. Using curves to model these unit operations allows HYSYS to accurately simulate actual plant equipment.

Learning Objectives

Once you have completed this module, you will be able to:

- Specify and attach head and efficiency curves to compressors
- Use single and multiple curves to model compressors
- Attach head curves to pumps
- Accurately model existing plant equipment with HYSYS

Prerequisites

Before beginning this module, you should have a reasonable understanding of the HYSYS program, and be familiar with adding and basic configuration of Pumps and Compressors.



Don't worry if you haven't built the case mentioned.

The "ADV6_AdvancedRecycles_Soln.hsc" file contains this case.

Compressor Curves

Using compressor curves in your HYSYS simulation allows you to accurately model existing plant equipment. You can determine if an existing compressor is able to meet the specifications of your process. Using compressor curves allows HYSYS to calculate heads and efficiencies that are dependant on the flow rate. If the flow rate through the compressor is known to be constant, a single pressure rise and efficiency can be supplied. If, however, the flow rate is expected to change, using a compressor curve will allow HYSYS to calculate new heads and efficiencies based on the current flow rate.

This results in greater accuracy in the simulation, and allows HYSYS to more closely model actual plant equipment.

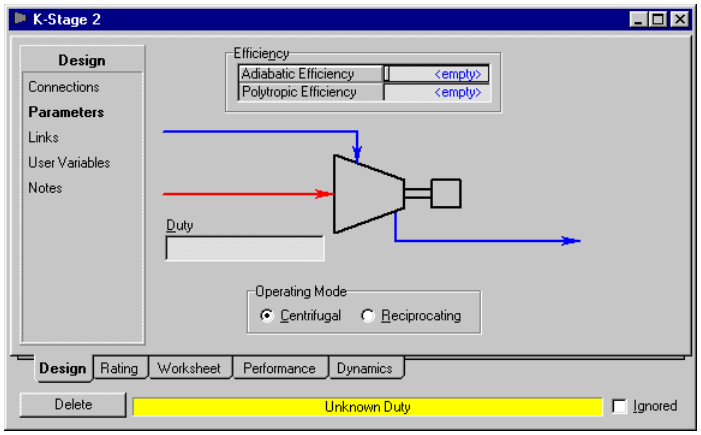
Workshop

In this workshop, you will add a set of multiple curves to the K-stage 2 compressor in the Advanced Recycle Module simulation.

1. On the **Parameters** page, ensure that the Polytropic and Adiabatic Efficiency boxes both read **<empty>**.

These values must read <empty> because the efficiencies will be calculated from the compressor curves, and defining the same value in two places will always result in a consistency error.

Figure 1

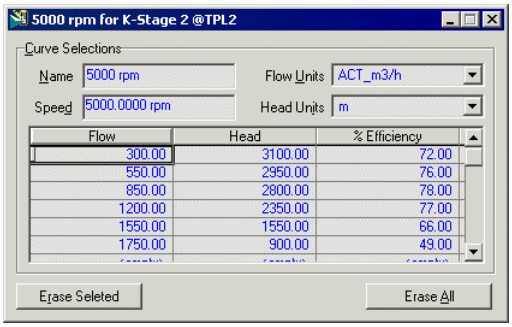


2. Ignore the **Adjust** that controls the outlet pressure of the 1st stage (ADJ-1).
3. On the **Curves** page (on the **Rating** tab), select the **Adiabatic** radio button in the **Efficiency** group. Click the **Add Curve** button, and enter the data as shown here:



Make sure you use the correct units for the variables, and that you set the units **before** entering the curve data.

Figure 2



Instead of manually typing the data you can paste it in from the Excel file 'Compressor Data.xls'. Before the compressor curve window will accept a table of pasted data it must first be set to have the correct number of rows. This can be accomplished by typing dummy data points into the left-hand column to give the required number of rows.

Figure 3

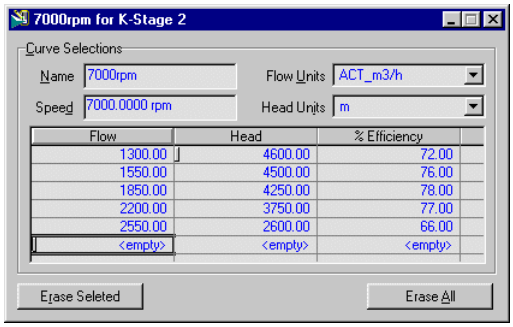


Figure 4

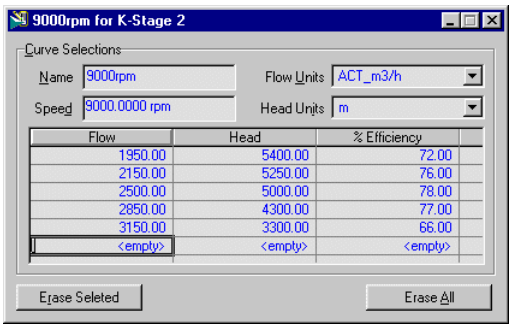
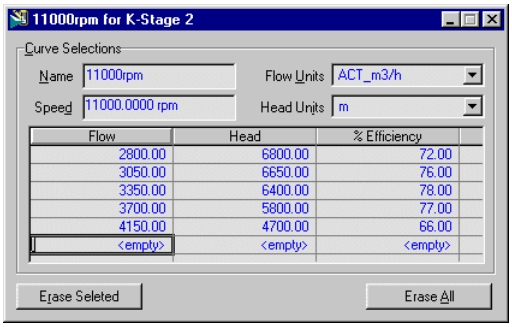


Figure 5



4. Activate the individual curves on the **Curves** page and ensure that the **Enable Curves** box is checked.

The pressure downstream of E-Stage 2 is fixed at 70 bar, so in this case the calculated variable on the compressor is the speed. As an alternative the speed could be specified and the downstream pressure calculated.



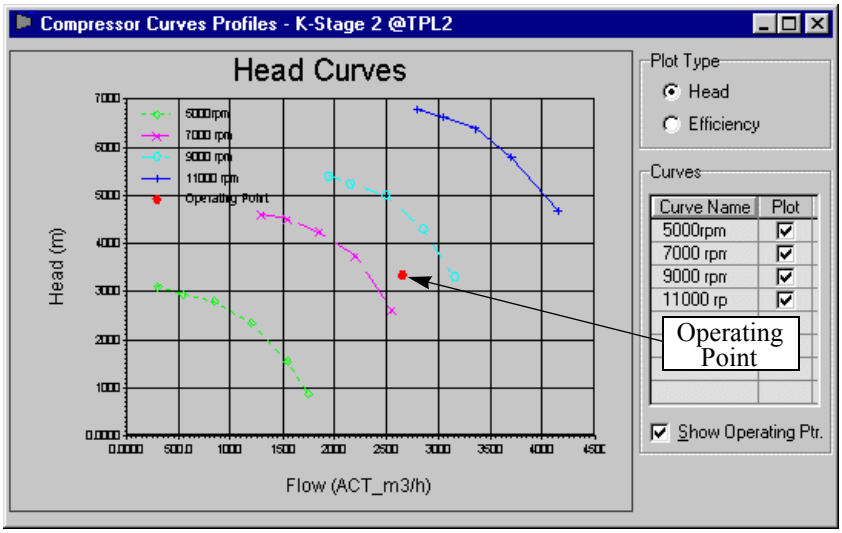
What is the speed of the machine?

What is the Adiabatic Efficiency?

and the Polytropic Efficiency?

5. Click the **Plot Curves** button. A graphical view of the curves and operating point is shown.

Figure 6



Save your case!

Optional Exercise

It is desired to have an outlet pressure of 7850 kPa at the battery limits to the facility (i.e. stream New Export Gas). How can you achieve this?

In a normal compressor installation, how would the unit be controlled? What would be set? What would be adjusted?

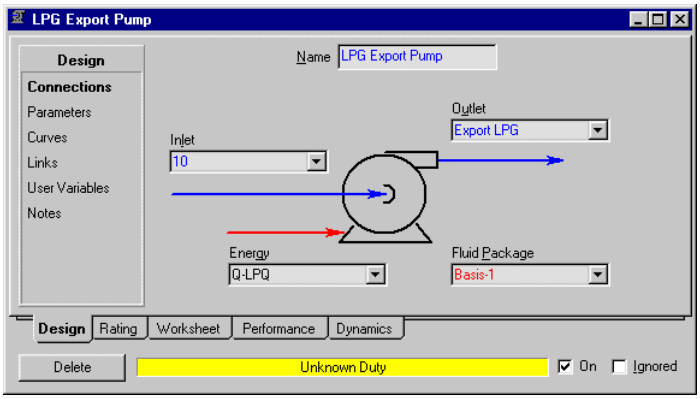
Can you think of a way to simulate this on the present flowsheet?

Pump Curves

As with compressor curves, pump curves are used to allow HYSYS to accurately model existing pumps. Pump curves allow the pressure rise across the pump to be dependent on the flow rate of liquid. The pump curves are entered into HYSYS using a form different to that used for compressor curves. With pump curves the coefficients of an expression, up to the fifth order, are entered into HYSYS rather than the actual data points.

Add a pump to the main flowsheet with the following information:

Figure 7

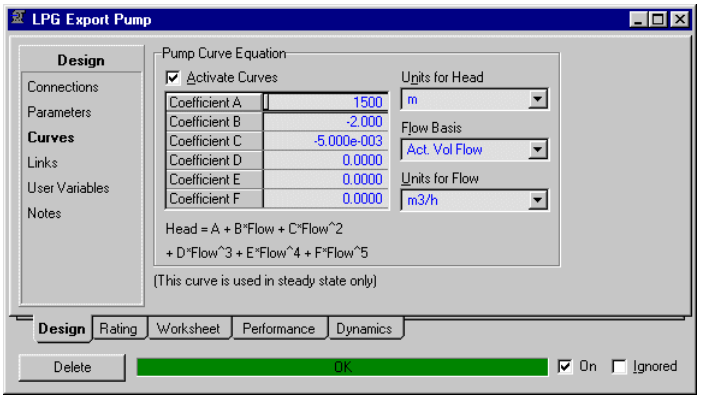




The coefficients can be obtained from a spreadsheet program capable of nonlinear regression, such as EXCEL or may be supplied by the pump's manufacturer.

6. Ensure the **Activate Curves** box is ticked.

Figure 8





What is the outlet pressure of the pump?

Using Neural Networks in HYSYS

© 2004 AspenTech. All Rights Reserved.

Introduction

HYSYS includes a Neural Network calculation tool that can be used to approximate part (or all) of a HYSYS model. It can be trained to replace either the first principles calculations usually done by HYSYS, or to simulate a unit operation that cannot be modeled using first principles.

Using a Neural Network solver offers a number of advantages:

- It is significantly faster than a first principles solution.
- It offers increased robustness so that a result will always be possible.

When using a Neural Network, always be aware that results are valid only within the range over which the Neural network was trained.

Workshop

In this module HYSYS' Neural Network capability will be used to replace the standard HYSYS solver for the Turbo Expander plant that has been constructed in this course.

Additionally an Exercise is included where the Parametric Unit Operation is trained with tabular input data.

Learning Objectives

After completion of this module, you will be able to:

- Use the Parametric Utility to incorporate a Neural Network into a HYSYS model.
- Use the Parametric Unit Operation with tabular data to model a unit operation as a 'black box'.

Prerequisites

Before starting this module you should be familiar with the HYSYS interface and be able to add and configure streams, operations, utilities and case studies.

Neural Networks

What is a Neural Network?

A Neural Network (strictly an 'Artificial Neural Network' as opposed to a 'Biological Neural Network') is a mathematical system with a structure based on that of the brains of mammals. The Artificial Neural Network is split into many basic elements (equivalent to neurons in biological systems), which are linked by synapses.

Neural Networks model the relationship between input and output data. They are particularly suited to the kind of problems that are too complex for traditional algorithm based modeling techniques, for example pattern recognition and data forecasting. There are a number of types of Neural Networks, but HYSYS uses a Multi-Layer Perceptron (MLP) type model.

The Neural Network is trained through a learning process where synaptic connections between neurons are constructed and weighted. The Neural Network is trained in an iterative manner. A set of input data and desired output data is repeatedly supplied and based on the errors between the Neural Network calculated outputs and the desired outputs, the connections are adjusted for the next iteration.

Neural Networks in HYSYS

The HYSYS Neural Network implementation allows part (or all) of the HYSYS flowsheet to be approximated by a Neural Network solver.

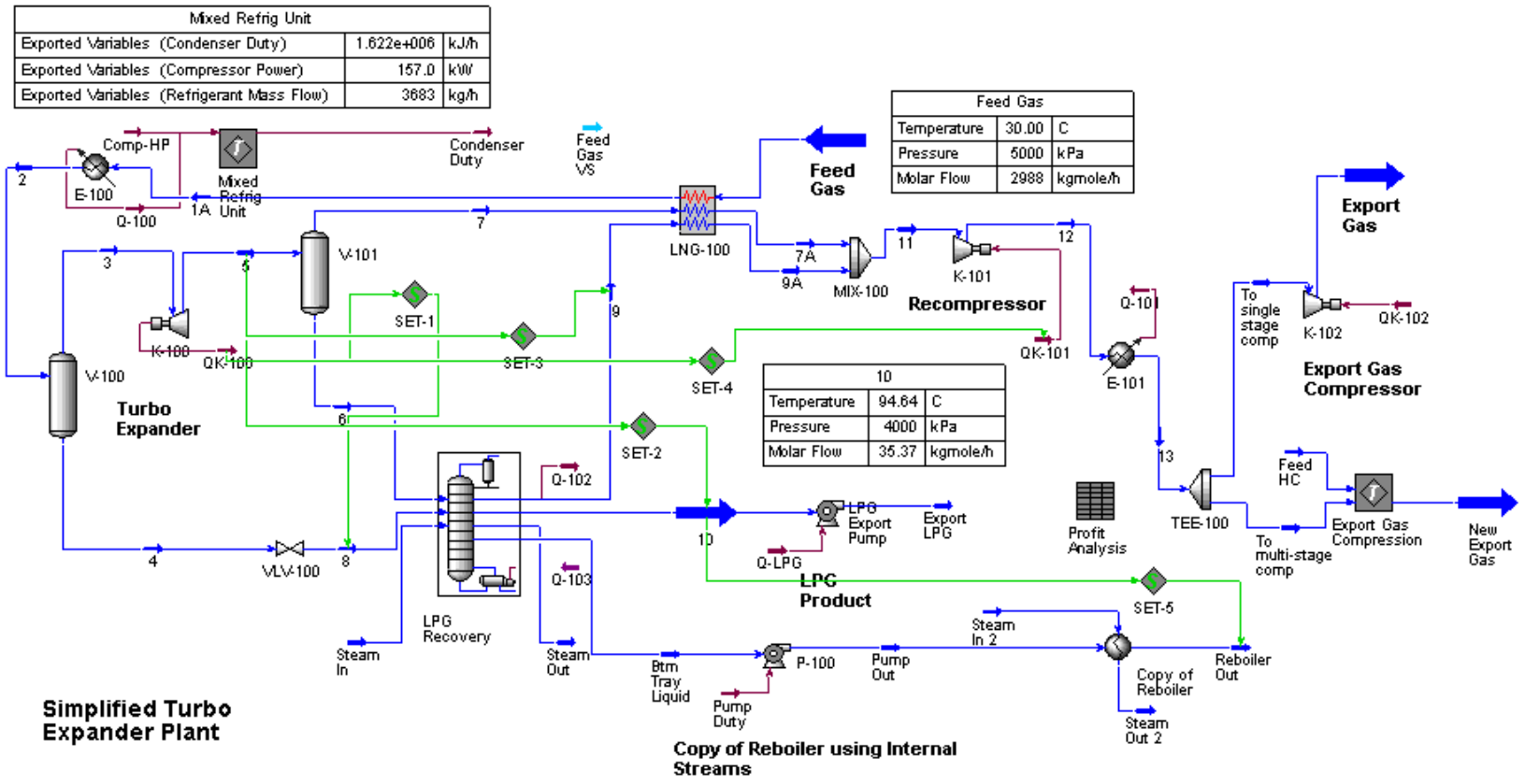
The Neural Network can either be trained with the results from the standard (first principles) solver, or can be supplied with tabular training data. In this way, it can be used as a 'black box' calculation engine based on experimental or plant data.

There are two parts to the HYSYS Neural Network implementation:

Parametric Utility. This is where the Neural Network is configured, and trained.

Parametric Unit Operation (Optional). This allows the Neural Network to appear as a unit operation on the flowsheet, and is typically used when taking a 'black box' approach.

Process Overview



Steps for using Neural Networks in HYSYS

The procedure for using Neural Networks in HYSYS is as follows:

1. Select scope: determine which streams/operations will be calculated by the Neural Network.
2. Select and configure input and output variables.
3. Supply training data: either tabular data or data generated by the normal HYSYS solver.
4. Train the Neural Network.
5. Validate the Neural Network. This is optional, but recommended.

Workshop

Process Description

In this module HYSYS' Neural Network capability will be used to replace the standard HYSYS solver for the Turbo Expander plant that has been constructed in this course.



Don't worry if you haven't built the Turbo Expander plant case.

The file 'ADV5_Spreads&CaseStud_Soln.hsc' contains this case.

1. Open the Turbo Expander plant case if it is not already open.

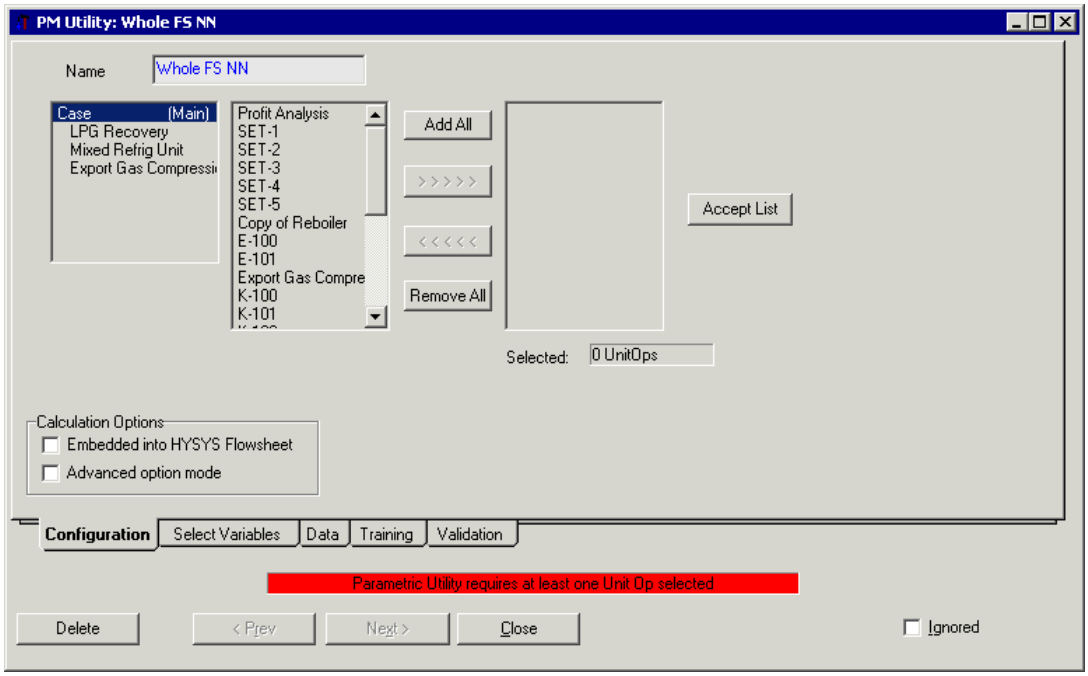
   This module assumes that the case has had at least the changes from the 'Templates and Sub-flowsheets' and 'Spreadsheets and Case Studies' modules made to it.

The main process variables that will be manipulated are the cooler outlet temperature (stream 2) and the Turbo Expander outlet pressure (stream 5).

If you have completed the Advanced Recycles module and have added the multi-stage compression sub flowsheet to your Turbo Expander plant, it is a good idea to ignore the Adjust operations to reduce the calculation time.

Adding the Parametric Utility

2. From the **Tools-Utilities** menu, add a **Parametric Utility**. Name the utility 'Whole FS NN'.



Setting the Scope

The first step in configuring the Parametric utility is to select the scope (i.e., how much of the flowsheet will be calculated using the Neural Network). In this case, the Neural Network will be applied to the whole flowsheet.

3. On the **Configuration** tab, ensure Case (Main) is selected in the left list box. Click the **Add All** button.
4. Click **Accept List**.

   Notice that now the **Next>** button is enabled to move the view to the next tab.

It is possible to only model a subset of operations in the flowsheet. Operations can be added and removed using the buttons marked >>>>> and <<<<<.

Selecting and Configuring Variables



A Validation tool is included to check the quality of the Neural Network calculations.

The variables that the Neural Network will use must now be configured. There are two important classes of variables:

Manipulated

Observable

The Neural Network solver will respond to changes in the Manipulated variables and calculate new values for the Observable variables based on the supplied training data.

The quality of the Observable values calculated by the Neural Network solver is dependent on the quality of the data used to train it. A Neural Network model is only as good as its training data. Going outside the range of the Manipulated variables used for training can lead to large errors.

In the Turbo Expander case the Manipulated variables are the temperature of stream 2 and the pressure of stream 5, while the Observable variables are the properties of all the streams in the flowsheet.



The Name column can be expanded by clicking and dragging between the two columns in the header.

5. On the **Select Variables** tab, generate a list of all possible Manipulated and Observable variables by clicking the **Build Flashable Streams** button.
6. With the Manipulated radio button selected, click the **Un-Select All** button.
7. In the **Selected Mvar** column check the items:
   - 5\Pressure
   - 2\Temperature
8. Click the **Remove Unselected** button to display only these two variables.



Changing the Range parameter above the table sets all the Low and High values to a given fraction above and below the Initial (Current) value.

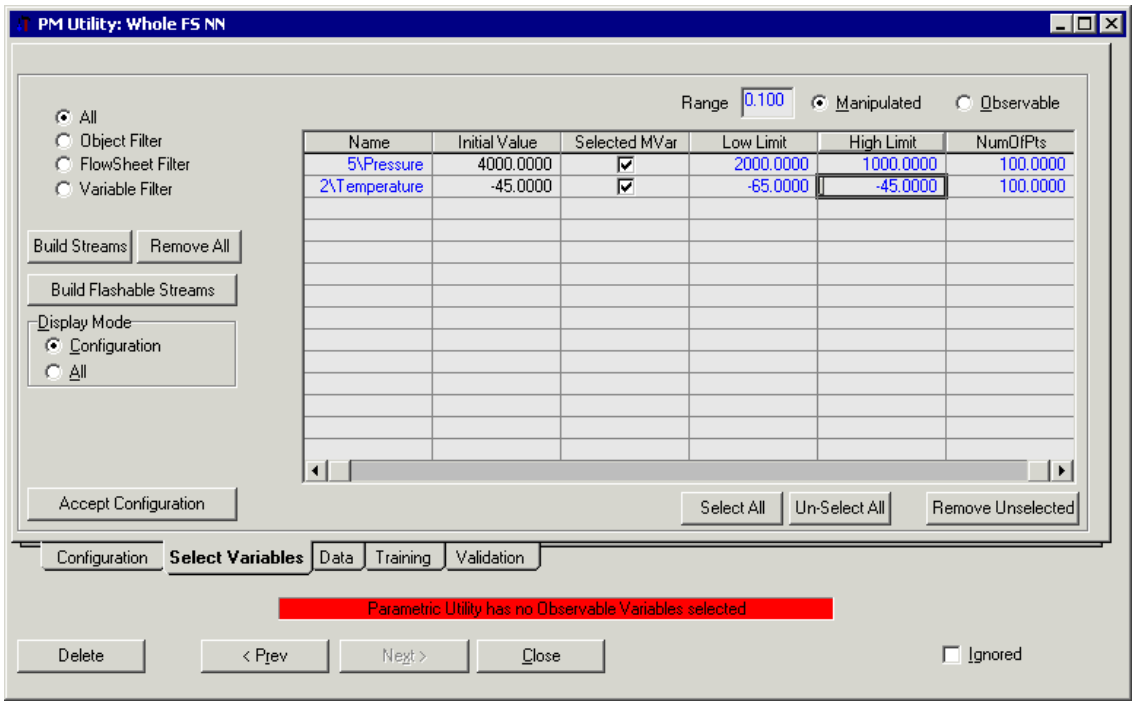
Now you need to set the range of manipulated variables for training the Neural Network.

9. Change the Low and High limits as follows:

5\Pressure	20 to 40 bar
2\Temperature	-65 to -45 °C

10. Click the **Accept Configuration** button.

The utility should now appear as follows:



11. Choose the **Observable** radio button and review the variables that will be calculated by the Neural Network.



When supplying training data, it is important to provide a good representation of the region in which the Neural Network will be operating.

By default the neural network output files go in the \Support subdirectory of the HYSYS installation. If required specify a different directory name.

Generating a Training Dataset

Now data must be generated to train the Neural Network. This involves supplying a set of values for each of the Manipulated variables, then running HYSYS to calculate the values of the observable variables for each of these sets.

Values for the Manipulated variables can either be supplied manually, read from a *.csv file, or may be generated using the Build Random dataset tool.

12. On the **Data** tab ensure the **Create as New** option is selected and supply the Output File Head Name ‘TurboExpander’.
13. Set the **Size** of the **Manipulated Data Set** to 32. This will give the Neural Network more data to train from. Often 8 is too low for accurate results.
14. Click the **Build Random Dataset** button to populate the table with training data.
15. Click the **Generate Data** button. HYSYS will run and solve for each of the datasets supplied and generate all the resulting training data.

If HYSYS displays any column errors or messages about empty values in the dataset simply OK them. HYSYS will offer to remove any empty training data before training the Neural Network.

For more complicated systems, the generation of training data can take a significant length of time. In this case, it should take less than a minute depending on computer speed.



The Init/Rest button should be used before the Neural Network is trained for the first time and whenever it needs to be retrained.

Training the Neural Network

The next step is to train the Neural Network using the training dataset just generated.

16. Select the **Training** tab and click the **Init/Reset** button.

    If prompted, choose the option to remove empty values from the dataset.

17. Click the **Train** button to train the Neural Network with the data generated.

In this case, the training process should only take a few seconds. When it has completed, you can view a comparison between the output of the parametric utility and the calculations from HYSYS by using the **View Table** and **View Graph** buttons and choosing the **Output** radio button.

Validating the Neural Network Results

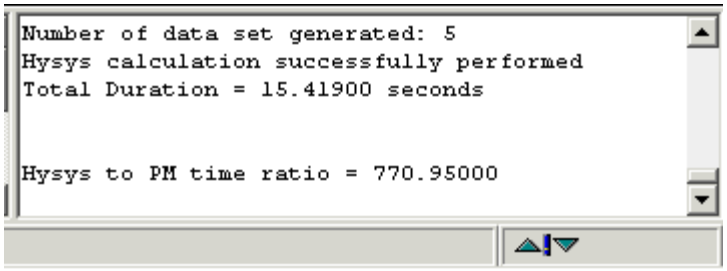


Validation is optional but recommended.

The final step before using the Neural Network is to validate the results. In the validation process, a new set of input data is given to both the HYSYS model and the Neural Network and the results are compared.

18. Select the **Validation** tab. Click the **Validation Setup** button to configure the validation runs. Select **OK** to accept the defaults.
19. Click the **PM Runs** buttons to run the Parametric model (i.e., Neural Network). This runs quickly so it may seem that nothing happened. But if you look at the Trace window (the bottom right white panel), it shows that the PM calculation was successful.
20. Click the **HYSYS Runs** button to run the traditional HYSYS model with the validation input.

The Trace window displays a comparison of the time taken by the Parametric utility and the standard HYSYS solver.



21. By clicking the **View Graph** or **View Table** buttons, the results from the HYSYS model can be compared to those from the Neural Network model.

In this case, the error should be negligible for all of the variables.

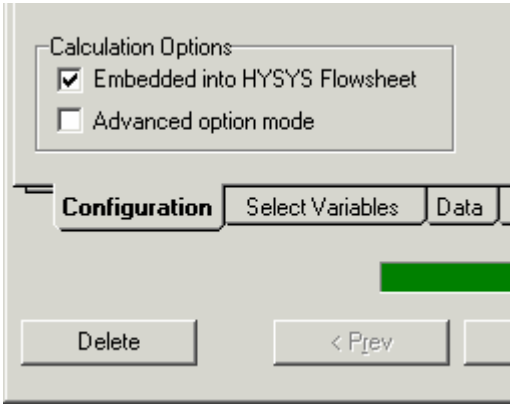


If the Build Streams button was clicked instead of the Build Flashable Streams button, then at this point HYSYS will display warning messages as it removes all observed variables that would lead to an over specification.

Embed the Neural Network into the HYSYS Flowsheet

Now the Parametric utility is ready to use to replace the main HYSYS solver.

22. Return to the **Configuration** tab and check the **Embedded into HYSYS Flowsheet** checkbox.



A Trace window message ('Using Whole FS NN for calculation') will appear. HYSYS is now using the Neural Network instead of the normal HYSYS solver.

Experiment with the Model



Find the case study on the Case Studies tab of the Databook (Tools-Databook menu).

To compare the speed of the Neural Network with that of the standard solver a Case Study will be used. Use the same Case Study that was set up in the Spreadsheets and Case Studies module (called 'Operating Analysis'). This varies the pressure and temperature over the same range as the Neural Network is trained for, and records the value of the Overall Profit from the spreadsheet.

1. With the Neural Network activated, start the Case Study. Keep track of how long it takes to run.
2. Switch the Neural Network solver off using the **Embedded into HYSYS Flowsheet** checkbox, and rerun the case study.



How much faster is the Neural Network solver in this case?

_____

(Typically the Neural Network takes a 1/10th the time of the standard solver for this model.)

Other Possible Investigations

Try changing one of the manipulated variables outside the training range. What happens?

If the Neural Network is switched on, what happens when a variable which is not a manipulated variable is changed? For example, change the temperature or composition of the Feed Gas stream.

With the Neural Network switched on, try setting an unfeasible value in one of the streams (for example, set 55 bar for stream 5's pressure). Compare the response of the model when the Neural Network is enabled and disabled.

Conclusions

Neural Networks can be significantly faster than a first principles solution. The Neural Network part of the calculation is typically about 1000 times faster than the standard solver, however HYSYS needs to do many other tasks as well (data storage, interface updates, etc.) that can reduce the actual speed increase seen.

Robustness is increased; a result will always be possible. Whereas the standard solver may fail in certain circumstances.

Neural Networks are only as good as the data they were trained with. If a parameter is changed so that it is outside the training range then the results may not be valid, and could include large errors.

Neural Networks will not predict the effect of changes in variables not included in the training data.

Exercise

Using the Parametric Unit Operation



It is also possible to link the Parametric Unit Operation to a Parametric Utility.

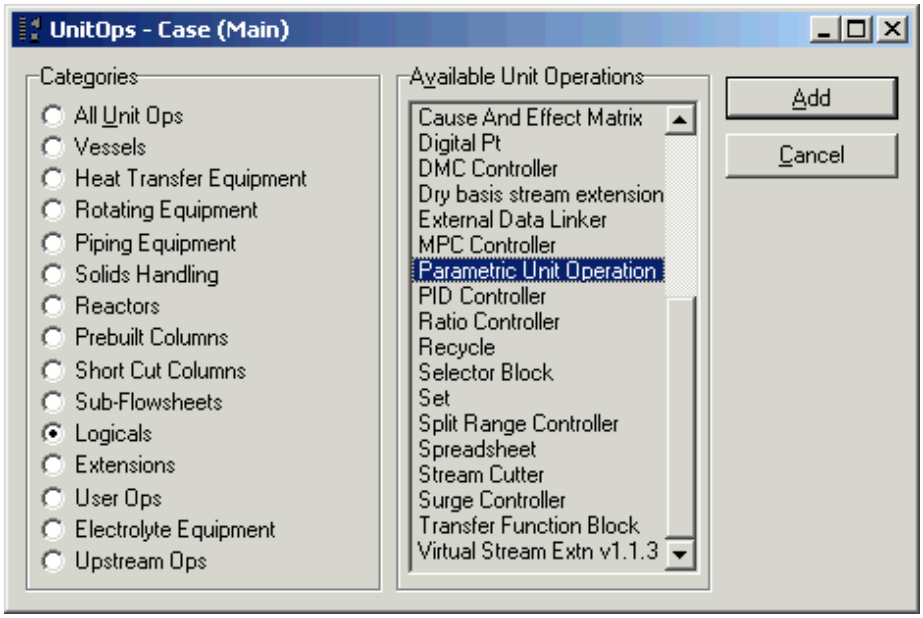
The Parametric Unit Operation allows the Neural Network to appear as a unit operation on the flowsheet, and is typically used when taking a 'black box' approach to modeling an operation. In this case the Neural Network can be trained with tabular data from lab experiments or plant measurements, so a system can be represented that may not necessarily be able to be modeled using a first principles approach.

In this exercise, a Parametric Unit Operation will be used to model an operation based on supplied tabular data.

1. Open the supplied HYSYS case **Parametric Unit Op Starter.hsc**.

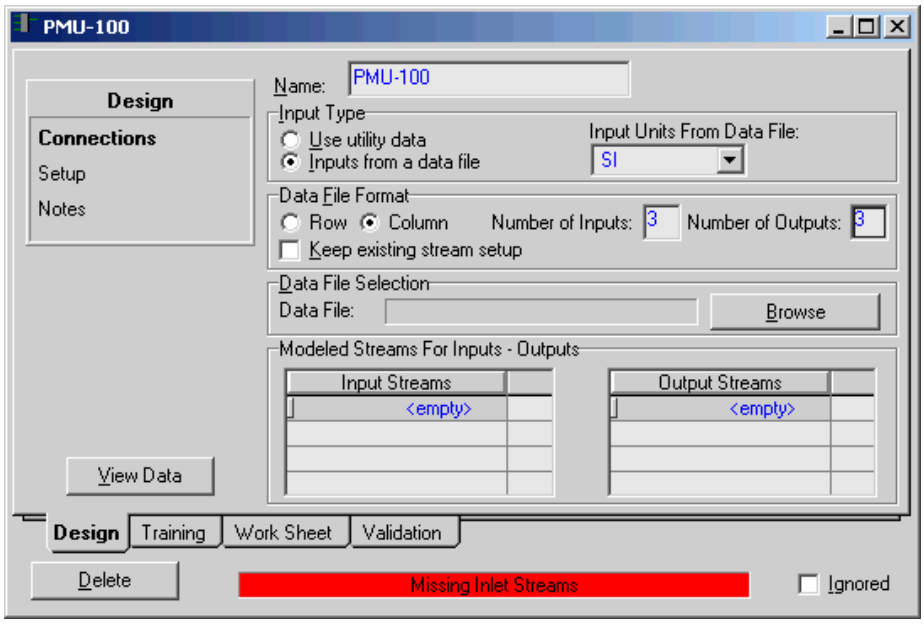
The shortcut key for the Flowsheet-Add Operation menu is F12.

2. Add a Parametric Unit Operation. (The Parametric unit operation does not appear on the object palette so it must be added using the **Flowsheet-Add Operation** menu). The Parametric Unit Operation is in the **Logicals** category.



3. Click **Add**.

4. Select the **Inputs from a data file** option and choose the **Column** data format option. Set the **Number of Inputs** to 3 and the **Number of Outputs** to 3 as shown.



5. Click the **Browse** button to navigate to the **Parametric Unit Op Data.csv** data file and select it. The file filter needs to be changed to show csv files. (Ignore the warning message about the lack of attached streams.)
6. Attach the **Fuel** and **Exhaust** streams as **Input** and **Output** respectively.



*Note that C, kPa and kg/h are the units used in the SI unit set, as selected in the **Input Units From Data File** drop-down list.*

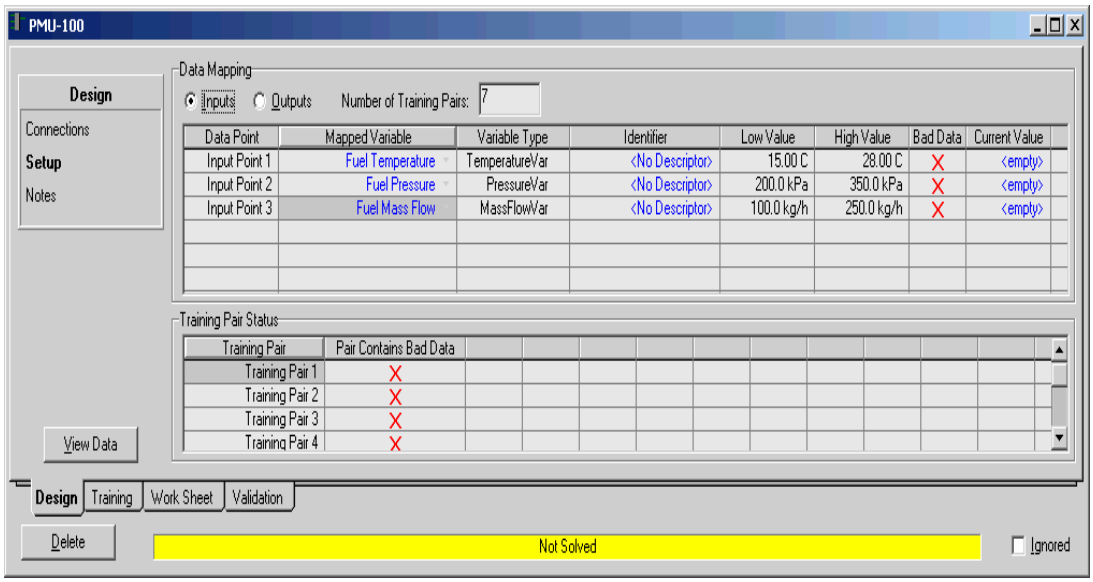
The 'Parametric Unit Op Data.csv' data file contains the following data (in a comma separated value format). The table below shows the variables that are being read by HYSYS.

Input Temperature (C)	Input Pressure (kPa)	Input Mass Flow (kg/h)	Output Temperature (C)	Output Pressure (kPa)	Output Mass Flow (kg/h)
15	200	100	20	175	100
18	225	125	23	190	125
20	250	150	27	210	150
22	275	175	32	225	175
24	300	200	37	240	200
26	325	225	45	250	225
28	350	250	52	260	250

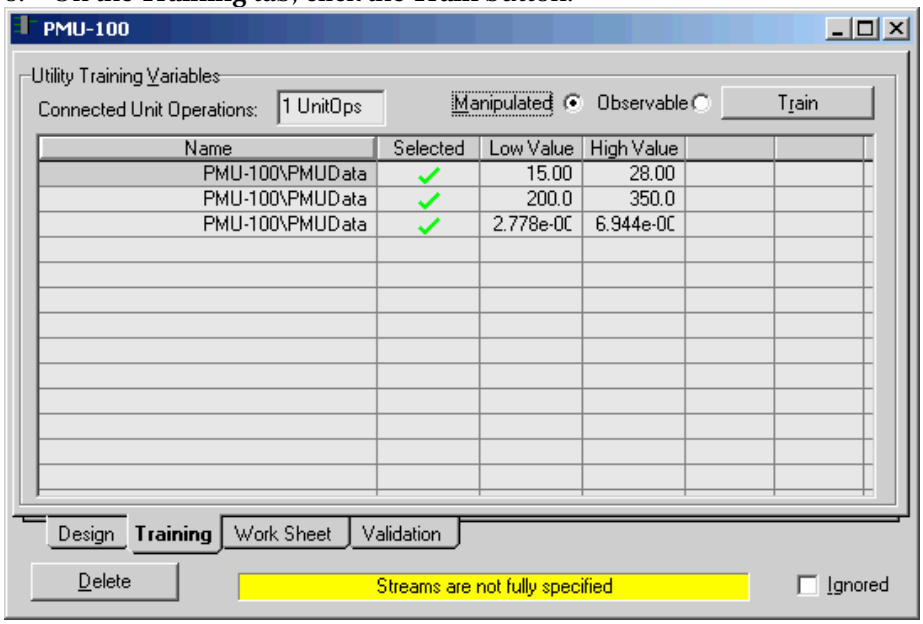
By clicking the **View Data** button, the contents of the data file can be displayed.

7. On the **Setup** page, map the Input 1, 2, 3 variables to the Fuel Temperature, Pressure and Mass Flow respectively. Similarly, map the Output Variables to Exhaust Temperature, Pressure, and Mass Flow.

The red cross in the Bad Data column means the data is OK. If the data is bad, a green checkmark appears.



8. On the **Training** tab, click the **Train** button.



9. Go to the **WorkSheet** tab and specify the **Fuel** stream as follows:

Temperature	35 °C
Pressure	300 kPa
Mass flow rate	200 kg/hr

The unit operation should now solve fully.

10. Experiment with changing the Temperature, Pressure and flow both inside and outside the range of the training data.

Modeling Real Separators in HYSYS

© 2004 AspenTech. All Rights Reserved.

Introduction

The HYSYS Separator unit operation normally assumes perfect phase separation, but it can also be configured to model imperfect separation by using the HYSYS Real Separator capabilities.

The real separator offers the user a number of advantages:

- Includes carryover so that your model matches your process mass balance or separator design specifications.
- Predicts the effect of exit devices on mitigating carryover.

This workshop will introduce the user to the concepts needed to use these real separator features. The workshop will then step the user through a typical real separator application.

Workshop

The workshop will focus on using the HYSYS Real Separator capabilities to model imperfect separation in a 3-phase oil-water-gas separator.

An exercise is included where a demister pad is added to the model as a secondary separation device to reduce liquid carryover into the gas.

Additionally, a demonstration is given of the carryover feature in a dynamic model.

Learning Objectives

After completion of this module, you will be able to:

- Account for carryover in process design problems.
- Calculate carryover based on vessel geometry and inlet conditions using several basic correlations.
- Model an exit device to reduce carryover in the vapour product.
- Understand how carryover effects are accounted for in a dynamic model of a separator.

Prerequisites

Before starting this module you should be familiar with the HYSYS interface and be able to add and configure streams, operations, utilities, and case studies.

Modeling Separators

Real World Considerations

In real world separators, separation is not perfect: liquid can become entrained in the gas phase and each liquid phase may include entrained gas or entrained droplets of the other liquid phase.

Recent years have seen increasing use of vessel internals (e.g., mesh pads, vane packs, weirs) to reduce the carryover of entrained liquids or gases.

Real Separators in HYSYS

Carryover Option

As with many other unit operations, HYSYS allows you to increase the fidelity of your separator model to account for non-ideal effects. HYSYS 3.2 introduces Real Separator capabilities like the carryover option. This option can be used to model imperfect separation in both steady state and dynamic simulation. Gas and liquid carryover can be specified or calculated (three different correlations are available for this purpose).

Vessel Internals

Internals used to reduce carryover can be included in your separator model with some of the provided carryover correlations.

Internals used to reduce liquid carryover in the gas product are termed “exit devices”. Weirs are used to improve heavy liquid - light liquid separation in horizontal vessels.

Nozzle Calculations

Included with the carryover correlations are calculation methods for inlet and outlet nozzle pressure drop. Inlet and outlet devices can be included in these calculations. The user can also specify pressure drop if the carryover option is not in use.

Dynamic Models of Real Separators

The dynamic model of a separator must account for changing pressure and flow due to liquid levels, nozzle pressure drop, and heat effects. As such, vessel geometry, including internals and nozzle geometry, and heat loss parameters need to be specified. Modeling imperfect separation with the carryover option and a specifiable PV work term are also available. Level taps can also be set for monitoring the relative levels of the different liquid phases. All of these items can be set up via the Rating tab.

Limitations of the carryover option:

As droplet distribution is not a stream property, this information is not passed onto the product streams. While droplet distribution is not passed on, product streams containing carryover will contain multiple phases with the phase flow rates equal to that predicted by the carryover calculations.

Specifying Carryover

The HYSYS separator allows the user to directly specify what fraction of each of the feed phases is entrained in the other phases. Product-based specifications are also allowed. This gives you a simple method to match your material balance to your design assumptions or your real world separator.

Calculating Carryover & Related Properties

There are also three sets of correlations available to calculate phase dispersion and carryover. A detailed description of each method is given in the next section. All three follow the same basic calculation sequence:

1. Calculate the initial phase dispersion based on the inlet feed. All three methods assume the dispersion follows a Rossin Rammler distribution.
2. Calculate the carryover after the primary separation (gravity settling) of each phase in every other phase; specifically:

   Light Liquid entrained in Gas

   Heavy Liquid entrained in Gas

   Gas entrained in Light Liquid

   Gas entrained in Heavy Liquid

   Light Liquid entrained in Heavy Liquid

   Heavy Liquid entrained in Light Liquid
3. Based on the exit dispersion from step 2, calculate the affect of any installed secondary separation device (e.g., demister pad or vanes) on the liquid carryover into the vapour product. (This is not applicable to the Generic correlations.)

Correlation Details

Three different correlation models are provided: Generic, Horizontal Vessel and ProSeparator™.

Generic Correlations

The generic correlations should be used when your only criterion for separation is specifying a critical droplet size. Inlet phase dispersion is calculated using a generic method that ignores vessel geometry — the user specifies inlet splits and Rossin Rammler parameters and these are used to calculate the inlet dispersion. Carryover is calculated by assuming that all droplets smaller than a user-specified critical droplet size are carried over.

Horizontal Vessel Correlations

The Horizontal Vessel correlations are designed with the horizontal 3-phase Separator in mind. Inlet phase dispersion is calculated using inlet device efficiency (rather than specified splits) and user-supplied Rossin Rammler parameters. Primary separation is calculated based on settling velocities rather than critical drop size. Each phase has a residence time in the vessel. A droplet will be carried over if it does not travel far enough (back to its parent bulk phase) in the time allowed.

ProSeparator Correlations

The ProSeparator correlations are rigorous but are limited to calculating liquid carryover into gas. Both light liquid and heavy liquid entrainment are calculated, so 3-phase Separators are also supported, but no carryover calculations are done for the liquid phases. Inlet phase dispersion is calculated based on inlet flow conditions and inlet pipe size. (ProSeparator calculates its own Rossin Rammler parameters using this information.) Primary separation is based on critical droplet size; however, the critical droplet size is not user-specified but calculated using gas velocity through the vessel.

Exit Devices & Other Calculations

Secondary separations accomplished by exit devices (e.g., demisting pad) can be calculated by specifying a critical drop size (Horizontal Vessel) or through the use of device specific correlations (ProSeparator).

Inlet flow regime, Nozzle Pressure Drop, Exit Device Sizing can also be calculated using one of the various Horizontal Vessel correlations.

Rossin Rammler Parameters

Rossin Rammler distributions are defined by:

$$F = \exp(-d/dm)^{x})$$

Where:

F = fraction of droplets larger than d

dm is related to d95

x = RR index

d95 = 95% of droplets are smaller than this diameter for the specified dispersion

RR Index = exponent used in the RR equation (also known as the “spread parameter”)

Using Sub-calculations

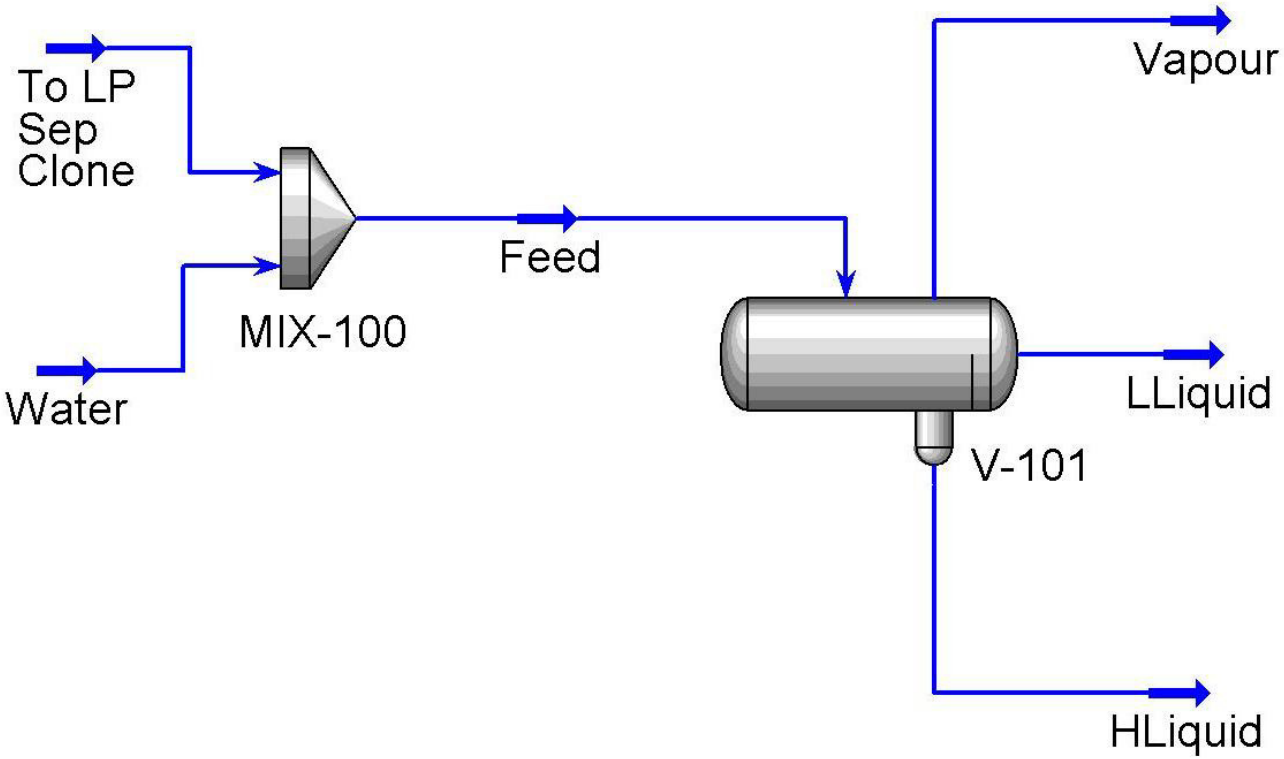
If desired, the user can use a different correlation for each of the calculation steps. In this case, a correlation is specified for each sub-calculation, rather than specifying an overall correlation. Only those parts of the correlation that apply to the particular sub-calculation will be used.

Sub-calculations will not used in this workshop.

Example

If the Generic correlation is used for the Inlet device and ProSeparator is used for primary L-L and G-L separation calculations, then the user-supplied data for the generic inlet calculations (i.e., inlet split and Rossin Rammler parameters) will be used to generate the inlet droplet dispersion. The ProSeparation primary separation calculations will then be performed using this inlet dispersion. **As ProSeparator correlations will not be used to calculate the inlet conditions, any ProSeparator inlet setup data is ignored.** Likewise, any critical droplet sizes entered in the Generic correlation will be ignored as the ProSeparator is being used for the primary separation calculations.

Process Overview



Workshop

Process Description

In this workshop, a 3-phase Separator is used to separate an oil/water/gas mixture. Entrained liquids in the gas product have been identified as a potential process issue. The HYSYS Real Separator will be used to account for liquid entrainment in the model.

Carryover of liquids can be troublesome, especially if the gas is then passed through a turbine/compressor where liquid droplets can cause major damage to the internals of the machine. We will determine if a demisting pad is appropriate to prevent carryover and how to size it appropriately.

The separator considered in this workshop is based on the LP Separator used in the two-stage compression module of the Turbo Expander plant constructed in the Process Modeling Using HYSYS course.

You will begin building the case by creating a copy of the existing separator. This means that while experimenting with the parameters of the separator, the rest of the Turbo Expander plant (recycles, adjust, … etc.) does not have to solve each time.

An exercise later will be to incorporate the rigorous separator into the full model.

Build an Ideal Separator



Don't worry if you have not built the Turbo Expander plant case.

The file 'ADV6_AdvancedRecycles_Soln.hsc' contains this case.

1. Open the two-stage compression case of Turbo Expander plant case.
2. Create a material stream called **To LP Sep Clone**.
3. Double-click on the **To LP Sep Clone** stream.
4. The stream property view appears. Click on the **Define from Other Stream** button.
5. In the Available Streams list, select **To LP Sep**.
6. In the Copy Stream Conditions group, check all the available conditions and click **OK**.
7. Create a stream called **Water**, and specify its temperature and pressure to be the same as **To LP Sep Clone** with a flowrate of 4000 kg/h.
8. Add a **Mixer** and provide the following information:

In this cell…	Enter…
Connections	
Name	MIX-100
Inlets	To LP Sep Clone Water
Outlet	Feed
Parameters	
Automatic Pressure Assignment	Set Outlet to Lowest Inlet

9. Add a **3-phase Separator** and specify it with the following information:

In this cell…	Enter…
Connections	
Name	V-101
Inlets	Feed
Vapour	Vapour
Light Liquid	LLiquid
Heavy Liquid	HLiquid

10. Open the separator unit operation and select the **Worksheet** tab.



What is the vapour fraction and molar flow of the product stream?

Vapour _____

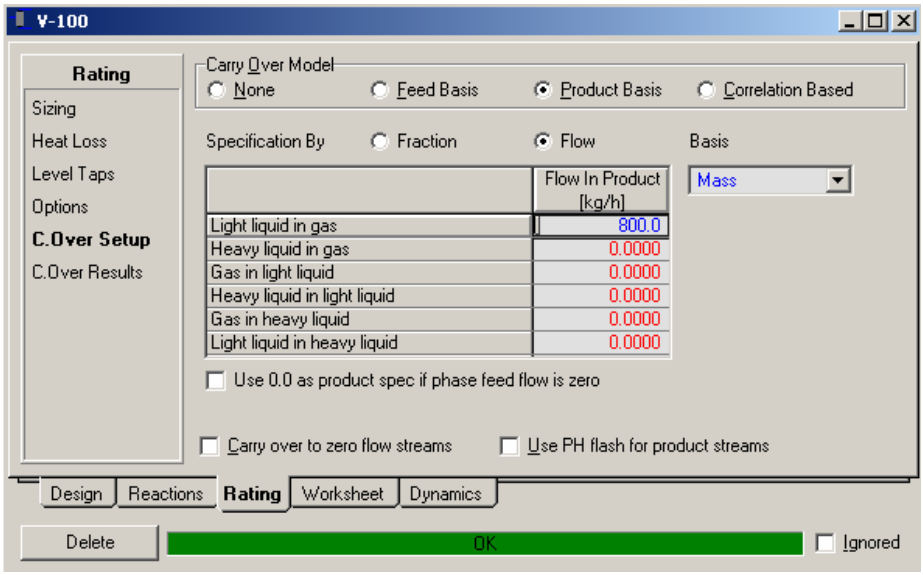
Light Liquid _____

Heavy Liquid _____

Add Carryover Effects

Let us say that we know (from a plant mass balance or as a design assumption) that approximately 800 kg/h of liquid is entrained in the vapour stream. How do we specify this in our model and ensure an accurate mass balance?

1. Select the **Rating** tab. Click on the **C.Over Setup** page to bring up the carryover models, and choose **Product Basis** as the active model.
2. Enter the entrainment data. Select **Specification By: Flow** and choose **Basis = Mass.** Enter **800 kg/h** for **Light liquid in gas**.



3. Examine the product streams and the **C.Over Results** page and compare to the ideal separation case.



What is the vapour fraction of the vapour product stream? _____

What is the rate of liquid carryover (kgmole/h)? _____

Using the Carryover Correlations



*The **Setup** and **Results** views will be different depending on which correlation is used.*

Refer to page 5 for a detailed description of each correlation and its required parameters.



Vessel dimensions can also be entered on the Sizing page of the Rating tab. Data on these two pages is linked.

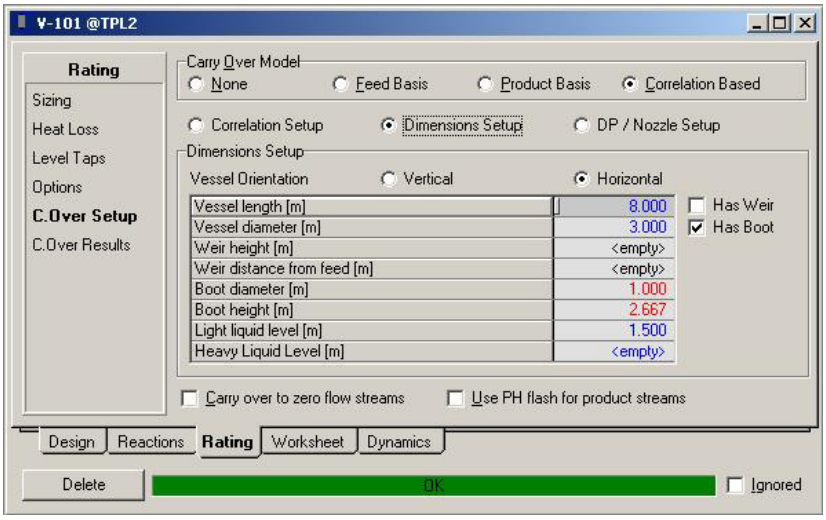
As an alternative to specifying the carryover, we can use correlations to predict the carryover:

1. Return to the **C.Over Setup** page and change the model selection to **Correlation Based.** For steps 2 – 4 select the appropriate radio button.
2. **Correlation Setup (radio button):**
   a) Select **Overall Correlation** and choose the **"ProSeparator"** correlation.
   b) Click the **View Correlation** button to enter inlet and separation parameters.

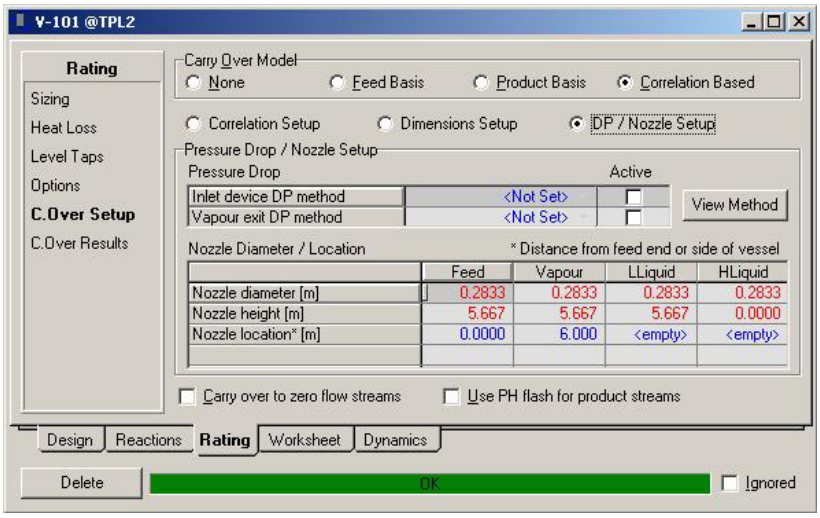
   In this case, the **Inlet** setup page can be left as is. The ProSeparator correlations will calculate the inlet dispersion without the need for further information.

   Since we do not have an exit device, we need to set this for the ProSeparator correlation: select the **Vap. Exit Device** page; select **Mesh Pad;** enter thickness = **0.0**.

   Close the View Correlation window.
3. **Dimensions Setup (radio button):** Enter the vessel dimensions as length **8.0** m, diameter **3.0** m, light liquid level **1.5** m.



4. **DP / Nozzle Setup (radio button):** Enter the following values for nozzle location (this is the horizontal or radial distance from the feed location): Feed **0.0 m**, Vapour **6.0 m**. Keep the default values for nozzle diameter and height.



Analyze the Results

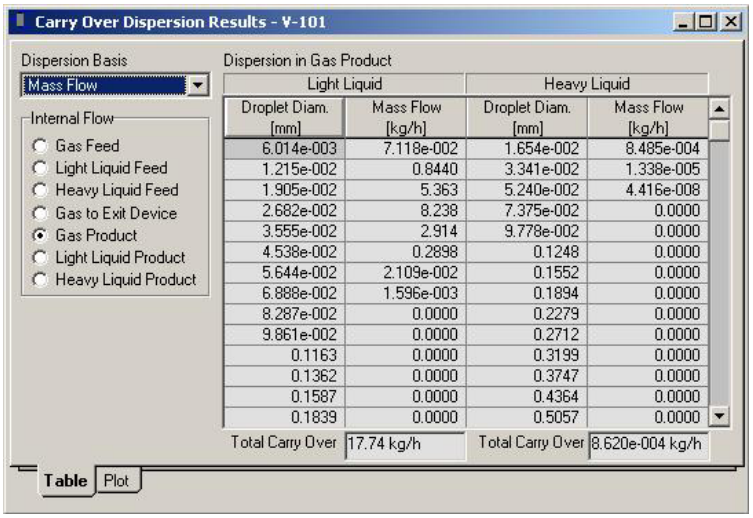
There are several pages where useful results are displayed:

a) Open the **Worksheet** tab.



What is the vapour fraction in the Vapour stream? _____

b) Open the **Rating** tab and select the **C.Over Results** page. To view the carryover details, click the **View Dispersion Results** button. You should see results similar to this:





We need to eliminate all droplets larger than 50 microns (0.05 mm). Do we need an exit device to do secondary separation? _____

Open the **Rating** tab and select the **C.Over Setup** page. Click the **View Correlation** button and open the **Results** tab.

Adding a Secondary Separation Device

1. Open the **Rating** tab and select the **C.Over Setup** page.
2. Click the **View Correlation** button and open the **Setup** tab.
3. Select the **Vap. Exit Device** page; select **Mesh Pad** and enter a thickness of **150.0** mm.



What effect does this have on the carryover? _____

Exercise 1

It is expected that the inlet hydrocarbon flow to the separator may vary by up to 25%. Anticipating that the separator may not be able to handle this increased flow, the engineer decides to model the new conditions in the separator and design a demister pad to remove the larger droplets.

1. Increase the flowrate of the **To LP Sep Clone** stream by 25%.
2. Select the **C.Over Results** page, then click the **View Dispersion Results** button.



What is the Total Carryover with no mesh? With 150mm of mesh?

_____

What is the removal efficiency of 50 micron droplets?

_____

Based on this predicted dispersion, the engineer decides to install a thicker mesh pad. How would you suggest the engineer use HYSYS to determine the correct thickness?



Perform the analysis yourself; how thick should the mesh pad be?

_____

Now what is the vapour fraction of the Vapour product stream?

_____

Exercise 2

Connect the real separator into the two-stage compression loop to replace the ideal separator that is currently in use. Keep the Water feed stream connected. Is the real separator still capable of stopping 50 micron drops reaching the compressor suction?

Carryover in Dynamic Models

Please open sample case “Dynamic Real Separator.hsc”. This case is based on the one you have been working on, but dynamic specifications, controllers and strip charts have been added as needed.

Specifically, the following changes were made to the model:

1. Valves were added to all boundary streams (e.g. Feed0 and VLV-100 were connected to the Feed stream).
2. Pressure-flow specifications were set on all boundary streams (you will find these specifications on the Dynamics tab of each boundary stream, e.g. Feed0 has a pressure specification of 30.05 kPa).
3. Dynamic specifications were set on the separator:

   All dynamic specifications used in this example or the separator were already entered on the Rating tab.

   a. Sizing & carry over data were left the same.
   b. Heat loss left at none
   c. Level taps and PV Work term options were not used
4. Strip charts were created for 2 sets of variables (open the databook tabs titled Variables to see the list of variables and Stripcharts to view the strip chart configurations):

   The Vessel Conditions strip chart tracks vessel pressure, temperature, and liquid level. The Carry Over strip chart monitors liquid phase flow out of the vapour nozzle, as well as inlet flow rate to the vessel.
5. Finally controllers were added to the alternate sample case called Controlled Dynamic Real Separator.hsc.

Demonstration

1. Open **Dynamic Real Separator.hsc**.
2. Click on the strip charts to bring them to the foreground.
3. Click the **Dynamic Mode** button.
4. Start the **Integrator**. When the liquid carryover flow achieves a steady value, stop the integrator.
5. Change the position of VLV-100 to **25% open**. Re-start the integrator. When the liquid carryover flow achieves a steady value stop the integrator.
6. Change the position of VLV-100 to **75% open**. Re-start the integrator. When the liquid carryover flow achieves a steady value stop the integrator.



Is the mesh pad thick enough to account for all process conditions?

_____

A thick pad creates more pressure drop; are there other mitigations to consider?_____

7. Open **Controlled Dynamic Real Separator.hsc**; repeat the same exercise.



What effect does controlling the liquid level have?

_____

Reactions

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
10 Reactions

Workshop

In this module, you will simulate a Synthesis Gas Production facility. This will introduce you to the powerful reaction modelling capability of HYSYS.

The production of synthesis gas is an important step in the production of ammonia. Synthesis gas is comprised of hydrogen and nitrogen at a molar ratio of 3:1. The main role of the synthesis gas plant is to convert natural gas, primarily methane, into hydrogen.

In most synthesis gas plants, four reactors are used. However, in our simulation five reactors will be used to model this process. This is because the combustor, a single vessel, will be modelled as two reactors in series, with two different reaction types. The first reactor is a Conversion reactor and the second is an Equilibrium reactor.

Learning Objectives

After completing this module, you will be able to:

- Simulate reactors and reactions in HYSYS
- Use Set and Adjust Operations to modify a HYSYS simulation

Prerequisites

Before beginning this module, you need to know how to:

- Navigate the PFD
- Add Streams in the PFD or the Workbook
- Add and connect Unit Operations

Reactions and Reactors

There are five different reactor types in HYSYS, by using combinations of these five operations, virtually any real reactor can be modelled. The five reactor types are:

- **Conversion**. Given the stoichiometry of all the reactions occurring and the conversion of the base component, calculates the composition of the outlet stream.
- **Equilibrium**. Determines the composition of the outlet stream given the stoichiometry of all reactions occurring and the value of the equilibrium constant (or the temperature dependant parameters that govern the equilibrium constant) for each reaction.
- **Gibbs**. Evaluates the equilibrium composition of the outlet stream by minimizing the total Gibbs free energy of the effluent mixture.
- **CSTR**. Assumes that the reactor contents are completely mixed in computing the outlet stream conditions, given the stoichiometry for all the reactions that are occurring and the kinetic rate constant (or the temperature dependence parameters for determining the kinetic constant) for each reaction.
- **PFR**. Assumes that the reaction stream passes through the reactor in plug flow in computing the outlet stream composition, given the stoichiometry of all the reactions occurring and a kinetic rate constant for each reaction.



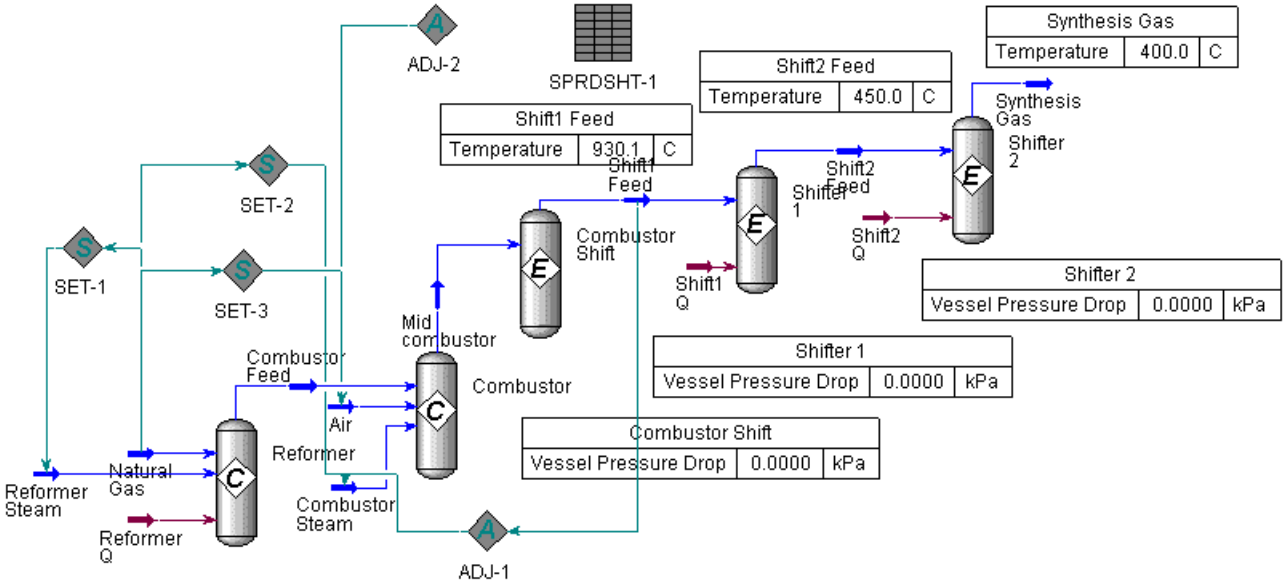
Note that Kinetic, Kinetic (Rev Eqb), and Langmuir-Hinshelwood reactions can be modelled in the CSTR, PFR and Separator.

Note that the required input is different depending on the type of reactor that is chosen. The last two types (CSTR and PFR) must have kinetic rate constants (or the formula to determine the kinetic rate constant) as inputs, as well as the stoichiometry of the reactions. All of the reactor types, except for the Gibbs type, must have the reaction stoichiometry as inputs.

The Tank, Separator, Three Phase Separator, and Column Unit Operations can also process reactions if a reaction set is attached.

The process for entering the reaction stoichiometry is discussed in this module, as is the process for adding reactor Unit Operations to a HYSYS simulation.

Process Overview



Reformer		
Pressure Drop	0.0000	kPa
Spec % Conversion (Rxn-1)	40.00	%
Spec % Conversion (Rxn-2)	35.00	%

Combustor		
Vessel Pressure Drop	0.0000	kPa
Spec % Conversion (Rxn-1)	35.00	%
Spec % Conversion (Rxn-2)	65.00	%
Spec % Conversion (Rxn-3)	100.00	%

Building the Simulation

The first step in simulating a synthesis gas plant is choosing an appropriate fluid package. We will be using the Peng Robinson (PR) EOS in this simulation.

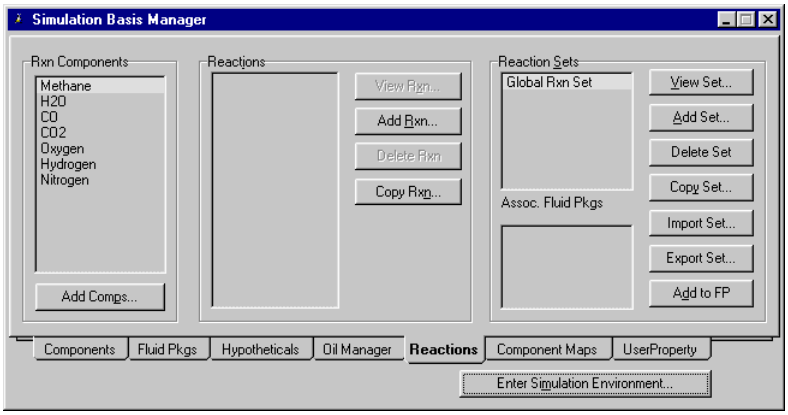
Add the following components to the simulation: **CH_4, H_2O, CO, CO_2, H_2, N_2,** and **O_2**.

Adding the Reactions

Reactions in HYSYS are added in a manner very similar to the method used to add components to the simulation:

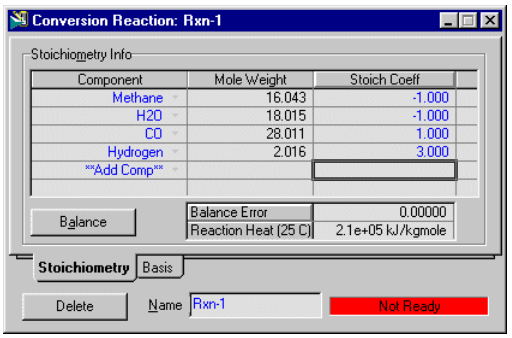
1. Click on the **Reactions** tab in the Simulation Basis Manager view. Note that all of the Components are shown in the Rxn Components list.

Figure 1



2. Click the **Add Rxn** button, and choose **Conversion** as the type from the displayed list. Enter the necessary information as shown:

Figure 2

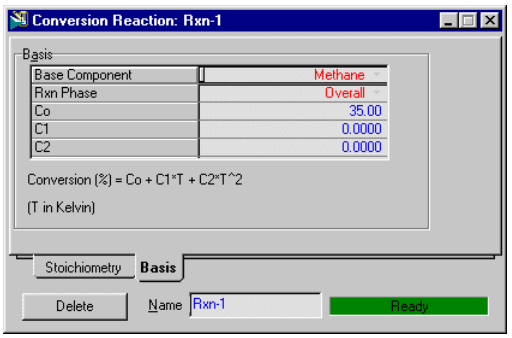




When entering the values for the Stoichiometeric Coefficients, it is important to remember that "Products are positive and Reactants are negative."

3. Move to the **Basis** tab and enter the information as shown:

Figure 3



4. Repeat Steps 2 and 3 for two more Conversion reactions. Use the following data.

Name	Reaction	Base Component	Co
Rxn-2	$CH_4 + 2H_2O \rightarrow CO_2 + 4H_2$	Methane	65
Rxn-3	$CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O$	Methane	100

5. Add an Equilibrium reaction by selecting the reaction type as **Equilibrium** rather than **Conversion**. Under the **Library** tab, highlight the reaction with the form $CO + H_2O \leftrightarrows CO_2 + H_2$. Click the **Add Library Rxn** button. This adds the reaction and all of the reaction's data to the simulation.

Adding the Reaction Sets

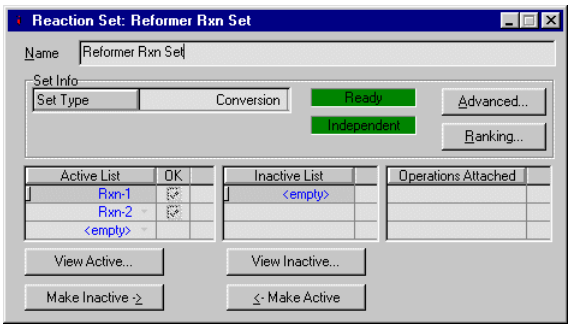
Once all four reactions are entered and defined, you can create reaction sets for each type of reactor.

1. Still on the **Reactions** tab, Click the **Add Set** button. Call the first set **Reformer Rxn Set**, and add **Rxn-1** and **Rxn-2**. Reactions are added by highlighting the **<empty>** field in the **Active List** group, and selecting the desired reaction from the drop down list. The view should look like this after you are finished:



Only reactions of the same type can be included in a reaction set. For example, Equilibrium and Conversion reactions can not be grouped into the same reaction set.

Figure 4



2. Create two more reaction sets with the following information:

Reaction Set Name	Active Reactions
Combustor Rxn Set	Rxn-1, Rxn-2, Rxn-3
Shift Rxn Set	Rxn-4

Attaching Reaction Sets to the Fluid Package

After the three reaction sets have been created, they must be added to the current fluid package in order for HYSYS to use them.

1. Highlight the desired Reaction Set and press **Add to FP**.
2. Select the only available Fluid Package and press the **Add Set to Fluid Package** button.
3. Repeat Steps 1 and 2 to add all three reaction sets (Reformer, Combustor, and Shift).
4. If desired, you can save the Fluid Package with the attached reaction sets. This will allow you to use this Fluid Package in any number of HYSYS simulations.

Once all three reaction sets are added to the Fluid Package, you can enter the Simulation Environment and begin construction of the simulation.

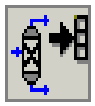
Installing the Material Streams

Create four new material streams with the following information:

Name	Natural Gas	Reformer Steam	Air	Combustor Steam
Temp., °C (°F)	370 (700)	250 (475)	16 (60)	250 (475)
Pressure, kPa (psia)	3500 (500)	<empty>	<empty>	<empty>
Molar Flow, kgmole/hr (lbmole/hr)	90 (200)	240 (520)	90 (200)	140 (300)
Molar Composition	100% - CH_4	100% - H_2O	79% - N_2 21% - O_2	100% - H_2O



Conversion Reactor icon



General Reactors icon

Adding the Conversion Reactors

The first reactor in the synthesis gas plant is the Reformer. This reactor will be modelled as a **Conversion Reactor.**

1. From the Object Palette, click **General Reactors.** Another palette appears with three reactor types: Gibbs, Equilibrium and Conversion. Select the **Conversion Reactor**, and enter it into the PFD.
2. Name this reactor **Reformer** and attach **Natural Gas** and **Reformer Steam** as feeds. Name the vapour outlet **Combustor Feed** and the energy stream as **Reformer Q**. Even though the liquid product from this reactor will be zero, we still must name the stream. Name the liquid product stream as **Reformer LP**.
3. On the **Parameters** page, choose the duty as **Heating**.
4. On the **Details** page of the **Reactions** tab, select **Reformer Rxn Set** as the reaction set. This will automatically connect the proper reactions to this reactor.
5. Once the reaction set is attached, select the **Conversion%** radio button. Change the **Co** value for Rxn-1 to be **40%**, and for Rxn-2 to **30%**.
6. On the **Worksheet** tab, enter a temperature of **930 ^{o}C** (**1700 ^{o}F**) for the outlet stream Combustor Feed.

At this stage the reactor will not yet be fully solved.

The second reactor in a synthesis gas plant is the Combustor. The Combustor will be modelled as a Conversion reactor and an Equilibrium reactor in series. This is because Conversion reactions and Equilibrium reactions cannot occur in reactors of the opposite type, i.e. conversion reactions cannot be associated with equilibrium reactors, and vice versa.

7. Add another **Conversion Reactor** with the following data:

In This Cell...	Enter...
Name	Combustor
Feeds	Combustor Feed, Air, Combustor Steam
Vapour Product	Mid Combust
Liquid Product	Combustor LP
Reaction Set	Combustor Rxn Set
Rxn-1 Conversion	35% (Default Value)
Rxn-2 Conversion	65% (Default Value)
Rxn-3 Conversion	100% (Default Value)

Adding the Set Operations

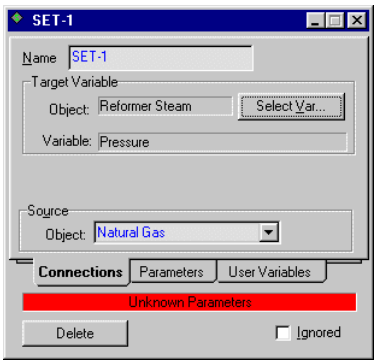
Recall that we did not enter any pressures, except for the natural gas, when we added the material streams to the PFD. This is so that we could now add **Set Operations** to the PFD to set the pressures of the remaining streams.


Set Operation icon

1. Select the **Set Operation** button from the Object Palette.
2. Enter **Reformer Steam Pressure** as the Target Variable, and **Natural Gas** as the Source Variable. This process links the Target Variable to the Source Variable, so that if the Natural Gas Pressure were to change, the Reformer Steam Pressure pressure would match it.

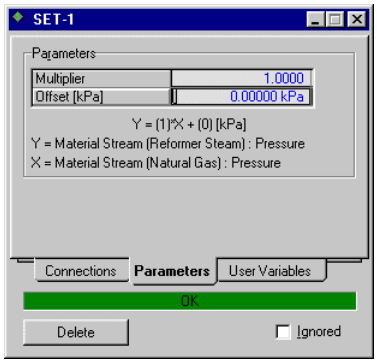
The completed view is shown here:

Figure 5



3. On the **Parameters** tab, set the Multiplier at **1** and the Offset at **0**. For this operation we want a y=x (1:1) relationship. A multiplier of 1 and an offset of 0 will result in this type of relationship.

Figure 6





HYSYS knows to use the pressure value of Natural Gas as the source because a pressure value was selected as the Target Variable.

4. Repeat Steps 1, 2, and 3 with **Combustor Steam Pressure,** and **Air Pressure** as Target Variables, and **Natural Gas** as the Source Variable in both cases. The parameters will be 1 and 0 for these Set operations, as well.

Adding the Shift Reactors

As mentioned before, the Combustor is to be modelled as a Conversion reactor followed by an Equilibrium reactor. The Shift Reactors will also be modelled as **Equilibrium Reactors**. Therefore, a total of three equilibrium reactors must be added to the PFD.



Equilibrium Reactor icon

1. Add an **Equilibrium Reactor** with the following information:

In This Cell...	Enter...
Name	Combustor Shift
Feed	Mid Combust
Vapour Product	Shift1 Feed
Liquid Product	Combustor Shift LP
Reaction Set	Shift Rxn Set

2. Enter another **Equilibrium Reactor** with the following information:



Remember: Set temperature values on the Work Sheet page.

In This Cell...	Enter...
Name	Shifter 1
Feed	Shift1 Feed
Vapour Product	Shift2 Feed
Liquid Product	Shifter 1 LP
Energy Stream	Shift1 Q
Duty	Cooling
Shift2 Feed Temperature	450°C (850°F)
Reaction Set	Shift Rxn Set

3. Enter the third **Equilibrium Reactor** with the following information:

In This Cell...	Enter...
Name	Shifter 2
Feed	Shift2 Feed
Vapour Product	Synthesis Gas
Liquid Product	Shifter 2 LP
Energy Stream	Shift2 Q
Duty	Cooling
Synthesis Gas Temperature	400°C (750°F)
Reaction Set	Shift Rxn Set



What is the mole fraction of Hydrogen in the Synthesis Gas stream? _____

What is the mole fraction of Nitrogen in the Synthesis Gas stream? _____

What is the ratio of H_2 / N_2 in the Synthesis Gas stream? _____

Save your case!

Adding the Adjust Operations

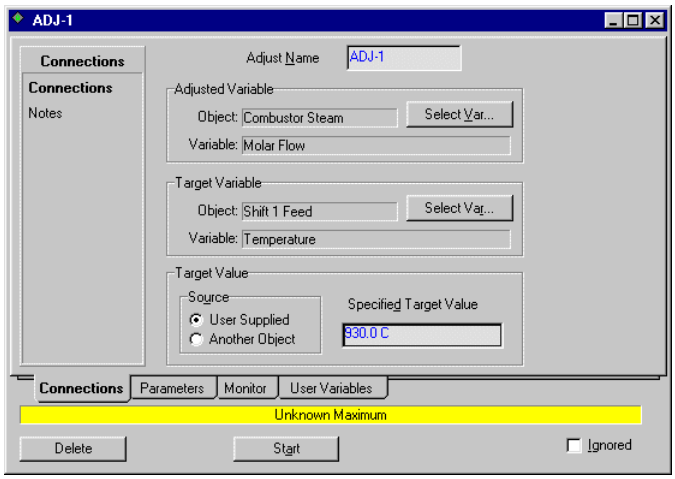
In order to control the temperature of the product stream leaving the Combustor (the second Conversion reactor), the flow rate of steam to this reactor is controlled. It is desired to have an outlet temperature from the first shift reactor of 930°C (1700°F). The steam flow can be adjusted manually until the desired temperature is achieved; however, this takes a lot of time and will not be automatically updated if something were to change. HYSYS contains an adjust function that instructs the solver to adjust one variable until the desired condition is met.



Adjust Operation icon

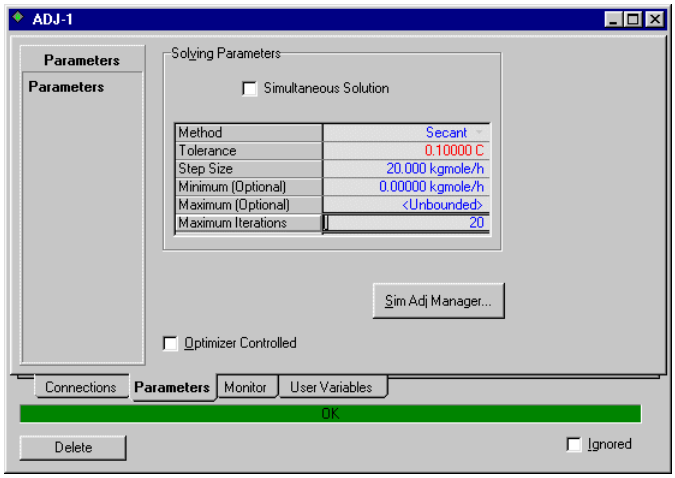
1. Select the **Adjust Operation** button from the Object Palette, and enter it into the PFD.
2. Enter the information as shown:

Figure 7



3. On the **Parameters** tab, enter the information as shown below. The step size in field units will be **44.092 lbmole/h**.

Figure 8





You don't have to be on the Monitor page to start the Adjust Operation, but it shows you the values that HYSYS is using in the calculations.

4. Move to the **Monitor** tab, and click the **Start** button. HYSYS will adjust the steam flow rate until the desired condition is met.

A second Adjust Operation will be used to control the Air Flow rate. The Air Flow rate determines the ratio of H_2 to N_2 in the synthesis gas product. We want this value to be set at 3.05.

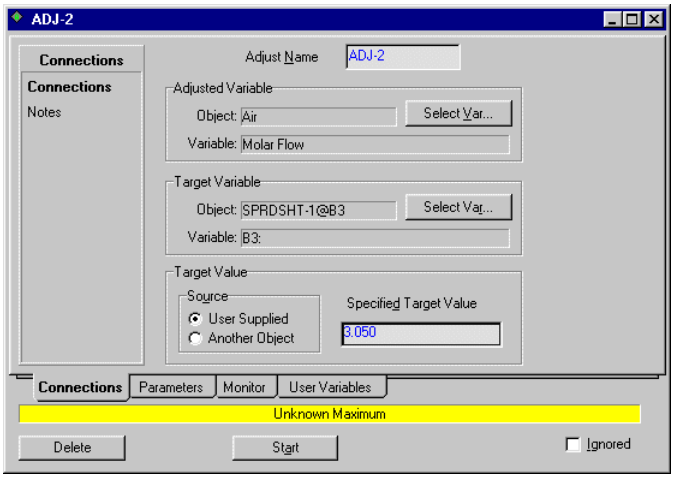


Spreadsheet icon

1. Add a **Spreadsheet** operation to the PFD. (The Spreadsheet is added in the same manner as other unit operations).
2. Import **Synthesis Gas Comp Molar Flow [Hydrogen]** and **Synthesis Gas Comp Molar Flow [Nitrogen]** into the Spreadsheet.
3. Add a ratio formula to an empty cell in the Spreadsheet, e.g. =A1/A2.

4. Add another Adjust operation. Select **Air - Molar Flow** as the Adjusted Variable, and **SPRDSHT-1- B3** (where "B3" is the cell that contains the result of the ratio calculation) as the Target Variable, with a Specified Target Value of 3.05.

Figure 9



5. On the **Parameters** page, choose a tolerance of **0.001** and a step size of **20 kgmole/hr (44.092 lbmole/hr)**.

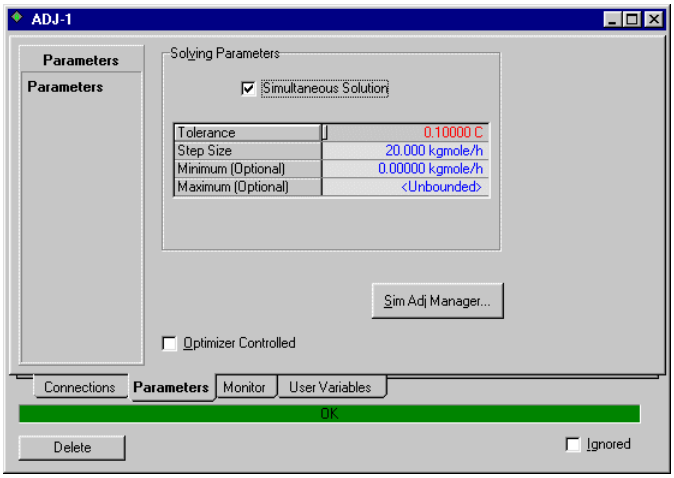
In this case the two Adjust operations may interfere with each other while they are solving. This is because changing either adjusted variable effects both target variables. To prevent this interference the Adjusts can be set to solve simultaneously. This uses a different solution algorithm, which makes the Adjusts solve cooperatively at the end of each flowsheet calculation step.



Press the Sim Adj Manager button to bring up the Simultaneous Adjust Manager. Here all the Simultaneous Adjusts can be controlled in one place.

6. On the **Parameters** tab of the **ADJ-1** operation, check the **Simultaneous Solution** checkbox, as shown below.

Figure 10



7. Repeat step 6 for the second Adjust operation.
8. Start the simultaneous Adjusts solving by using the Start button on the Adjust or in the Simultaneous Adjust Manager.



What is the required Air Flow rate? _____

What is the molar flow rate of Synthesis Gas product? _____

Save your case!

Rating Heat Exchangers

© 2004 AspenTech - All Rights Reserved.

Workshop

A heat exchanger is a vessel that transfers heat energy from one process stream to another. A common physical configuration for heat exchangers is a shell and tube exchanger, where a bundle of tubes sits inside a shell. There is no mixing of fluid between the shell and the tubes.

Learning Objectives

In this workshop, you will learn how to:

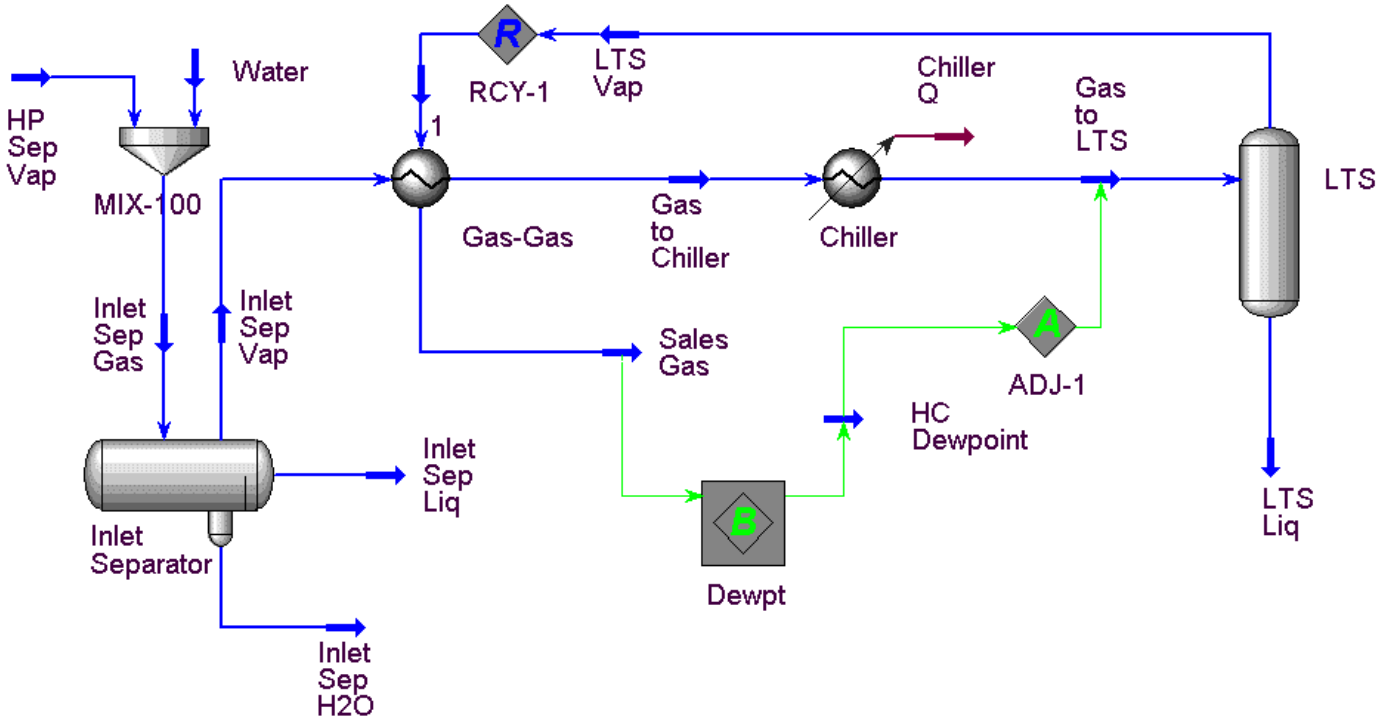
- Use the Heat Exchanger Dynamic Rating Method in HYSYS for heat exchanger design
- Determine if an existing heat exchanger will meet the process specifications

Prerequisites

Before beginning this workshop, you need to:

- know how to install and converge simple Heat Exchangers
- understand the principles of Heat Exchanger design

Process Overview



Modelling Heat Exchangers

In this workshop, we will examine a gas to gas heat exchanger from a Refrigerated Gas Plant. Heat exchangers are modelled in HYSYS using one of three configurations:

- Shell and Tube
- Cooler/Heater
- Liquified Natural Gas (LNG) exchanger

The **Cooler/Heater** operations are single-sided unit operations where only one process stream passes through the operation. The **LNG Exchanger** allows for multiple (more than two) process streams.

A shell and tube heat exchanger is a two-sided unit operation that permits two process streams to exchange heat.

In this module, a shell and tube exchanger of given dimensions will be rated to see if it will meet the requirements of the process.

Heat Exchanger Calculations

The calculations performed by the Heat Exchanger are based on energy balances for the hot and cold fluids. The following general relation defines the heat balance of an exchanger.

$$\left(M_{cold}(H_{out} - H_{in})_{cold} - Q_{leak}\right) - \left(M_{hot}(H_{in} - H_{out})_{hot} - Q_{loss}\right) = BalanceError \qquad (1)$$

where: M = *Fluid mass flow rate*

H = *Enthalpy*

Q_{leak} = *Heat Leak*

Q_{loss} = *Heat Loss*

The Balance Error is a Heat Exchanger Specification which, for most applications, will equal zero. The subscripts “hot” and “cold” designate the hot and cold fluids, while “in” and “out” refer to the inlet and outlet.

The Heat Exchanger duty may also be defined in terms of the overall heat transfer coefficient, the area available for heat exchange and the log mean temperature difference:

$$Q = UA(LMTD)F_t = M_{hot}(H_{in} - H_{out})_{hot} - Q_{loss} = M_{cold}(H_{out} - H_{in})_{cold} - Q_{leak} \quad (2)$$

where: *U= Overall heat transfer coefficient*

A= Surface area available for heat transfer

LMTD = Log mean temperature difference

F_t= LMTD correction factor

Log Mean Temperature Difference (LMTD)

The LMTD is calculated in terms of the temperature approaches (terminal temperature differences) in the exchanger using the following equation:

$$LMTD = \frac{\Delta T_1 - \Delta T_2}{Ln(\Delta T_1 / \Delta T_2)} \quad (3)$$

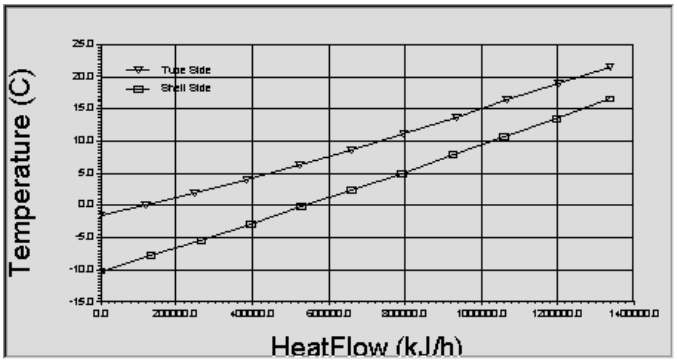
where: $\Delta T_1 = T_{hot,out} - T_{cold,in}$

$\Delta T_2 = T_{hot,in} - T_{cold,out}$

The LMTD can be either terminal or weighted. This means that it can be calculate over the exchanger as a whole (terminal) or over sections of the exchanger (weighted). The need for this type of calculation is shown on the next page.

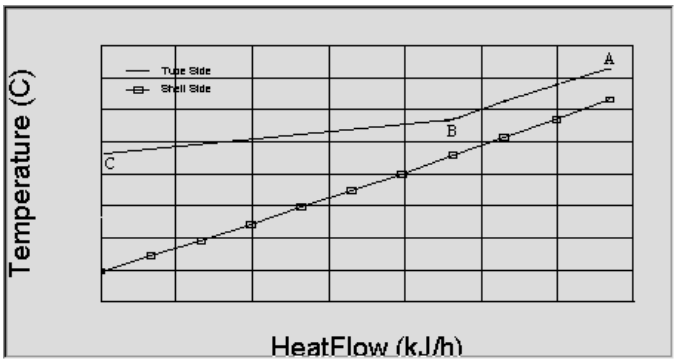
The following plot is a heat loss curve for a single phase stream. It compares the temperatures of the process streams with the heat flow over the entire length of the exchanger. For single phase streams, these plots are linear.

Figure 1



The following curve represents a superheated vapour being cooled and then condensed. Note that it is not linear because of the condensation that takes places inside the exchanger.

Figure 2



If the LMTD is calculated using the hot fluid temperatures at points A and C, the result would be incorrect because the heat transfer is not constant over the length of the exchanger. To calculate the weighted LMTD:

1. Break the heat loss curve into regions at point B.
2. Calculate the terminal LMTD for each region.
3. Sum all of the LMTDs to find the overall LMTD.

HYSYS will do this automatically if the Heat Exchanger model is chosen as **Weighted**. Therefore, if condensation or vaporization is expected to occur in the exchanger, it is important that **Weighted** is chosen as the model.

Heat Exchanger Specifications

As with all other unit operations in HYSYS, the Heat Exchanger is assumed to adequately meet the process requirements. There are several choices for specifications for the heat exchanger. The choices are given here:



Typical specifications for most heat exchangers are Pressure Drops, and one of either, Temperature, Minimum Approach, Duty, or UA.

- **Temperature**. The temperature of any stream attached to the Heat Exchanger. The hot or cold inlet equilibrium temperature may also be defined. The temperature difference between the inlet and outlet between any two streams attached to the Heat Exchanger can also be specified.
- **Minimum Approach**. The minimum temperature difference between the hot and cold stream at any point in the exchanger, i.e. not necessarily at the inlet or outlet.
- **UA**. The overall UA can also be specified. This specification can be used to rate existing exchangers.
- **LMTD**. The overall log mean temperature difference.
- **Pressure Drops**. The pressure drops on both the shell and tube sides on the exchanger are important specifications that should not be ignored. If the pressure drops are not known HYSYS may be able to estimate them.

Care must be taken when choosing specifications because it is possible to select specifications that are either infeasible or impractical. This may result in a Heat Exchanger that will not solve.

Specifications are added on the **Specs** page of the Heat Exchanger Property view. Enough specifications must be added to ensure that the Degrees of Freedom equals 0.

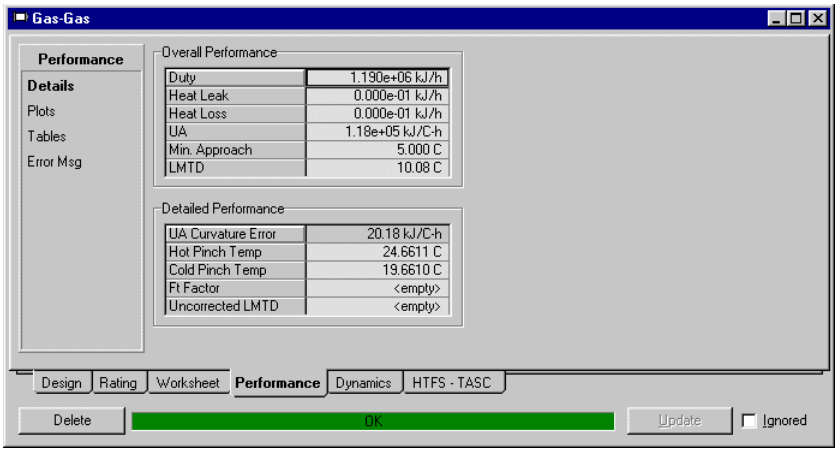
Heat Exchanger Performance



Typically, heat exchangers are solved using delta T minimum approach and UA target values.

A summary of the Heat Exchanger's performance can be viewed on the **Details** page of the **Performance** tab:

Figure 3



Heat exchangers are sometimes compared on the basis of UA values, i.e., for a fixed surface area, what is the amount of heat (duty) that can be exchanged?

1. Open the HYSYS case, **Gas-Gas.hsc** on the disk that was supplied with this module.
2. Double-click the **Gas-Gas** heat exchanger, and answer the following questions.



What is the UA value of the Gas-Gas Exchanger? _____

What is the resulting minimum approach temperature if the UA is fixed at 15 000 kJ/C-h (8000 BTU/F-Hr)? _____

What are the temperatures of streams Gas to Chiller and Sales Gas? _____ ***and*** _____

Heat Exchanger Rating

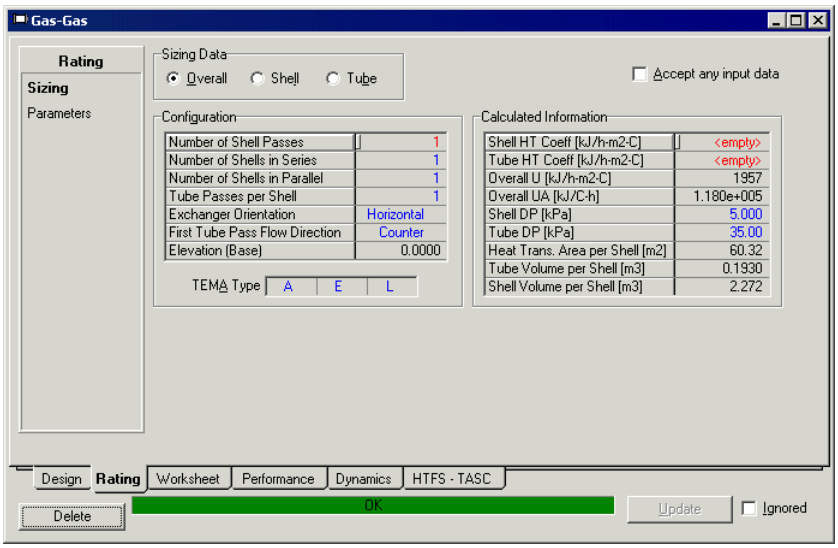
The **Rating** option can be chosen by selecting **Dynamic Rating** from the **Heat Exchanger Model** drop-down menu on the **Parameters** page on the **Design** tab. Delete the Delta P on both the tube and shell side. This is because with this type of model the required information must be specified elsewhere.

Dynamic Rating Model

The physical design specifications of an exchanger must be supplied on the **Sizing** page of the **Rating** tab.

1. Firstly, specify the TEMA type to match the desired conditions.

Figure 4



The radio button selection in the **Sizing Data** group will dictate the type of information shown at any given moment. Each parameter will be defined later on in this module.

The radio buttons in the **Sizing Data** group include:

- **Overall**. Required information about the entire exchanger. Most of the information entered here is used only in dynamic simulations.
- **Shell**. Required information concerning the shell side of the exchanger.
- **Tube**. Required information concerning the tube side of the exchanger.

The **TEMA Type** is selected as part of the Overall sizing data. There are three drop down lists which allow you to specify the geometry of the front end stationary head type, the shell type and the rear end head type for the exchanger. The following tables provide brief descriptions for each designated TEMA Type letter. Drawings of the various TEMA types can be found on page 11-4 of **Perry's Chemical Engineers Handbook, Sixth Edition.**

TEMA - Front End Stationary Head Types

TEMA Type	Description
A	Channel and Removable Cover
B	Bonnet (Integral Cover)
C	Channel Integral with TubeSheet and Removable Cover (removable tube bundle only)
N	Channel Integral with TubeSheet and Removable Cover
D	Special High Pressure Closure

TEMA – Shell Types

TEMA Type	Description
E	One Pass Shell
F	Two Pass Shell with Longitudinal Baffle
G	Split Flow
H	Double Split Flow
J	Divided Flow
K	Kettle Type Reboiler
X	Cross Flow

TEMA - Rear End Head Types

TEMA Type	Description
L	Fixed TubeSheet like 'A' Stationary Head
M	Fixed TubeSheet like 'B' Stationary Head
N	Fixed TubeSheet like 'N' Stationary Head
P	Outside Packed Floating Head
S	Floating Head with Backing Device
T	Pull Through Floating Head
U	U-Tube Bundle
W	Externally Sealed Floating TubeSheet

Rating Parameters

Brief explanations are provided below for each Simple Rating parameter. The parameters are categorized according to the radio buttons in the **Sizing Data** group box. Some of these parameters are only available when the model on the parameters page is selected as **Detailed**.

Overall Information

- **Number of shell passes**
- **Number of shells in series**
- **Number of shell in parallel**
- **Tube passes per shell**
- **Exchanger orientation**. The orientation of the exchanger, used only in dynamic simulations.
- **First tube pass flow direction**
- **Elevation**. The height of the base of the exchanger, used only in dynamic simulations.
- **TEMA**. Described earlier.

Shell Side Required Information

- **Shell Diameter**. Can be specified or calculated from inputted geometry.
- **Number of Tubes per Shell**

- **Tube Pitch**. The shortest centre to centre distance between 2 tubes.
- **Tube Layout Angle**. A choice between four different configurations.
- **Shell Fouling**. The fouling factor on the shell side.
- **Baffle Type**. A choice of single, double, triple, NTIW or grid.
- **Baffle Orientation**. A choice between horizontal or vertical.
- **Baffle Cut (% Area)**. The percent of the cross-sectional profile unobstructed by the baffle.
- **Baffle Spacing**. The distance between adjacent baffles.

Tube Side Required Information

- **Tube OD**. The outside diameter of the tubes.
- **Tube ID**. The inside diameter of the tubes.
- **Tube Thickness**. Usually calculated from the two numbers inputted above.
- **Tube Length**. The tube length per shell (one side for a U-tube).
- **Tube Fouling**. The tube side fouling factor.
- **Tube Thermal Conductivity**. The thermal conductivity of the tubes, used in determined the overall heat transfer coefficient, U.
- **Tube Wall Cp, and Tube Wall Density**. Two physical properties of the tube material, used only in dynamics.

If you want HYSYS to use general correlations to determine the shell and tube side pressure drops and heat transfer coefficients, select the **Detailed** model on the **Parameters** page. This will allow HYSYS to calculate the desired terms.

The Rating model in HYSYS uses generalized correlations for heat transfer coefficients and pressure drop. These correlations are suitable for approximate results in most cases but may not be valid for every exchanger. For more accuracy, a rigorous model may be required. Please contact your Hyprotech representative for a list of available third party heat exchanger packages that are compatible with HYSYS through OLE Extensibility.

Exploring with the Simulation

You are asked to find a heat exchanger that will serve as the Gas-Gas exchanger. However, since you are on a very strict budget, you can only consider used equipment. A heat exchanger has been found in the surplus supply of a nearby plant. If the critical process parameter is to maintain a Sales Gas temperature of at least 10 °C (50 °F), can this heat exchanger be used for the Gas-Gas service? The surplus exchanger has been thoroughly cleaned. The TEMA definition of this exchanger is A,E,L. The pressure drops on both sides of the exchanger should be deleted; this will allow HYSYS to calculate these parameters.

The dimensions of the exchanger are given here:

- Tube Length = 1.5 m
- Number of tubes = 300
- Tube Pitch = 30 mm
- Baffle Type = Double
- Baffle Orientation = Vertical
- Baffle Cut (% Area) = 15%
- Baffle spacing = 100 mm
- All other parameters are the HYSYS default values

Use the **Dynamic Rating** mode to determine if the exchanger is suitable; on the **Rating** tab, **Parameters** page, use the **Detailed Model** in HYSYS.



What is the temperature of the Sales Gas using this exchanger? _____

Previous experience has shown you that after about six months in operation, the exchanger becomes fouled and the fouling factor for both shell-side and tube-side is 0.0001 °C-h-m^2/kJ.



What will the temperature of the Sales Gas be after 6 months of service?
_____

Will this exchanger be adequate after 6 months of service? _____

Save your case!

Challenge

Why was the Recycle needed in this Flowsheet?

For an interesting challenge, disconnect the recycle operation and stream **1**. Connect the stream **LTS Vap** in place of stream **1**.



What one piece of information is stopping the Exchanger from solving ? _____

Apart from putting back the Recycle, how else could this be resolved

_____

Automation Introduction

© 2004 AspenTech - All Rights Reserved.
EA1000.32.02
12 Automation Introduction

Introduction

Automation is the ability to programmatically interact with an application through objects exposed by that application.

By using an Automation client like Microsoft Excel, or Visual Basic, the end user can write code to access these objects and interact with HYSYS. Code can also be written in HYSYS itself in the form of User Variables or Macro Language Editor (MLE) macros. The available objects are the same.

The exposed objects make it possible to programmatically perform nearly any action that can be accomplished through the HYSYS graphical user interface.

Workshop

In this module you will review and begin to understand an Automation front-end to the Turbo Expander case using Microsoft Excel. Additionally a simple HYSYS User Variable will be created.

Learning Objectives

In this module, you will gain an understanding of the possibilities that Automation access to HYSYS can bring. The examples given should give a starting point for any further learning.

If you want to learn more about Automation programming with HYSYS, AspenTech offers another course that will meet your needs. Ask the instructor for more information.

Prerequisites

No prior programming experience is assumed. However before beginning this module, you should have a reasonable understanding of the HYSYS program.

Why Use HYSYS Automation?

The main reason for using the Automation capabilities of HYSYS is to improve the efficiency of your work processes. By reducing the amount of time spent on repetitive, tedious tasks, and hence by reducing the amount of human errors, more time is left for engineering tasks.

The efficiency of your work processes can be increased by using HYSYS Automation for:

- Automating manual data entry tasks, for example extract HYSYS data to an Excel spreadsheet for delivery to a vendor.
- Creating hybrid solutions across applications, for example allow for seamless data transfer between HYSYS and any other Automation enabled application.
- Hide the complexity of HYSYS while taking advantage of its full capabilities, for example build custom front ends for plant personnel.
- Extend the functionality of HYSYS to meet particular needs, for example use a User Variable to report custom stream properties.

The benefits that you will see from HYSYS Automation will depend entirely on what you use HYSYS for. If you find yourself performing the same task or calculation several times during a project, writing a MLE macro or a User Variable will save a lot of valuable project time. This will be especially true if the calculation is complex or requires several variables simultaneously.

Excel Front-End to the Turbo Expander Plant



Don't worry if you haven't built the Turbo Expander plant case.

The file "ADV5_Spreads&CaseStud_Soln.hsc" contains this case.

A simple front end to the Turbo Expander plant has been constructed using Microsoft Excel.

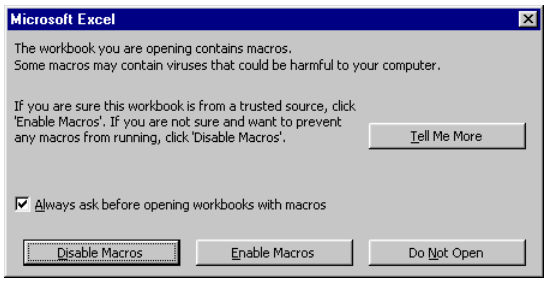
Rather than typing a large amount of code into the Excel Visual Basic Editor, in this Workshop you will review some prewritten code using VBA's debugging tools.

The instructor may choose that the class does this individually, or as a group.

Preparation

1. Open the Turbo Expander HYSYS case.
2. Open the Microsoft Excel file: "Adv Automation - Solution.xls".
3. In order to use VBA macros in Excel you need to tell Excel to **Enable Macros**.

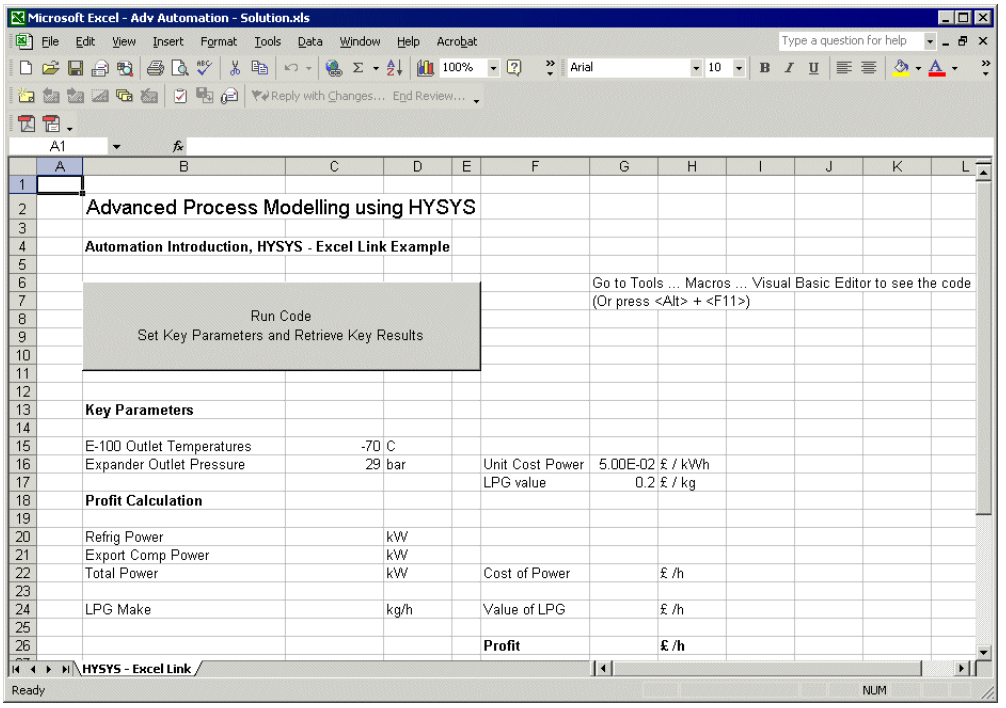
Figure 1



In Excel 2000 you may need to change the **Security** setting to **Medium** on the **Tools... Macro... Security** menu option, before you see this window on opening the file.

The Excel spreadsheet has already been set up with some labels and values.

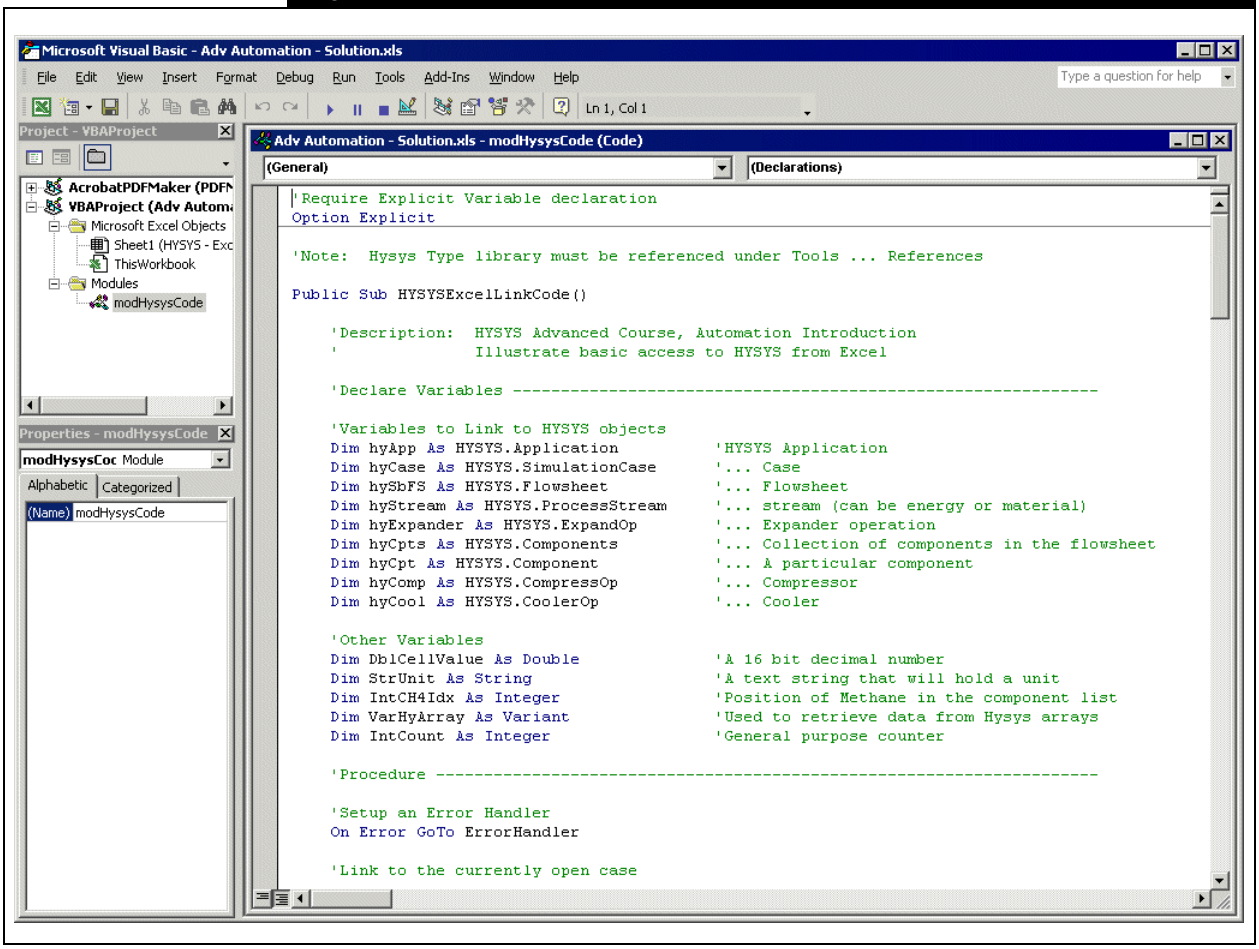
Figure 2



4. Open the Visual Basic Macro Editor by going to **Tools... Macro... Visual Basic Editor**, or by pressing **Alt + F11.**

The code to link to HYSYS will now appear.

Figure 3



Making a Type Library reference

The first step in accessing HYSYS via VBA is to make a Type Library reference.

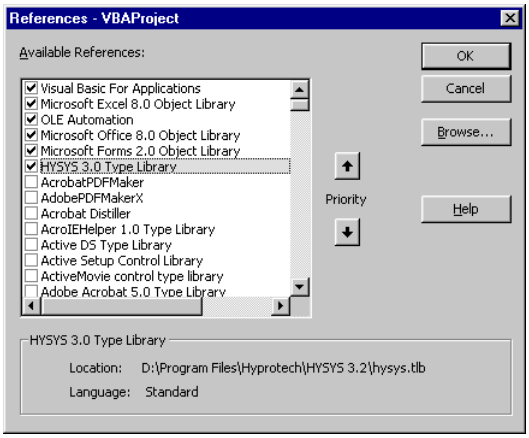
1. In the VBA Editor go to the **Tools … References** menu option.
2. Ensure the **HYSYS 3.0 Type Library** entry is checked.



Don't worry if the type library version number doesn't correspond to the HYSYS version being used.

Just check that the Location is for the correct HYSYS version.

Figure 4



VBA Basics

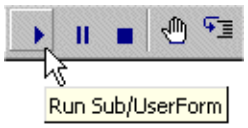
The intention of this section is to introduce the very basics of manipulating VBA code rather than to teach the details of HYSYS VBA programming.

The techniques described here will be used later when examining the prewritten code.

Running Macros

To run a macro, place the cursor within the macro in the VBA Editor, and click the Run Sub / UserForm toolbar button.

Figure 5



Macros can also be run within the VBA Editor, from the Run menu or by pressing **F5**.

Additionally it is possible to trigger a macro by clicking a button on the worksheet. The example spreadsheet has a button to do this. (To set which macro is triggered, right-click on the button and choose Assign Macro.)



Breakpoints cannot be placed on comment lines or variable declaration lines.

Comment lines are those starting with ‘ marks, that appear green in the VBA editor. These are ignored when the code runs.

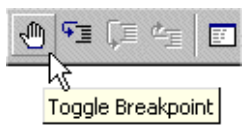
Simple Debugging - Breakpoints

In this module the example code will be run in Break mode. This allows the code to be stepped through one line at a time. This helps to gain understanding of what the code is doing, and is also useful when fixing bugs.

To make VBA enter Break mode it is first necessary to add a breakpoint. First select the position in the code at which to add a break point then either:

- Select **Toggle Breakpoint** from the **Debug** menu in the main menu bar.
- Press the **F9** hot key.
- Click in the grey column on the left side of the code window beside the desired breakpoint location.
- Use the **Toggle Breakpoint** toolbar icon.

Figure 6

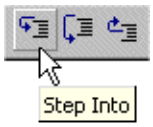


Next trigger the code as above. When VBA encounters the breakpoint execution will stop and the code window will appear. The line of code that is about to be processed is highlighted in yellow.

There are a number of ways to step through the code:

- Select **Step Into** from the **Debug** menu.
- Press the **F8** hot key.
- Use the **Step Into** toolbar icon.

Figure 7



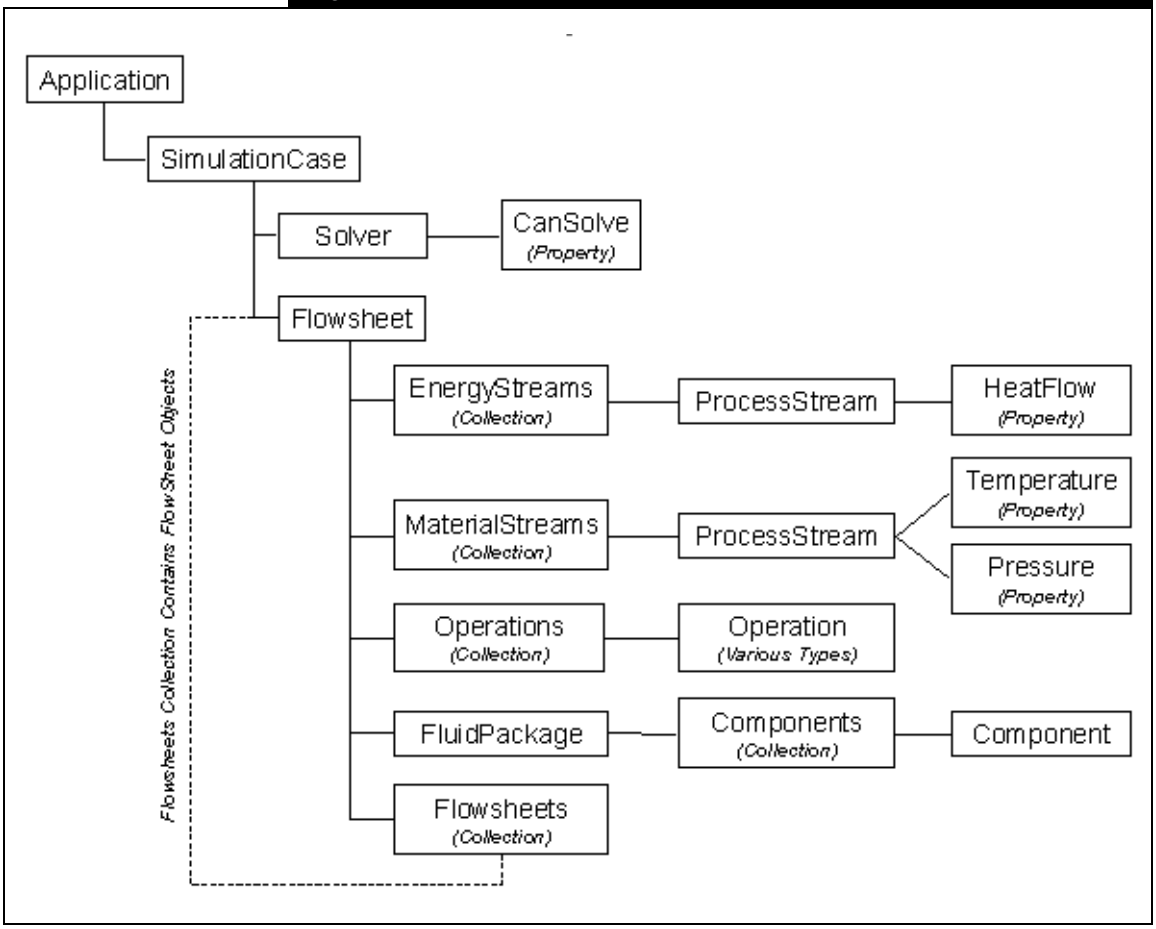
HYSYS Automation Basics

Each object within HYSYS, for example: a stream, the flowsheet, a case, or even the HYSYS application itself has a corresponding Automation object. It is via these objects that HYSYS can be accessed and controlled through code.

This example illustrates access to some of the most commonly used Automation objects within HYSYS.

HYSYS Objects are organised into a tree: The object hierarchy. The objects that will be accessed in this example are illustrated below:

Figure 8



The first stage of linking to HYSYS is to link in to an object at the top of the tree. In this example the line of code:

```
Set hyApp = GetObject(, "HYSYS.Application")
```

is used. This sets up the object variable hyApp to refer to the HYSYS application. The resulting object is of type **Application**.

To link to further objects within the tree, dot notation is used. For example the Turbo Expander Excel interface uses the line:

```
Set hyCase = hyApp.ActiveDocument
```

to refer to the currently active simulation. The resulting object is of type **Simulation Case**.

i.e. to refer an object on the next layer down in the tree a full stop is used.

Using the Object Browser

The objects available within HYSYS, and the HYSYS object hierarchy can be viewed using a VBA tool called the Object Browser. To view the Object Browser either:

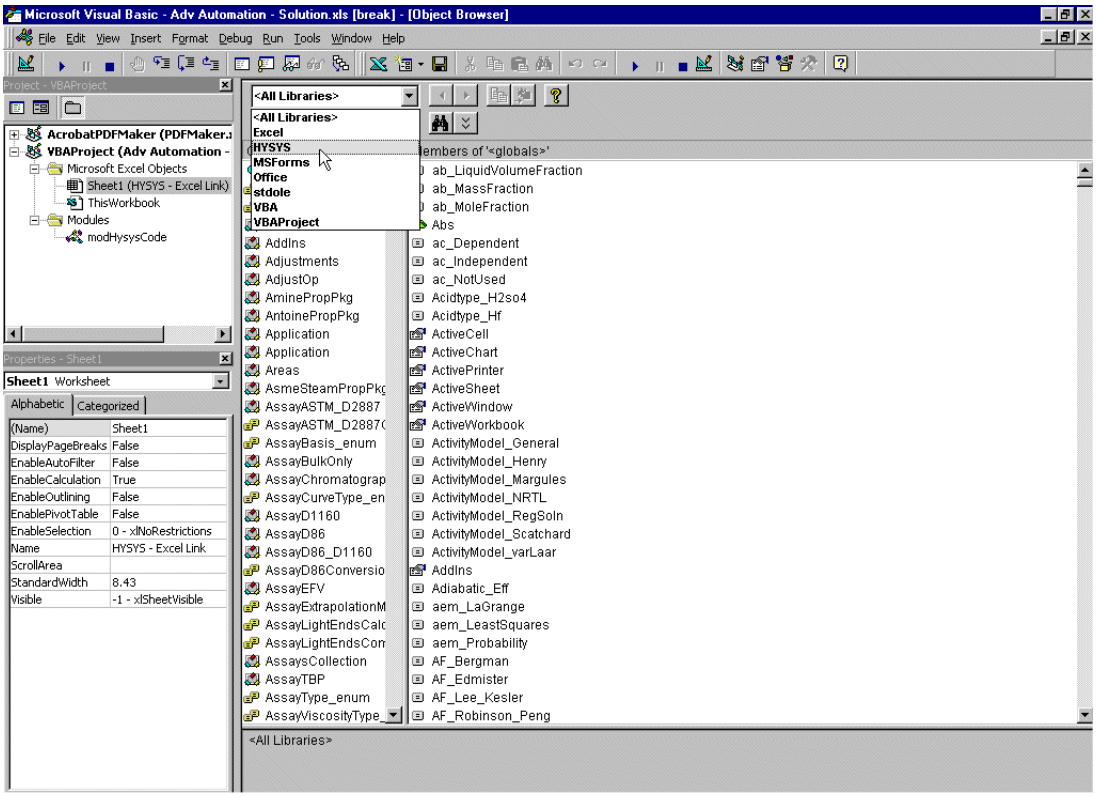
- Select **Object Browser** from the **View** menu.
- Press the **F2** hot key.
- Use the **Object Browser** toolbar icon.

Figure 9



The object browser window will appear. By default the object browser will list the objects available in all the type libraries that are selected in the References lists (Tools … References). In order to limit it to just HYSYS objects change the <All Libraries> drop-down at the top left to HYSYS.

Figure 10



Clicking on an object (or 'Class') in the left hand list then shows all the members (objects or properties) that are associated with that object. The Object Browser also allows searching.

Using Watches to monitor Variables



It is good practice to declare all variables at the top of the procedure.

Variables are key to any programming language. They hold values that can be manipulated by the program as it runs.

When accessing HYSYS via Automation, an Object Variable can be used in the code to link to a HYSYS object.

By adding a **Watch** it is possible to see how the value of a variable changes as the code executes in Break Mode.

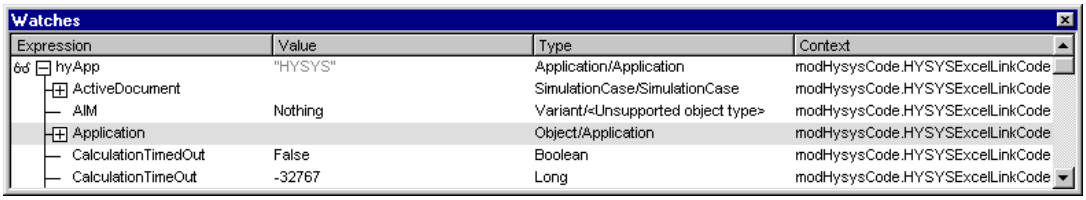
First the Watches window must be shown. Do this by choosing the **Watch Window** option in the **View** menu, or by using the **Watch Window** toolbar icon.

Figure 11



The Watches window will then appear.

Figure 12



Secondly select the variable of interest in the code by clicking and dragging, then either:

- Drag the selected variable onto the Watches window.
- Choose **Add Watch…** or **Quick Watch…** from the **Debug** menu
- Press the **SHIFT F9** hot key.

If the variable is an object variable then it will appear with a '+' sign next to it, clicking this will show any sub-objects and properties within the object.

The Watches window illustration above shows what happens if the hyApp object is added to the watch window and the code run in break mode.

Running the Code in Break Mode

1. First look at the top of the procedure. It is good practice to include a description of what the procedure does at the beginning. All the variables are then declared. Note that the variables that will be linked to HYSYS objects are declared as specific types depending on the kind of object they are linked to.

Figure 13

```
Public Sub HYSYSExcelLinkCode()

    'Description:  HYSYS Advanced Course, Automation Introduction
    '              Illustrate basic access to HYSYS from Excel

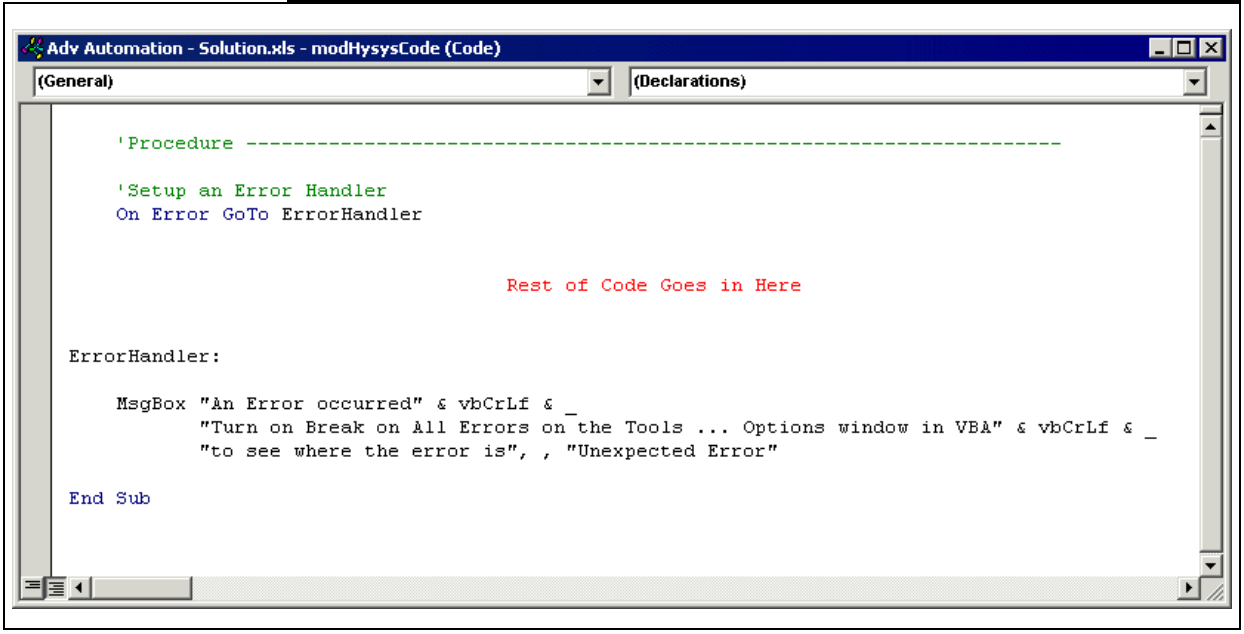
    'Declare Variables ------------------------------------------------------------

    'Variables to Link to HYSYS objects
    Dim hyApp As HYSYS.Application          'HYSYS Application
    Dim hyCase As HYSYS.SimulationCase      '... Case
    Dim hySbFS As HYSYS.Flowsheet           '... Flowsheet
    Dim hyStream As HYSYS.ProcessStream     '... stream (can be energy or material)
    Dim hyExpander As HYSYS.ExpandOp        '... Expander operation
    Dim hyCpts As HYSYS.Components          '... Collection of components in the flowsheet
    Dim hyCpt As HYSYS.Component            '... A particular component
    Dim hyComp As HYSYS.CompressOp          '... Compressor
    Dim hyCool As HYSYS.CoolerOp            '... Cooler

    'Other Variables
    Dim DblCellValue As Double              'A 16 bit decimal number
    Dim StrUnit As String                   'A text string that will hold a unit
    Dim IntCH4Idx As Integer                'Position of Methane in the component list
    Dim VarHyArray As Variant               'Used to retrieve data from Hysys arrays
    Dim IntCount As Integer                 'General purpose counter
```

After the variable declarations, there is a VB instruction that sets what happens when an error occurs. In this case, jump to the label ErrorHandler at the bottom of the code, and display an error message.

Figure 14



2. Add a breakpoint to the On Error line and start the code running. (If you need to, you can refer back to **page 8** for details of breakpoints, and making the code run.) Make the Set hyApp … line execute.

Figure 15

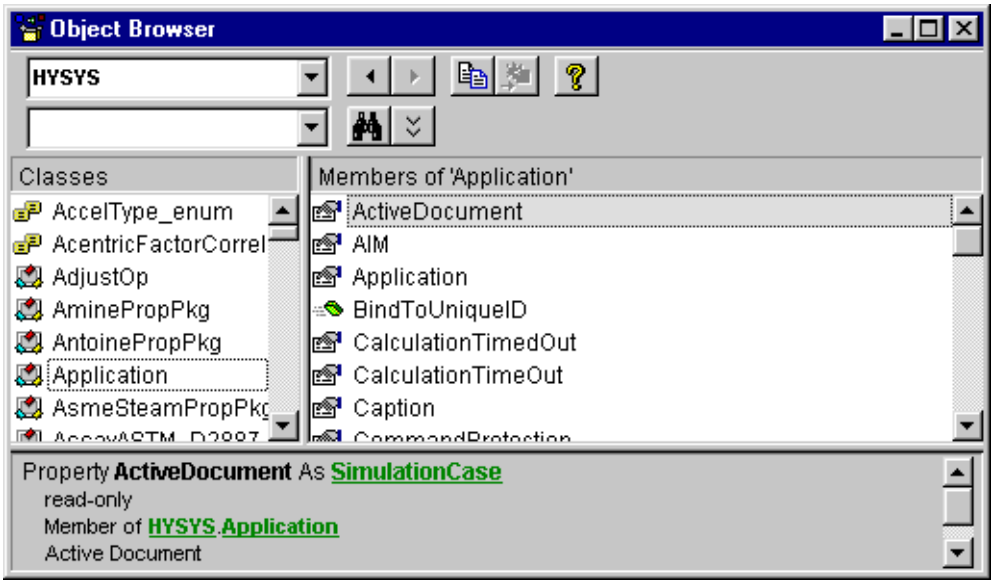
```
'Procedure ---------- 'Setup an Error Handler On Error GoTo ErrorHandler 'Link to the currently open case Set hyApp = GetObject(, "HYSYS.Application")     'Only works if Hysys is open Set hyCase = hyApp.ActiveDocument
```

3. Add a Watch for the hyApp variable, click the '+' sign next to the variable to show the properties of this object, note that ActiveDocument is one of them, and that it's type is SimulationCase.

4. Execute the next line of code, add a Watch for hyCase and note that it shows the same information as the ActiveDocument property of the hyApp object.
5. Use the object browser to view the HYSYS Application object. Note again that the type of the ActiveDocument property is SimulationCase.

Hence when hyCase was declared, it's type was SimulationCase.

Figure 16



The next section of code checks if there was a case open in HYSYS using the VB 'Is Nothing' construction.

Figure 17

```
'Check that a case was open
If hyCase Is Nothing Then
    'Tell the user
    MsgBox "Make sure the Turbo Expander case is open in HYSYS", , "Error"
    'Finish our procedure
    Exit Sub
End If
```

Try running the code with no case open. What happens now?

What is the value of the hyCase variable in the Watches window?



Next some values are read in from the Excel spreadsheet.

Figure 18

```
'Read the required E-100 temperature value and unit from an Excel cell----------

'There are a number of ways to do this
DblCellValue = ActiveSheet.Range("C15").Value       'Cell name
StrUnit = ActiveSheet.Cells(15, 4).Value            'Row, Col Address
'ActiveSheet is an Excel object for the currently open Worksheet
'It is implicitly used by Excel VBA anyway so it's not necessary to have it every time
```

There are a number of ways to refer to a particular cell in the spreadsheet:

- Range() with a cell name. e.g. Range(“C15”).Value
- Range() with a named range. e.g. Range(“MyRange”).Value
- Cells() with a row column reference. e.g. Cells(15,4).Value

Try adding a Watch for ActiveSheet.Range(“C15”), what kind of properties does this have?

What is the type of this variable? Where in the Object Browser could this object type be found?



6. Run the code down to the Set hyStream = …. line

Figure 19

```
'Set an object reference for the outlet stream of the cooler
'Reference an item in the collection of all Material streams
'in the main flowsheet
Set hyStream = hyCase.Flowsheet.MaterialStreams.Item("2")
'Note that because hyStream is an object variable the Set keyword must be used

'Each time a change is made in the HYSYS case via Automation, HYSYS resolves
'since multiple changes are going to be made, it makes sense to turn off the
'HYSYS solver, make all the changes and then turn it back on again.
hyCase.Solver.CanSolve = False
```



Note that it is important to enclose the stream name in "quotation marks".

Here a particular stream within the flowsheet is placed into the hyStream object variable.

7. Look up SimulationCase in the Object browser and navigate through to the MaterialStreams property of the Flowsheet object.

This is a Collection. In programming terms Collections are objects that hold a group of sub-objects - in this case all the material streams in the flowsheet. The Item property is used to refer to a specific stream.

8. Add a Watch for the hyStream object. Look at all the properties this has. Note that the Type of this variable is ProcessStream.
9. Execute the hyCase.Solver.CanSolve = False line. In HYSYS the solver is now turned off (Red Traffic light).

Now the temperature in the stream can be set using the two values that were retrieved from the Excel spreadsheet earlier.

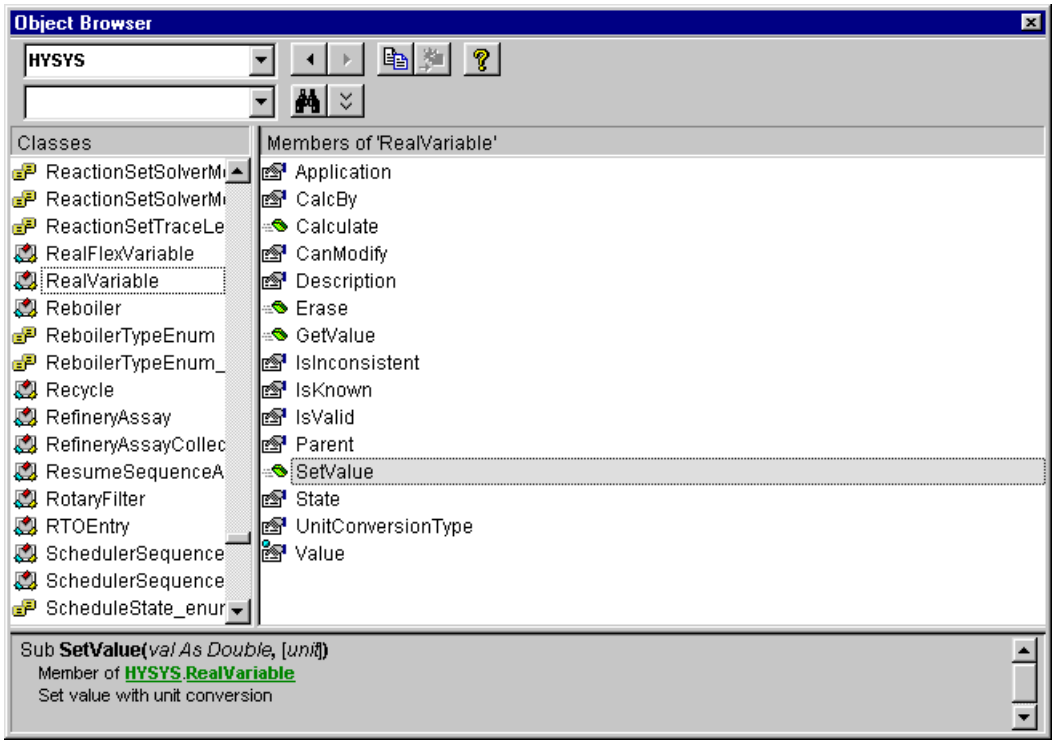
Figure 20

```
'Now set it's temperature
hyStream.Temperature.SetValue DblCellValue, StrUnit
'Use the Setvalue method of the Temperature property of the ProcessStream object
'This takes two parameters: the value to set, and the unit to set it in
```

10. Execute this line - check that the value in the HYSYS case matches that entered in Excel.
11. Look up the ProcessStream object in the Object Browser and find the Temperature property. This is of type RealVariable.

12. Look up RealVariable in the object browser.

Figure 21



RealVariables are special HYSYS objects that hold numerical values, but also contain other information that is useful for the programmer.



Method icon in the Object Browser

- **IsKnown**. Whether the value is known or is <empty>
- **CanModify**. Whether the value can be changed (i.e. True = a Blue HYSYS number, False = Black, calculated by something else)
- **CalcBy**. Which HYSYS object calculated the value
- **UnitConversionType**. What kind of unit the value has

RealVariables also have two useful methods: SetValue and GetValue. These allow numbers to be put into, and retrieved from HYSYS. In general Methods are used to tell objects to do something.

The SetValue method takes two parameters:

```
Sub SetValue(val As Double, [unit])
```

The first parameter is the new value to be set, and the second is the unit to set the value in. Note that the square brackets mean that the second parameter is optional. If it is not supplied then HYSYS assumes the value is being supplied in it's internal units. (°C in the case of temperature.)



Look at the Temperature in the Watch window. What are the values of some of the properties listed above?

13. Execute the code down to the line just above where the HYSYS solver is turned back on.

Figure 22

```
'Similarly set the pressure of the expander outlet stream------------------------

'Reference the expander object
'Like material streams, there is a collection of all the operations in the main
'flowsheet
Set hyExpander = hyCase.Flowsheet.Operations.Item("K-100")
'Now link to it's product stream
Set hyStream = hyExpander.ProductStream

DblCellValue = Range("ExpndrOutP").Value            'Can also reference a named range
StrUnit = Range("ExpndrOutP").Offset(0, 1).Value    'Or a given offset from another cell

'The ProcessStream object has another property called Pressure
'Set this is in the same way as temperature
hyStream.Pressure.SetValue DblCellValue, StrUnit

'Turn the Solver back on
hyCase.Solver.CanSolve = True
'HYSYS will completely solve before moving on to the next command
```

This section of code is similar to the section above where a temperature is set. However instead of referring directly to a named stream, the product stream from a given operation is used. The flowsheet object has another collection called Operations that includes all the operations on the flowsheet.

14. Add a **Watch** for the hyExpander object and look at some of the properties it has.

15. Reposition the VBA editor window and the HYSYS window so that both are visible.
16. Execute the line hyCase.Solver.CanSolve = True, and observe that HYSYS resolves the case.

The next parts of the code retrieve various information from the newly modified HYSYS case.

Figure 23

```
'Retrieve Results Data from the HYSYS case-------------------------------------

'Link to the compressor energy stream on the refrig loop sub flowsheet

'Get an object for the sub flowsheet
Set hySbFS = hyCase.Flowsheet.Flowsheets.Item("TPL1")
'Each Flowsheet object has a collection called Flowsheets which contains
'all it's sub flowsheets

'Now access the stream as before

'This time use the EnergyStreams collection of the main flowsheet
Set hyStream = hySbFS.EnergyStreams.Item("Comp Duty")

'Get the heat flow
StrUnit = Range("ProfitResultsStart").Offset(0, 1)
'All the properties that have a .SetValue method also have
'.GetValue methods to get results out
'This takes only one parameter - the required unit
Range("ProfitResultsStart") = hyStream.HeatFlow.GetValue(StrUnit)
```

Since the “Comp-HP” stream is on a sub flowsheet, it must be accessed from an object for the sub flowsheet. Like the MaterialStreams collection, each flowsheet has a collection (called Flowsheets) which contains objects for all it's sub flowsheets. Hence the Item property is used on the Flowsheets collection of the main flowsheet, to refer to a particular sub flowsheet within the collection.

Flowsheets are referred to by their Tag (not their Name). This can be found on the Connections tab.



Variables can be reused when they are finished with. There is no need to declare a different variable for each stream or operation that is accessed.

Next a new stream is assigned to the hyStream variable, using the Item property of the EnergyStreams collection of this new sub flowsheet object. The EnergyStreams collection contains only the Energy streams in the flowsheet.

The ProcessStream object type can hold both Material and Energy stream objects, however Energy streams have far fewer accessible properties. In this case the HeatFlow property is used.

The GetValue method is then used. This is analogous to the SetValue method except that it takes only one parameter: the required unit.

17. Execute the Set hyStream = … line.
18. Look at the Watch for the hyStream variable now - Note that it now carries details of the energy stream.



Is there a way to tell whether a ProcessStream object refers to a material or energy stream?

Hint: Use the object browser to look for properties of the ProcessStream object starting with 'i'.

Similarly the compressor duty and LPG product mass flow are obtained. Note that there is no need to obtain an object variable for a stream or operation before running the GetValue method against one of it's properties. However if the same object is to be used more than once it makes sense to create an object variable for it.

Figure 24

```
'Similarly for the Export Gas Compressor
'It is not necessary to make an object for the stream every time
Range("ProfitResultsStart").Offset(1, 0) = _
  hyCase.Flowsheet.EnergyStreams.Item("QK-102").HeatFlow.GetValue(StrUnit)
'The _ operator allows the code to be written on multiple lines for clarity
'VBA treats the code as one line when it runs it

'Get the flowrate of the LPG product stream
StrUnit = Range("ProfitResultsStart").Offset(4, 1)
Range("ProfitResultsStart").Offset(4, 0) = _
  hyCase.Flowsheet.MaterialStreams.Item("10").MassFlow.GetValue(StrUnit)
```

19. Execute the code above.

The next section of the code retrieves some additional values from some operations on the main flowsheet.

Figure 25

```
'Report some other Results ------------------------------------------------------

'Get Compressor K-101 Pressure Rise
Set hyComp = hyCase.Flowsheet.Operations.Item("K-101")
StrUnit = Range("OtherResultsStart").Offset(0, 1)
Range("OtherResultsStart").Offset(0, 0) = hyComp.DeltaP.GetValue(StrUnit)

'And E-101 Temperature Drop
Set hyCool = hyCase.Flowsheet.Operations.Item("E-101")
StrUnit = Range("OtherResultsStart").Offset(1, 1)
Range("OtherResultsStart").Offset(1, 0) = hyCool.DeltaT.GetValue(StrUnit)
```

20. Execute this section of the code.

The final sections of the code deal with obtaining component specific properties.

Figure 26

```
'Mole Fraction of Methane in Export Gas stream----------------------------------
'First must determine the position of Methane in the component list

IntCH4Idx = -1

'Get an object for the collection of components in the fluid package
Set hyCpts = hyCase.Flowsheet.FluidPackage.Components

'All collections have a Count property that says how many items they have
'Now count through the components to find Methane
For IntCount = 0 To hyCpts.Count - 1
    'Get a particular component
    Set hyCpt = hyCpts.Item(IntCount)
    'Check it's name
    If hyCpt.Name = "Methane" Then
        IntCH4Idx = IntCount
        'Exit the For ... To ... Next loop
        Exit For
    End If
Next IntCount
```

21. Execute this section of the code.

Note that each time execution reaches the Next IntCount line, it jumps back up to Set hyCpt = … line. This is the first time a VB loop has been used in this code. Each time the loop executes the value of the IntCount variable is increased until it reaches the number of components minus one.



Why shouldn't the loop count up to hyCpts.Count?

All the HYSYS collections are "zero based" - The first term is at position zero.

The result of this loop is that if Methane is present in the case then it's position in the list of components will have been placed into the IntCH4Idx variable.

Figure 27

```
If IntCH4Idx <> -1 Then
    'Nethane is in the case

    'Get an array of all the component mole fractions in the Export Gas stream
    VarHyArray = hyCase.Flowsheet.MaterialStreams _
      .Item("EXPORT GAS").ComponentMolarFraction.GetValues("")

    'Here the GetValues() method of a RealFlexVariablei is used - this is how
    'HYSYS deals with arrays of values.
    'Since in this case a mole fraction is being used there are no units

    'Retrieve the Methane value in this array - this is at index IntCH4Idx
    Range("OtherResultsStart").Offset(2, 0).Value = VarHyArray(IntCH4Idx)

Else
    Range("OtherResultsStart").Offset(2, 0).Value = ""
End If
```

All HYSYS values that contain arrays of data (e.g. component mass fractions, or mole flows, or tray by tray data in a column) have a special object type called RealFlexVariable.

22. Look up the ComponentMolarFraction parameter of the ProcessStream object in the Object Browser and confirm that it is of type RealFlexVariable.

RealFlexVariables have methods called SetValues and GetValues, which are analogous to those for RealVariables except that the value passed or returned is a Variant. This is a special kind of VB variable that can contain any kind of data. In this case it contains an array of data.



Note that the CanModify property of the RealFlexVariables also returns a Variant array of Boolean (True / False) values.

23. Note that at the top of procedure VarHyArray is defined as type Variant.
24. Run the code above, when the VarHyArray = … line is reached, add a Watch for VarHyArray.

Figure 28

Watches

Expression	Value	Type	Context
VarHyArray		Variant/Double(0 to 9)	modHysysCode.HYSYSExcelLinkCode
VarHyArray(0)	1.52695858778962E-02	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(1)	2.04769365142728E-03	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(2)	0.934815194771531	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(3)	4.62891782626868E-02	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(4)	1.51597009641312E-03	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(5)	4.41589819330258E-05	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(6)	1.64791293136072E-05	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(7)	1.32210429195148E-06	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(8)	3.99732885168376E-07	Double	modHysysCode.HYSYSExcelLinkCode
VarHyArray(9)	1.7391622470792E-08	Double	modHysysCode.HYSYSExcelLinkCode



What is the value in this array at the position held in the IntCH4Idx variable? Does this agree with the Methane mole fraction displayed in HYSYS?

Finally all the component mass fractions in the Product LPG stream are to be reported.

Figure 29

```
'Report Product LPG Mass Fractions-----------------------------------------

'Get a variant containing the cpt names
VarHyArray = hyCpts.Names

'Write them
For IntCount = 0 To hyCpts.Count - 1
    Range("CptData").Offset(IntCount, 0).Value = VarHyArray(IntCount)
Next IntCount

'Get a variant containing the mole fraction data
VarHyArray = hyCase.Flowsheet.MaterialStreams.Item("10").ComponentMassFractionValue

'Note that this time the .ComponentMassFractionValue property is used rather than
'using the .ComponentMassFraction.GetValues() method
'All HYSYS properties also have  ....Value or ....Values properties
'(eg Temperature and TemperatureValue and Pressure and PressureValue)
'In all cases ...Value or ...Values returns a number in HYSYS internal units

'Write them
For IntCount = 0 To hyCpts.Count - 1
    Range("CptData").Offset(IntCount, 1).Value = VarHyArray(IntCount)
Next IntCount
```

25. Execute this section of the code.

Here the VarHyArray variant array is first filled with the names of all the components and then the component mass fractions, and each are written into the spreadsheet.

Note that instead of using ComponentMassFraction.GetValues() as above, this time the ComponentMassFractionValue property of the ProcessStream object is used.

26. Examine the ProcessStream object in the Object Browser. Note that each of the properties also has a corresponding …Value property. (e.g. Temperature and TemperatureValue or Pressure and PressureValue)

What are the types of the …Value properties?





RealVariables and RealFlexVariables also have properties called Value and Values, which return numbers in HYSYS internal units.

In each case the …Value property simply returns a value of type Double (in the case of single valued properties like temperature), or Variant (in the case of component properties), in the HYSYS internal calculation units.

Rather than use the...Value properties of the ProcessStream object, it is generally better practice to use the.Value(s) property of the Real(Flex)Variable.

The final lines of the main code, clear all the object variables and then instruct VBA to skip the error handler if no error has occurred.

Figure 30

```
'Clear Object Variables
'This is good programming practice
Set hyApp = Nothing
Set hyCase = Nothing
Set hySbFS = Nothing
Set hyStream = Nothing
Set hyExpander = Nothing
Set hyCpts = Nothing
Set hyCpt = Nothing
Set hyComp = Nothing
Set hyCool = Nothing

'End of normal procedure
Exit Sub
```

HYSYS User Variables

Introduction

User Variables can be used to add to the internal functionality of HYSYS objects, such as streams and unit operations, by attaching variables and code to those objects from within HYSYS itself. User Variables can be used like the variables built in to HYSYS objects; so can be added to spreadsheets, targeted by logic controllers, have their values specified by user input, etc.

Typical uses for User Variables are:

- Calculation of Custom Properties:
- Calculation of a dew point temperature.
- Automation of actions:
- Automatically adding an energy stream each time a pump is added.
- Adding extra intelligence to the HYSYS model:
- Relating the pressure drop through a heater to the flow rate.

The first two examples can be found in Section 5.7 of the Customization Guide. In this exercise a User Variable will be implemented that relates the heater pressure drop to the flow rate in one of the exchangers in the Turbo Expander Case.

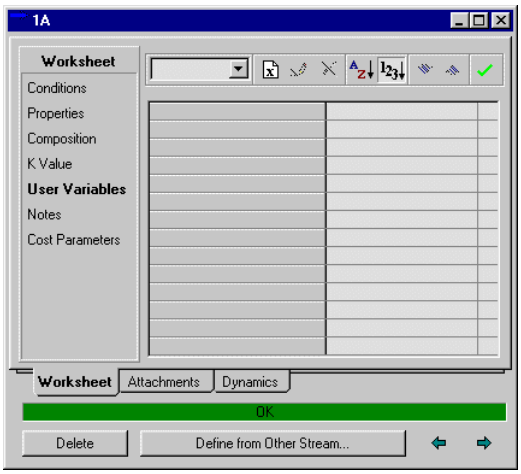
Adding a User Variable

The location of the User Variable information within the HYSYS property windows, depends on the object they are being added to.

Object	Location
Operations (except Logicals)	**Design** tab... **User Variables** page
Streams	**Worksheet** tab... **User Variables** page
Logical Operations	**User Variables** tab
Flowsheet	**Flowsheet** menu... **Flowsheet User Variables** option
Simulation Case	**Simulation** menu... **Simulation Case User Variables** option

When any of these locations is opened the view will be similar. (Below is the view for a stream).

Figure 31

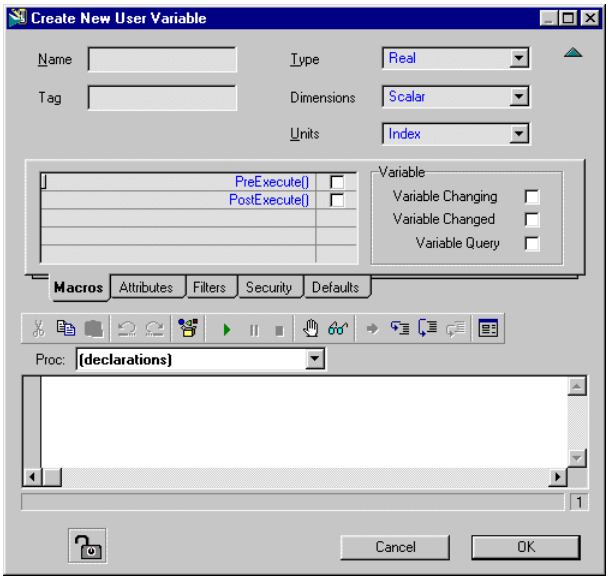




Create New User Variable icon

To add a new User Variable, click the Create New User Variable icon. The Create New User Variable dialog box then appears.

Figure 32

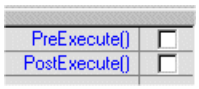
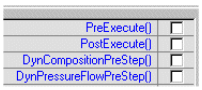
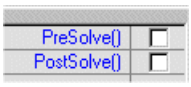
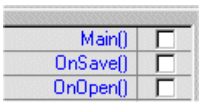




The only User Variables that can run in Dynamics are the DynComp... and DynPres... User Variables for operations.

These are called before each composition / pressure flow spec respectively.

This window is the same for all types of user variables. The only difference is the available macro types.

Stream	Operation	Flowsheet	Simulation Case
			



The only way to close the window and save any changes made is to click the OK button.

This choice sets when the code gets called. For Streams, Operations and Flowsheets the choices are before and after the object does it's calculations. Simulation Case user variables are only ever executed when the user clicks the button on the User Variable window.

Before the User Variable code window can be closed (by clicking OK), a name must be set for the User Variable.



Show / Hide Variable Details icon



Edit the Selected User Variable icon

The Show / Hide Variable Details icon (green triangle) in the top right corner of this window is used to toggle the display of the User Variable details tabs.

In order to edit an existing User Variable, click the Edit the Selected User Variable icon on the User Variable page of the HYSYS object, or double-click the user variable value cell.

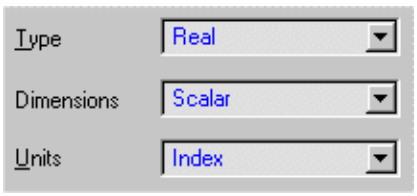
Important User Variable Parameters

Two of the most important parameters are:

1. User Variable Type

Changing the combobox values at the top-right allows the user to choose the type of User Variable.

Figure 33



- Numeric (Real), Text, Code Only...
- Single Value (Scalar), or vectors, matrices or cubes
- Units of any numeric values (Temperature, Pressure...)

2. User Variable Activation

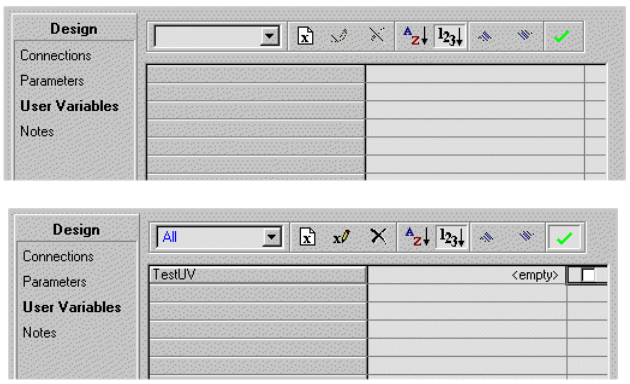
Setting the Activation parameters on the Attributes tab, tells HYSYS on which objects to enable the User Variable. (Automatic = Enabled on all objects of the same type, User Enabled = Allow the user to pick on which objects the User Variable should be enabled.)

Figure 34



If the User Variable doesn't appear on the User Variables page of an object on which it is to be enabled it may be necessary to ensure the Show / Hide Variable Enabling Checkboxes button (the green tick) is selected.

Figure 35



A more comprehensive description of the User Variable code window can be found in Section 5 of the HYSYS Customization Guide.

Writing User Variable Code

The HYSYS VB code editing window offers most of the same functionality of the VBA editor found in Microsoft Excel.

Breakpoints can be added by clicking in the left margin or clicking the Toggle Break toolbar icon. When running in break mode, Watches can be set on particular variables.

There is also a built in Object Browser. Since the code is executed from HYSYS there is no need to make a reference to the HYSYS type library - this is already set internally.

The code that is written in the User Variable uses almost exactly the same VBA syntax as when accessing HYSYS from Excel. The only difference is how to connect into the HYSYS object hierarchy, and how to interact with the displayed user variable value.

Accessing Top Level Objects

Two built-in objects allow entry into the object hierarchy

1. ActiveObject
   - This returns on object for the owner of the code. The type of this object depends on what kind of User Variable is using it:
   - Stream User Variable = Returns a ProcessStream type object for the stream containing the user variable.
   - Operation User Variable = Returns an object for the operation containing the user variable. (Type of object depends on the type of operation.)
   - Flowsheet User Variables = Flowsheet object for owner flowsheet.
   - Simulation Case User Variables = Not supported - use ActiveCase object instead.
2. ActiveCase - Always returns a SimulationCase object for the case containing the User Variable.

Interacting with the User Variable Value

Using the code **`ActiveVariableWrapper.Variable`** will return an object for the User Variable. The type of object depends on the user variable type.

User Variable... Type	... Dimensions	Object Type
Real	Scalar	RealVariable
	Vector, Matrix, Cube	RealFlexVariable
Enumeration	Scalar	RealVariable
	Vector	RealFlexVariable
Text	Scalar	TextVariable
	Vector	TextFlexVariable
Code Only	No variable available	

Hence the methods GetValue() or SetValue() can then be used as in the Excel - HYSYS example.



User Variables are saved into files with .huv extensions.

Importing/Exporting User Variables

You may import and export User Variables between cases via the **Import and Export User Variables** window. (Access this by going to the **Simulation ... Import and Export User Variables** option.)

Figure 36



Exporting a User Variable

1. Open the **Import and Export User Variables** view.

A list of User Variables currently attached to the case is displayed in the **User Variables in Case** group. The list box on the right displays a list of variables attached to the object (stream, operation, flowsheet, or simulation case) selected in the list box on the left.

2. Select the User Variable to export then click the **Export** button.
3. On the file dialogue that appears set the file name of the required User Variable export file (This will have a .huv extensions).

Importing a User Variable

1. Open the **Import and Export User Variables** view.
2. Click the **Select File** button, then navigate to the location of the .huv file.
3. Select the variable you wish to import and click the **Import** button.



Don't worry if you haven't built the Turbo Expander plant case.

The file "ADV5_Spreads&CaseStud_Soln.hsc" contains this case.

Exercise - User Variables

A simple User Variable that relates the pressure drop on a Cooler to the mass flow rate will be demonstrated.

Again, rather than typing a large amount of code into the User Variable code window, in this Workshop you will review use some prewritten code.

1. Open the Turbo Expander HYSYS case.
2. Add a User Variable to Cooler E-101 (the Recompressor after cooler) - See page 28 for guidance on how to do this.
3. Set the following User Variable parameters:

Parameter	Value
Name **(Tag will automatically be set to the same)**	Pressure Drop Calc
Type	Real
Dimensions	Scalar
Units	Pressure Drop
Execution (Macro tab)	PreExecute()
Activation (Attributes tab)	User Enabled
Variable... Calculate Only (Security tab)	Checked

4. Either type the code below into the code window, or paste it from the supplied text file. ("**Adv Automation - UV Code.txt**")

Figure 37

```
'Require Explicit Variable declaration
Option Explicit

Sub PreExecute()

    'Description:  HYSYS Advanced Course, Automation Introduction
    '              Illustrate a simple User Variable that links the pressure drop on
    '              a Cooler, to the mass flow rate

    'Declare Variables ---------------------------------------------------------

    'Variable to link to the Heate Container operation
    Dim hyCooler As CoolerOp
    Dim hyFeedStrm As ProcessStream

    Dim DblFeedFlow As Double
    Dim DblCalcPD As Double

    'Define Constants ----------------------------------------------------------
    'It is convention to give constants names in ALL CAPITALS

    Const MINFLOW As Integer=0       'Min flowrate (kg/h)
    Const MAXFLOW As Long=100000     'Max flowrate (kg/h)
    Const MINPD As Integer=0         'Min pressure drop (kPa)
    Const MAXPD As Integer=50        'Max pressure drop (kPa)

    'MAXFLOW is too big for the Integer data type hence use Long (Integer)

    'Procedure ------------------------------------------------------------------

    'Setup an Error Handler
    On Error GoTo ErrorHandler

        'Clear any value that is currently in the User Variable
        ActiveVariableWrapper.Variable.Erase
        'Here ActiveVariableWrapper.Variable gives a RealVariable object for the User Var

        'Set up object variables for the Cooler container op and it's feed stream
        Set hyCooler = ActiveObject
        Set hyFeedStrm = hyCooler.FeedStream

        'Is the mass flow rate known?
        If hyFeedStrm.MassFlow.IsKnown=True Then
            'Here hyFeedStrm.MassFlow is an InternalRealVariable

            'Flow is known - get the value in kg/h using the GetValue() method
            DblFeedFlow = hyFeedStrm.MassFlow.GetValue("kg/h")

            'Interpolate between data points to determine pressure drop
            DblCalcPD=(MaxPD-MinPD)/(MaxFlow-MinFlow)*DblFeedFlow

            'Set this value into the Cooler delta P
            hyCooler.PressureDrop.SetValue DblCalcPD,"kPa"
```

```
            'and the User Variable value
            ActiveVariableWrapper.Variable.SetValue DblCalcPD, "kPa"

        End If

        'Clear object variables - Good Practice
        Set hyCooler=Nothing
        Set hyFeedStrm=Nothing

    'End of normal procedure
    Exit Sub

ErrorHandler:

End Sub
```

5. Place a breakpoint on the Sub PreExecute() line, then make the code run by changing the flow rate of the Feed Gas stream.
6. Step through the code and ensure that it is behaving as expected.



It's best to trigger the User Variable for debugging by changing a value in the flowsheet, and hence making HYSYS call the code.

If the code is triggered by clicking the Start / Resume toolbar button then any ActiveObject references will not point to the correct object.

Note that the first time the code is called after the change is made, the mass flow through the cooler is not known hence the code in the If hyFeedStrm.MassFlow.IsKnown=True Then … End If section is not executed.

This is because when the solver performs steady state calculations there are two solve passes which it performs: the forget pass and the calculate pass. When the value of a variable changes, the solver first does one solve pass with the value marked as unknown. This is the forget pass. This allows HYSYS to correctly propagate the effects of any change.

7. Export the User Variable and import it into another case. (You could try one of the HYSYS sample cases, or one of the solution cases for this course.)

Challenge

Try using this code in a similar User Variable for a Heater operation or a Valve. Does it work? What modifications need to be made?

Try adapting the code to base the pressure drop on some other parameter - for example the molar flow, or the density, or the composition of a particular component.